



Capital[®]

Capital Harness XC™ v2012.1

Student Workbook

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**Chapter 1
Introduction to Capital HarnessXC**

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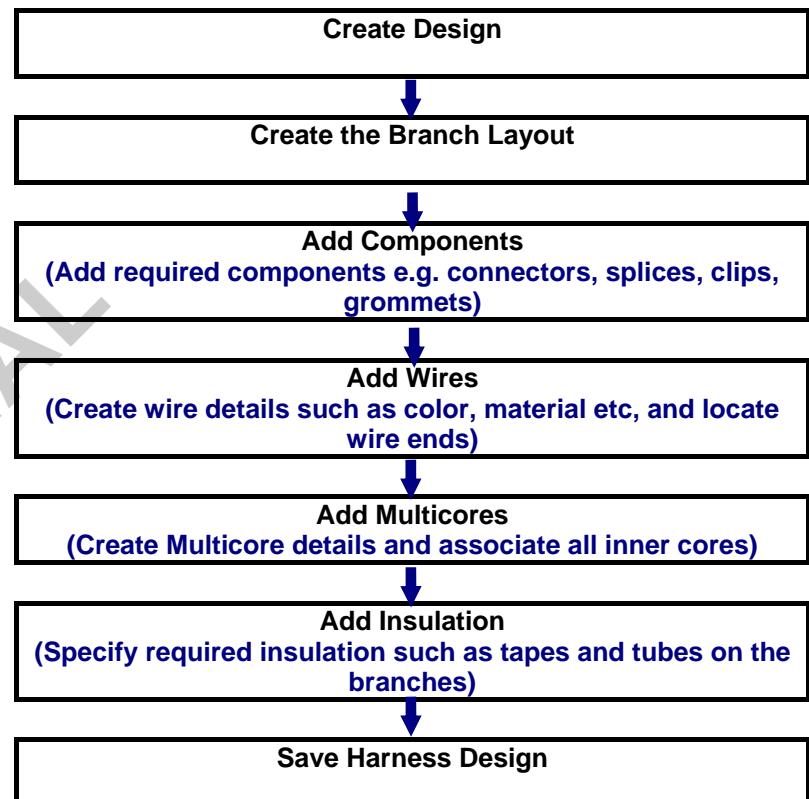
Capital HarnessXC

Capital HarnessXC is a 2D graphical design tool to enable the authoring of wiring harnesses. The tool is an integral part of the Capital suite and directly interacts with Capital Manager, Capital Library, Capital Symbol, Capital Project, Capital Logic and Capital Integrator.

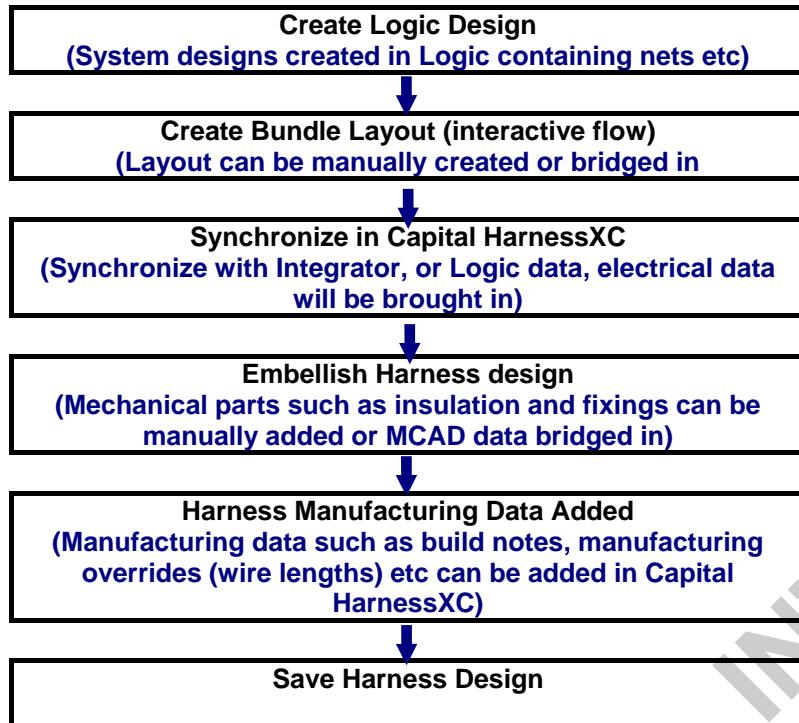
This course will explain in detail, with supporting exercises, how to create a wiring harness from scratch and also how to synchronize harness information from Capital Logic and Capital Integrator.

In order to aid understanding the other tools will be introduced at a higher level.

Process from Scratch



Process from Synchronization with Logic



FLOW

Project & User Management

Library Management

Create
Logical
Designs

Create
Topology
Designs

Create
Harness
data

Reporting

Design Analysis

CAPITAL TOOLS

Capital Project

Capital User

Capital Library

Capital Symbol

Capital Logic

Capital Integrator

*Capital Harness
XC*

*Capital Enterprise
Reporter*

Capital Analysis

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**Chapter 2
Library overview**

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Library

Overview

Capital Library is the component management tool.

Capital Library provides component management services for the Capital product suite. It is used to capture corporate conventions and notations such as color codes, standard part types and wire specifications. These may then be used to classify individual components.

- ⇒ The following concepts are available for further definition of component parts:
- ⇒ Part number management; customer and supplier part numbers can be managed alongside corporate part numbers
- ⇒ Part relationships that are used to validate component configurations; for example connector-to-seal or wire-to-terminal associations
- ⇒ User-defined properties based on component types.

Capital Library manages the part library for use by the other Capital applications i.e. data entered in Capital Library can be used to identify part numbers in Capital Logic and/or in Capital HarnessXC.

Capital Library

In its simplest form, the Library can be thought of as a *catalogue interface* that coordinates all of the information your company associates with the library parts used in its wiring designs

A part definition includes:

- ⇒ The collection of data specified for a particular part number
- ⇒ The reference to a symbol body that is used to graphically represent that part on a diagram.

The collection of data normally specified in a part's definition includes the properties, property values, and property attributes defined for that part. Each part definition (part number) requires that it be categorized according to a predefined 'Component Type' in which the component type is of a specific 'Group Name'.

Group Name and Component Type is discussed in greater detail later in this chapter.

Component Groups

Component Groups are hard-coded into the software; they allow Capital Manager to intelligently process the components, both in Capital Library (e.g. when establishing components relationships), and when they are instantiated by the applications (Capital Logic, Capital Harness etc...).

The following Component Groups exist:

Assembly	IDC Connector
Backshell	In-house Assembly
Backshell Plug	Multicore Wire
Backshell Seal	Other
Cavity Group	Solder Sleeve
Cavity Plug	Splice
Cavity Seal	Tape
Clip	Terminal
Connector	Tube
Connector Seal	Ultrasonic Weld
Device	Wire
Grommet	
Fixtures	
Heat Shrink Sleeve	

Note

Depending on which Component Group a component belongs to, Capital Library assigns different attributes for processing.

For Example:

- ⇒ If the Component Group **Connector** is used, it implies that it has cavities and can receive wires, terminals, cavity seals etc...
- ⇒ If the Component Group **Wire** is used, it implies that attributes such as the color and wire size must be defined
- ⇒ If the Component Group **Terminal** is used, it implies that wire sizes have to be allocated to the crimping ranges
- ⇒ If the Component Group **Tube** is used, it implies that it is fitted around a bundle, and therefore must have an inner bore size
- ⇒ If the Component Group **Multicore** is used, it implies that there is more than one wire grouped together in one of two ways:
 - Sheathed: the system recognizes there is an outer sheath and would need an outside diameter
 - Twisted: the system recognizes that the wires are twisted together and would need to calculate the individual wire lengths taking into consideration the twist

Codes

Codes are used within Capital Library to help distinguish between one component and another and to help readability of reports and designs. Each code is made up of a short code and a description.

They are used in component creation/maintenance and design details.

Color Codes: to represent the color of an object e.g. B=Black

Material Codes: to represent the material of an object e.g. PVC=Poly Vinyl Chloride

Component Type Codes: to sub-divide a component group in order to provide more details. For example specifying if a terminal is male, female or a ring eyelet e.g. TERF=Female terminal

Wire Sizes: to maintain fundamental relationships between:

Your company's specifications for wire size

Wire cross-sectional areas (C.S.A.) as usually specified by customers

The wire insulation material

Wire Insulation Thickness Records: to maintain the relationships between:

The (radial) thickness of a wire's insulation material

The corresponding minimum and maximum copper cross-sectional areas (C.S.A.), for each material

Note

To create a component the following codes MUST be set before hand:

Color codes

Material Codes

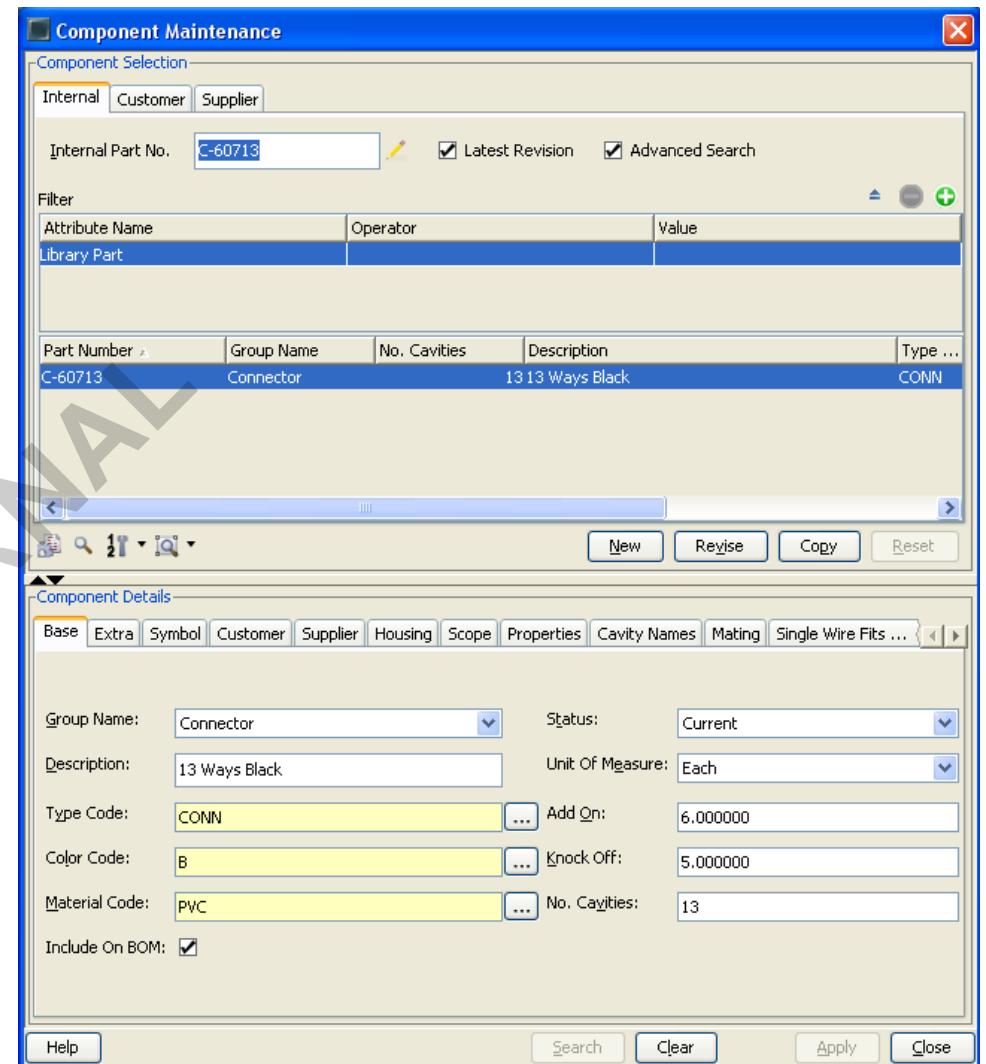
Component Type Codes

Wires Sizes must be defined for creating wires.

Exercise 1:

Your trainer will show how to create simple codes

Component Maintenance



The Component Maintenance form is used to register new Internal Component Part Numbers onto the database. Before any attributes can be assigned to a component it must first have an Internal Component Part Number.

This form is also used to register certain mandatory 'base' information about all components. As well as the part number, this includes determining the component group, type code, color, material, and unit of measurement.

Connectors will have the number of cavities defined, and terminals the amount of insulation that must be stripped off a wire before it can be crimped.

Components can be set to '**New**' if they have never been used before, or have not been approved to be used, '**Current**' if they are in current use (these can be set to be modified only by users with the assigned authority), or '**Obsolete**' if the component is not used anywhere and can be removed from the system.

Note

Depending upon the component group, on clicking Apply various tabs will become active, giving access to different information. Most forms can be accessed using these tabs but some must be entered using the main menu.

Searching/Reporting Components

When associating a Library part to an object you will be presented with a read-only version of the Component Maintenance form to search for your part.

The different ways to search are as follows:

⇒ If you wish to view all of the base details records in the database, leave the part number boxes empty and press Search.

Entering a single letter into the part number box, will display only those component records that begin with that letter

For example, typing the character **C** into the box will list only those records that begin with C regardless of their length

% (percentage symbol) represents any number of characters.

For example, typing the characters **%Z%1** into the part number field, will display those records that have a Z in any position, followed at any position, by a 1, e.g. CZL1, ZAL1, ABCDZAAAA1

_ (underscore character) represents any single character.

For example, typing the characters **___X** into the part number field, will display only those records that are 4 characters long and have an X as their fourth character e.g. ABCX, 111X, 222X, A23X

Part Association

Library allows you to associate parts and other attributes specific to the type of component. The following section is going to introduce you to the associations you should be familiar with.

Customer Part Numbers

The screenshot shows the 'Customers' dialog box. At the top, there's a filter section with columns for 'Attribute Name', 'Operator', and 'Value'. Below this is a tree view under 'Customer' with nodes for 'Address1', 'Address2', and 'Address3', each with a 'starts with' operator and a value field. A list of customers is displayed below, showing columns for 'Name', 'Internal Ref', 'Contact', 'Address1', and 'Address2'. One entry is selected: 'European Motors' with Internal Ref 'EM-717' and Contact 'S. Harrison'. At the bottom, there are buttons for 'New', 'Reset', and 'Delete'.

Address & Contact Details

Name:	Air Jet	Country:	UK
Address1:		Internal Ref:	AJ-1
Address2:		Contact:	R.Smith
Address3:		Telephone Number:	
Address4:		Fax Number:	
Post Code:			

Customer Manufacturing Options

Length Addon Per Junction:	5.000000	Wire Length Minimum Addon:	0.000000
Wire Length Maximum Addon:	6.000000	Wire Length Rounding Value:	0.000000
Wire Length % Addon:	0.00	<input type="checkbox"/> Use Pitch Table	
Datum:	Back	Rounding Method:	Default

Buttons at the bottom include 'Help', 'Apply', and 'Close'.

This form allows the internal part number to be cross-referenced with any Customer, or Customers that use that component. The customer name is recorded against the internal part number, along with the part number used to refer to that component.

Every customer that uses a particular component on their harnesses can refer to that part with their own individual Customer Part Numbers. There is no limit to the number of customers that can be registered as using a particular component.

Note

Customers and Suppliers must be registered in the 'Companies' section of Capital Library before they can be referenced in these forms.

Supplier Part Numbers

The screenshot shows the 'Suppliers' dialog box. It has a similar structure to the 'Customers' box, with a filter section and a tree view under 'Supplier' for 'Address1', 'Address2', and 'Address3' with 'starts with' operators. A list of suppliers is shown below, with columns for 'Name', 'Internal Ref', 'Contact', and 'Address'. One supplier is selected: 'A.A.G. Group Ltd.' with Internal Ref 'MIL' and Contact 'Gerrard Brothers'. At the bottom, there are buttons for 'New', 'Reset', and 'Delete'.

Address & Contact Details

Name:	A.A.G. Group Ltd.	Country:	
Address1:		Internal Ref:	
Address2:		Contact:	
Address3:		Telephone Number:	
Address4:		Fax Number:	
Post Code:			

Supplier Manufacturing Options

Length Addon Per Junction:	5.000000	Wire Length Minimum Addon:	0.000000
Wire Length Maximum Addon:	6.000000	Wire Length Rounding Value:	0.000000
Wire Length % Addon:	0.00	<input type="checkbox"/> Use Pitch Table	
Datum:	Back	Rounding Method:	Default

Buttons at the bottom include 'Help', 'Apply', and 'Close'.

The above form allows the internal part number to be cross-referenced with any Supplier or Suppliers that provide that component. The Supplier name is recorded against the internal part number, along with the name that they use to refer to that component.

Suppliers that sell a particular component can refer to that part with their own individual Supplier Part Number. There is no limit to the number of suppliers that can be registered as providing a particular component. However if more than one supplier is listed for supplying the same component, the Preferred Supplier box can be checked to state preference.

Wire Attributes

The screenshot shows the 'Component Maintenance' dialog box. In the 'Component Selection' section, the 'Internal' tab is selected, showing 'Internal Part No.' SW-51085. Below it, there's a table for 'Library Part' with columns for Attribute Name, Operator, and Value. A single row is visible: 'Library Part'. In the 'Component Details' section, the 'Base' tab is selected. It contains fields for Group Name (Wire), Description (WIRE), Type Code (WIRE), Material Code (TWC), and Specification (28/.3). Other tabs include Extra, Customer, Supplier, Properties, History, and Scope. Buttons at the bottom include Help, Search, Clear, Apply, and Close.

This form is used to detail key wire characteristics, e.g. specification, color and optionally the outer diameter of wires.

Wire colors are made up of the color codes used by your organization. If the wire has more than one color then the color codes are separated by delimiters.

The following are allowable delimiters along with a **suggested** use for the delimiters:

/	To indicate that the color after this symbol is a stripe
\$	To indicate that the color after this symbol is a spiral stripe
#	To indicate that the wire number is stamped along the insulation
#ABC	To indicate that the text ABC (or whatever replaces ABC) is stamped along the insulation

Multicore Wire Attributes

The 'Sheath Type' drop down box defines whether the multicore is sheathed or a twisted pair.

The 'Outside Diameter' field is an alternative way to define the outer sheath thickness.

Capital Library allows an unlimited number of single wire Innercores as well as the creation of multicores within multicores. This means it is possible to enter the attributes of a sheathed multicore that contains sheathed multicores or twisted wires as innercores. Setting up multicores within multicores is dealt with later in the training course.

Connector Details

This form allows users to add extra information to the connector, for example by selecting the Mating tab users can insert information for the mating half of the connector. The Housing tab allows users to assign sub components to the connector. Tab buttons can be added or removed using the configuration gui.

Ring Terminals

Component Maintenance

Component Selection

Internal Part No. RT1 Latest Revision Advanced Search

Filter

Attribute Name	Operator	Value
Library Part		

Part Number	Group Name	Description	Type ...	Color ...	Materi...
RT1	Terminal	Ring Terminal	RT	-	-

New Revise Copy Reset

Component Details

Base Extra Symbol Customer Supplier Housing Scope Properties Cavity Names Single Terminations Mult...

Group Name:	Terminal	Status:	New
Description:	Ring Terminal	Unit Of Measure:	Each
Type Code:	RT	Strip Length:	0.000000
Color Code:	-	Mult Strip Length:	0.000000
Material Code:	-	Add On:	0.000000
Include On BOM:	<input checked="" type="checkbox"/>	Knock Off:	0.000000
		No. Cavities:	1

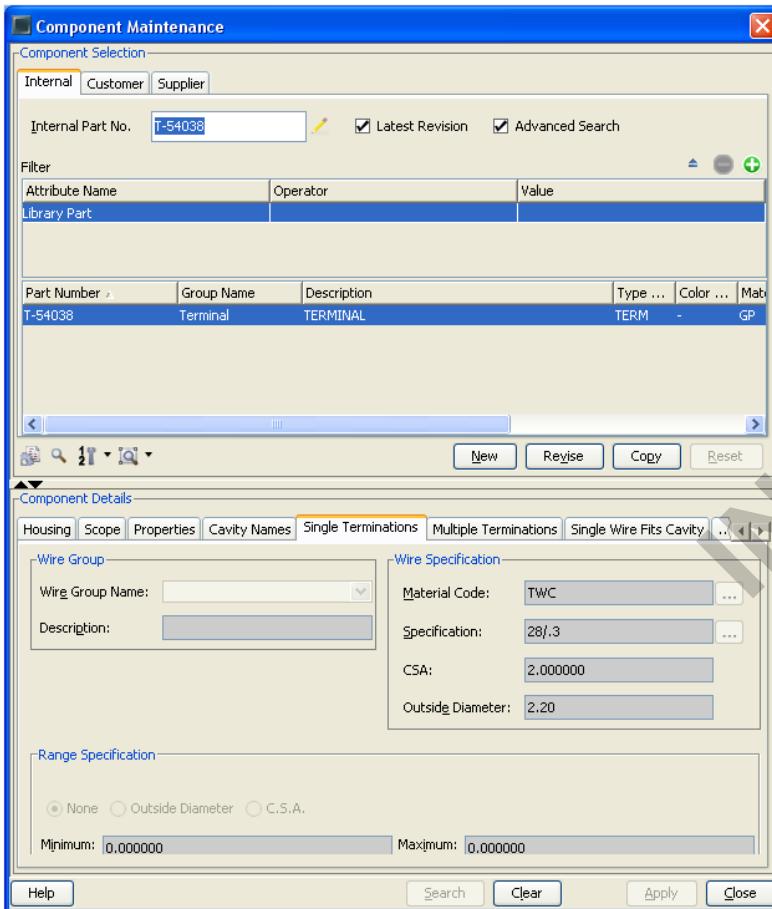
Help Search Clear Apply Close

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This form enables users to define a terminal as a ring terminal, allowing users to enter a value for the number of cavities (typically 1) for the terminal, as well as the add-on or knock off value for the wire.

Single Wire Termination Records

This form allows the user to determine which wire specifications will fit a cavity seal or can be crimped in a terminal. First the Material of the wire must be entered. It is then possible to either select and apply a single wire specification, or alternatively, choose to automatically select several specifications by entering a Cross Sectional Area range.



Multiple Wire Termination Records

As well as needing to specify the individual wires that fit Terminals and Seals, it is also necessary to register any combinations of multiple wires. This is done within the Multiple Wires Fits Cavity tab.

Note

It is possible to register any number of wire combinations, and every valid combination of wires must be registered.

For example if a terminal is only registered to accept a combination of two 16/.2 wires and one 32/.2, this does not mean that it can also crimp a combination of just two 16/.2 wires.

Splice Attributes

Component Details

Base	Extra	Symbol	Customer	Supplier	Housing	Scope	Properties	Splice Attributes	Revisions	Cavity Names
Min. Total C.S.A:	3.100000	Min. No. Wires(Total):	2							
Max. Total C.S.A:	6.000000	Max. No. Wires(Total):	15							
Min. Single Wire C.S.A:	0.350000	Max. No. Wires(Per Side):	0							
Max. Single Wire C.S.A:	2.000000			<input checked="" type="checkbox"/> Auto Selectable						
L/R C.S.A Ratio:	0.000000									

Help Search Clear Apply Close

When entering a Splice onto a harness, it is possible that the splice part number that should be used will depend on the formation of the wires within the splice. The splice attributes tab allows the user to specify certain parameters that will allow Harness XC to automatically select the Splice part number.

Min / Max CSA:

This is the acceptable total Cross Sectional Area of all the wires in the splice.

⇒ Min Wire / Max Wire CSA:

This is the acceptable Cross Sectional Area of individual wires in the splice.

⇒ Min / Max Number of Wires:

This is the acceptable number of wires within the splice.

⇒ L/R CSA Ratio:

This defines the balance of wires on the left hand side compared to the right hand side of the splice

Heat Shrink Sleeve Parameters

When placing a Heat Shrink Sleeve over a splice, it is possible that the Heat Shrink part number to be used will depend on the formation of the wires within the splice. The heat shrink sleeve tab allows the user to specify certain parameters that will allow Capital Harness to automatically select the Heat Shrink part number.

Tape Attributes

The screenshot shows the 'Component Selection' tab of the 'Component Maintenance' dialog. The internal part number 'TA-57743' is entered in the search field. The 'Latest Revision' and 'Advanced Search' checkboxes are checked. The 'Filter' section shows a table with columns 'Attribute Name', 'Operator', and 'Value'. The 'Library Part' row is selected. Below this is a table showing component details: Part Number TA-57743, Group Name Tape, Description TAPE 25mm BLACK, Type Code TAPE, Color Code B, Material Code PVC. At the bottom are buttons for New, Revise, Copy, and Reset.

Component Details

Base				Extra				Customer				Supplier				Housing				Scope				Properties				Spot Tape Selection				History			
Group Name:	Tape	Status:	Current	Description:	TAPE 25mm BLACK	Unit Of Measure:	Per Roll/Reel	Type Code:	TAPE	Color Code:	B	Material Code:	PVC	Include On BOM:	<input checked="" type="checkbox"/>	Wall Thickness:	0.200000	Tape Width:	25.000000																

Help Search Clear Apply Close

Attributes for tape are defined within the base details tab.

Tube Attributes

The screenshot shows the 'Component Selection' tab of the 'Component Maintenance' dialog. The internal part number 'TU-45388' is entered in the search field. The 'Latest Revision' and 'Advanced Search' checkboxes are checked. The 'Filter' section shows a table with columns 'Attribute Name', 'Operator', and 'Value'. The 'Library Part' row is selected. Below this is a table showing component details: Part Number TU-45388, Group Name Tube, Description 14mmID GREY TUBE, Type Code TUBE, Color Code S, Material Code PVC. At the bottom are buttons for New, Revise, Copy, and Reset.

Component Details

Base				Extra				Customer				Supplier				Housing				Scope				Properties				History			
Group Name:	Tube	Status:	Current	Description:	14mmID GREY TUBE	Unit Of Measure:	Per Length	Type Code:	TUBE	Color Code:	S	Material Code:	PVC	Include On BOM:	<input checked="" type="checkbox"/>	Wall Thickness:	2.000000	Bore Size:	14.000000												
																				<input checked="" type="checkbox"/> Slit <input checked="" type="checkbox"/> Convoluted											

Help Search Clear Apply Close

Tube details such as bore size and wall thickness are defined in the base details tab, assuming the GUI has been configured to display these fields.

Other Components

Some components do not have specific attributes but still require the base details (Component Maintenance) such as:

- ⇒ Cavity Plugs
- ⇒ Clips
- ⇒ Grommets

Housing Definition

The housing tab allows users to state the relationship between components and sub-components.

The screenshot shows the 'Component Maintenance' dialog box with the 'Housing' tab selected. The main area displays a table of sub-component relationships:

Cavity Group	Group Name	Sub-Component	Quantity	Cavity	Position
	Cavity Seal	CS-6317	0.0		
	Other	AB-4581	1.0		
	Cavity Plug	CP-6322	0.0		
	Terminal	T-54071	0.0		

Below the table are several input fields and buttons:

- Expand Cavity Group
- Sub-Component: CS-6317
- Status: Current
- Group Name: Cavity Seal
- Usage: Optional
- Quantity: 0.000000
- Factor: 0
- First Cavity:
- Last Cavity:
- Mode: Cavity Position
- Position Blocked:
- Buttons: New, Reset, Delete, Help, Search, Clear, Apply, Close

For example:

Connectors require Terminals, Seals, Plugs, Anti-backouts etc.

Clips require Spot Tape

The Usage field allows us to state whether a sub-component is:

- ⇒ Mandatory (will always be selected with the main Component)
- ⇒ Optional (may or may not be selected with the main component)
- ⇒ Customer (the sub-component is selected if the customer of the harness has a Customer Part number for the component).

Further fields allow us to add a selection factor to a terminal, and also to enter different cavity component part numbers for different cavities (however if a terminal fits all cavities in a connector then this field should be left blank).

Note

If more than one terminal is suitable for selection at a cavity, the Factor field can be used to give priority of one terminal over the other(s). This field is alphanumeric, so 11 has a higher priority than 2.

The **Expand cavity group?** Check box will display all components held within a pre-defined cavity group.

Scope

The scope name is defined in Capital Library under the codes section. If a component is to have its use restricted, for example to a particular manufacturing site, this can be defined under the scope tab.

- ⇒ Open the scope tab
- ⇒ Select the required scope and click Add to move it to the selected area
- ⇒ Click Apply

When inserted on the harness diagram the system will highlight a conflict if the design or project has a specific scope that does not match the scope assigned to the component. If no scope is assigned to the component it is assumed that the component is fit to be used on any harness.

User Properties

Property Codes

The screenshot shows the 'Components Main Menu' with the 'Codes' section expanded. Under 'Codes', there are several items: Color Codes, Material Codes, Component Type Codes, Customer Color Codes, Wire Sizes, Wire Insulation Thickness, and Property Codes. The 'Property Codes' item is highlighted. Below this, a 'Property Codes' dialog box is open. The dialog has two main sections: 'Properties' and 'Filter'. The 'Properties' section contains a table with two rows:

Property	Description	Data Type	Pin Property
I Max	Maximum current for a device	String	<input checked="" type="checkbox"/>
milspec	milspec	String	<input type="checkbox"/>

The 'Filter' section shows a table with three columns: 'Attribute Name', 'Operator', and 'Value'. There is one entry: 'Property Code' under 'Attribute Name', 'starts with' under 'Operator', and an empty field under 'Value'.

User Property Codes allow for definition of additional property names that can be allocated to specific Component Type Codes.

The property is set up to have a specific data type value from the following options:

- ⇒ String (allowable characters are alphanumeric, dollar and dash)
- ⇒ Integer (a whole number)
- ⇒ Float (a decimal number)

Examples:

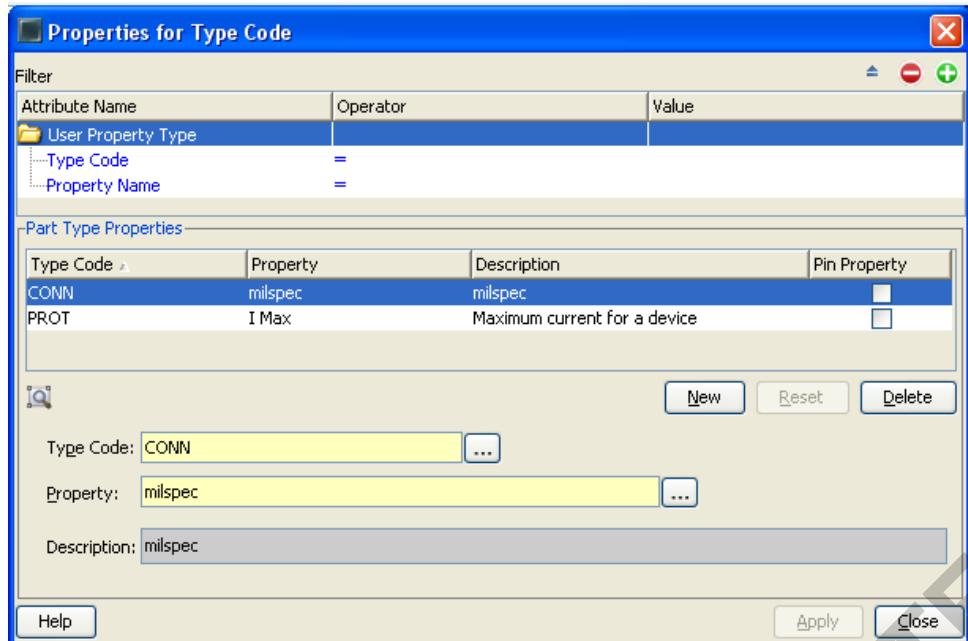
String

It may be necessary to define the environment where a component is to be used e.g. used in a dry or a wet area, therefore the data type would be string and the description 'Environment'.

Integer/Float

It may be necessary to define the resistance of a wire per meter, therefore the data type would be integer/float and the description could be Resistance/m

Properties for Type Codes



This function allows you to assign the previously defined User Property Code to any library part of the appropriate component type code.

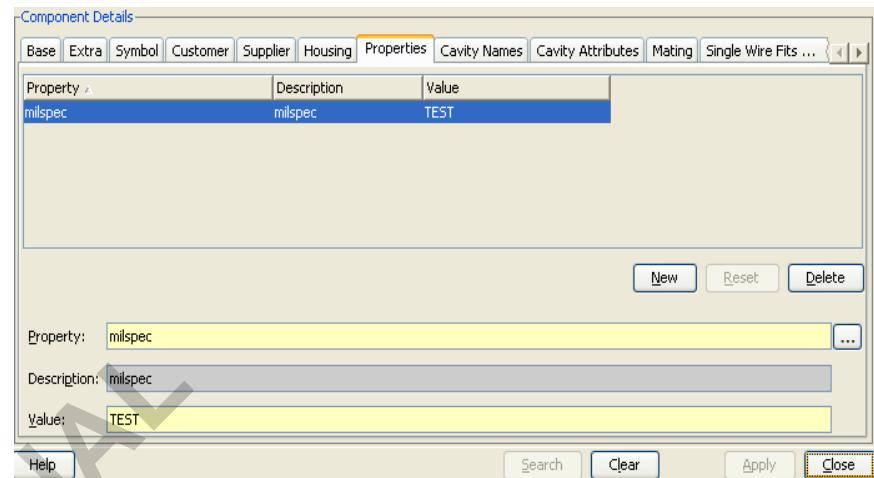
For example:

Environment property could be associated with a type code for a specific type of component.

Note

A User Property Code can be assigned to more than one type code and a type code can have several different User Properties assigned to it.

Component User Properties



This function allows users to associate the User Property to the desired components in the component maintenance form.

When the property is selected the user enters a value for that component.

Note

Once the part number has been selected, only the properties that have been set against that component type code will be available for selection.

The value must match the data type previously set.

There is no limit to how many user properties can be assigned to a component.

The method of assigning User Properties to a library part may be summarized as follows:

- ⇒ Define the User Property Name (Update User Property Codes)
- ⇒ Associate the User Property Code to Component Type Code(s) (Update Type Code User Properties)
- ⇒ Assign Property Value to specific Library Part (Component Maintenance)

Exercise 2:

Creating library components.

Create the library components as shown in exercise 2.

Component Revisions

Library components may be revisioned and attributes added or removed in cases where parts may have changed over time.

In order to make a revision of a component in the library:

- ⇒ Specify the name of the library part you wish to make a revision of
- ⇒ Click **Search** and view the details of the original part
- ⇒ In the Name field type in the name of the new revision
- ⇒ Click the **Revise** button

The new revision is now created but it will contain the attributes of the previous component. Users may change these attributes as required.

- ⇒ Click **Apply** once the changes are complete

In some cases it may be that previous revisions have been made but have not been recognized as having been derived from a previous part. It is now possible to associate previously revised parts to a new library part so that the revision history is linked.

To associate a library part as a revision to a component in the library:

- ⇒ Specify the new library part in the Name field and click Search
- ⇒ Click the **Revisions** tab
- ⇒ Click **New** and insert an **Associate part**
- ⇒ Click **Apply**

Library Domains

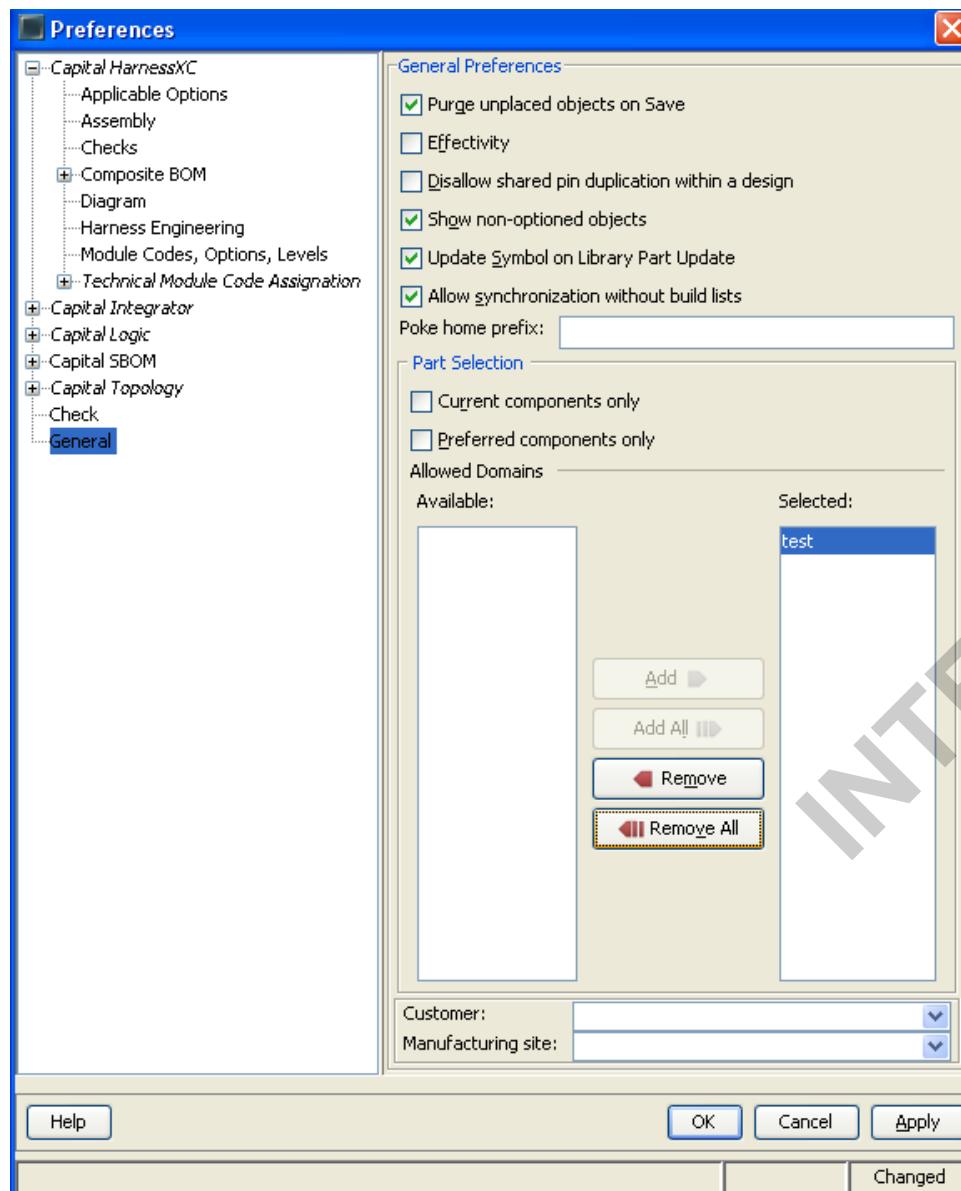
In some cases it is essential to keep library components separate, for example some components are only to be used when working on military based projects, and others may be used only on civilian based projects. In this case a domain can be created and assigned to User Accounts, individual projects and library parts.

To create a Domain:

- ⇒ Open Capital User
- ⇒ Right Click **Domains** and **Edit**
- ⇒ Type the name of a new domain
- ⇒ Click **Apply**
- ⇒ Right click the relevant user account to which the domain is to be assigned
- ⇒ Select **Read-only or Read-write domain**
- ⇒ Assign the new domain to the user account

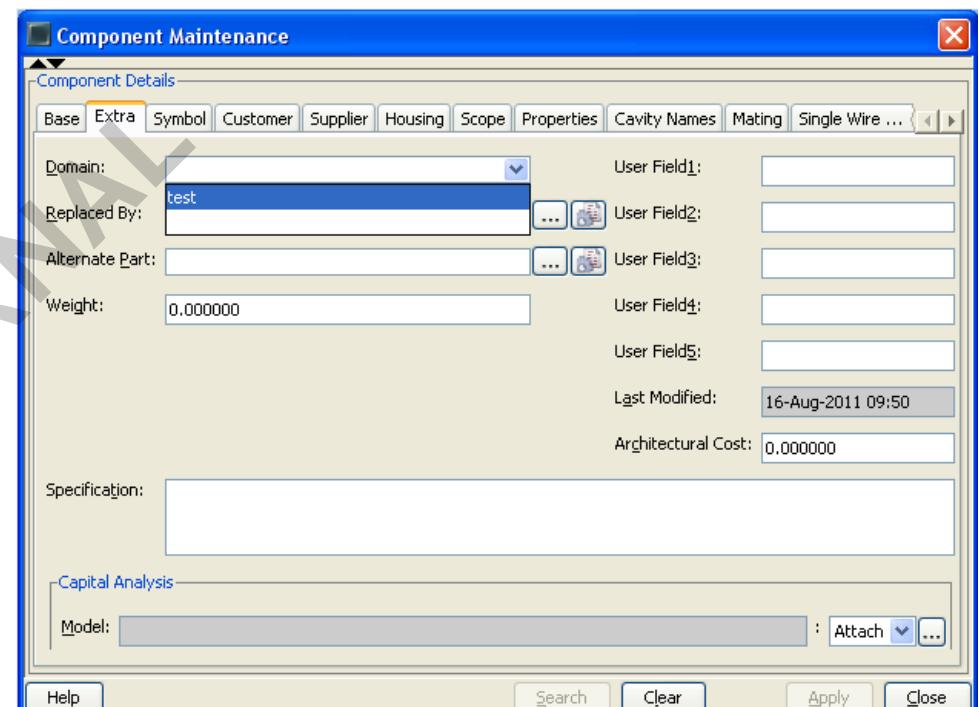
Now the domain is assigned to the user, it must also be assigned to a project to ensure that only library parts from that domain are used in the project.

- ⇒ Open Capital Project
- ⇒ Open the project to which a domain is to be assigned
- ⇒ Select **preferences**
- ⇒ Select the General folder
- ⇒ Select and assign the required library domain to the selected area



Within the Capital Library tool the part number will also have a domain assigned to it:

- ⇒ Open the Component Maintenance form
- ⇒ Select the library part you wish to assign a domain to
- ⇒ Click the **Extra** tab
- ⇒ Click the drop down button next to the **domain** field
- ⇒ Select the relevant domain
- ⇒ Click **OK**



**Capital
Capital HarnessXC
Training Course V2012.1**



Chapter 3 Introduction to Capital Project

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Capital Project Concepts

Overview

Capital Project enables administrators and designers to organize and configure projects. A project may be considered to be a multi-configuration product such as a specific aircraft type or a car vehicle model-year.

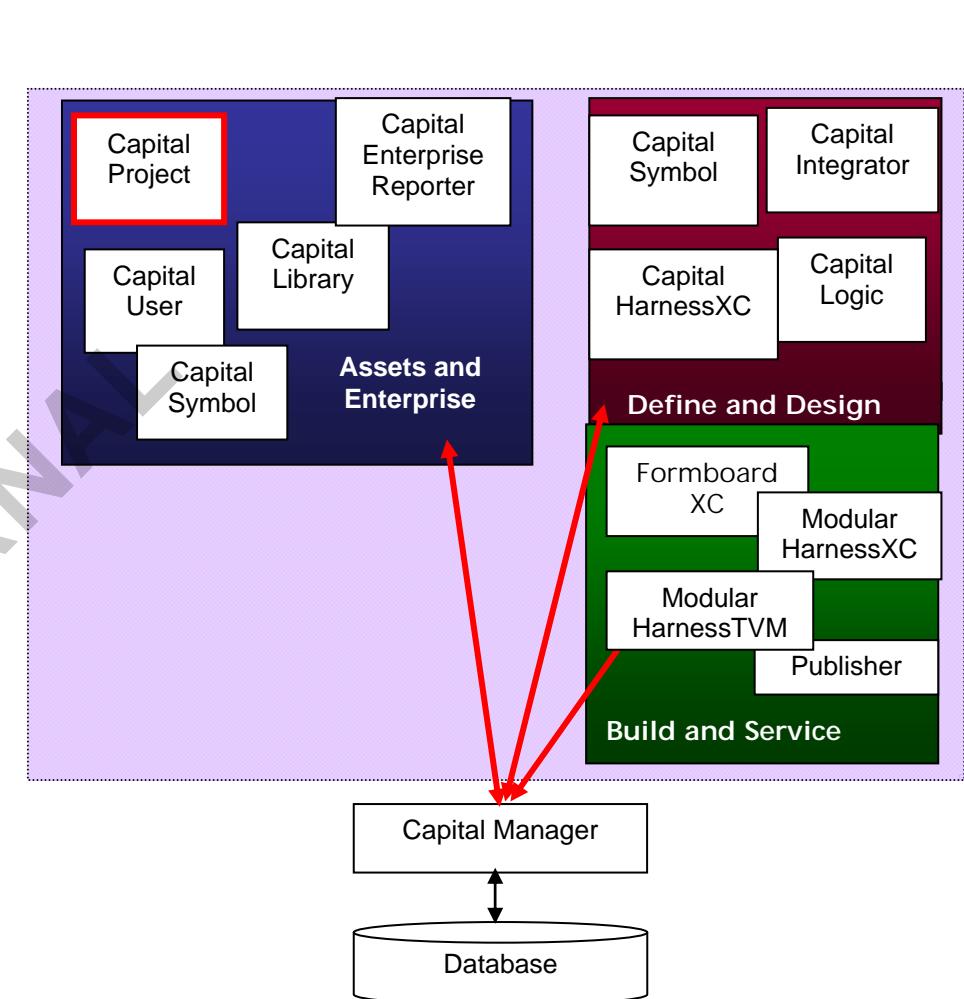
The electrical system of the project will typically be defined by

- ⇒ many designs covering functional systems
- ⇒ physical wiring implementations
- ⇒ topological architectures
- ⇒ harness definitions

A project would also normally support several configurations and multiple versions of each design.

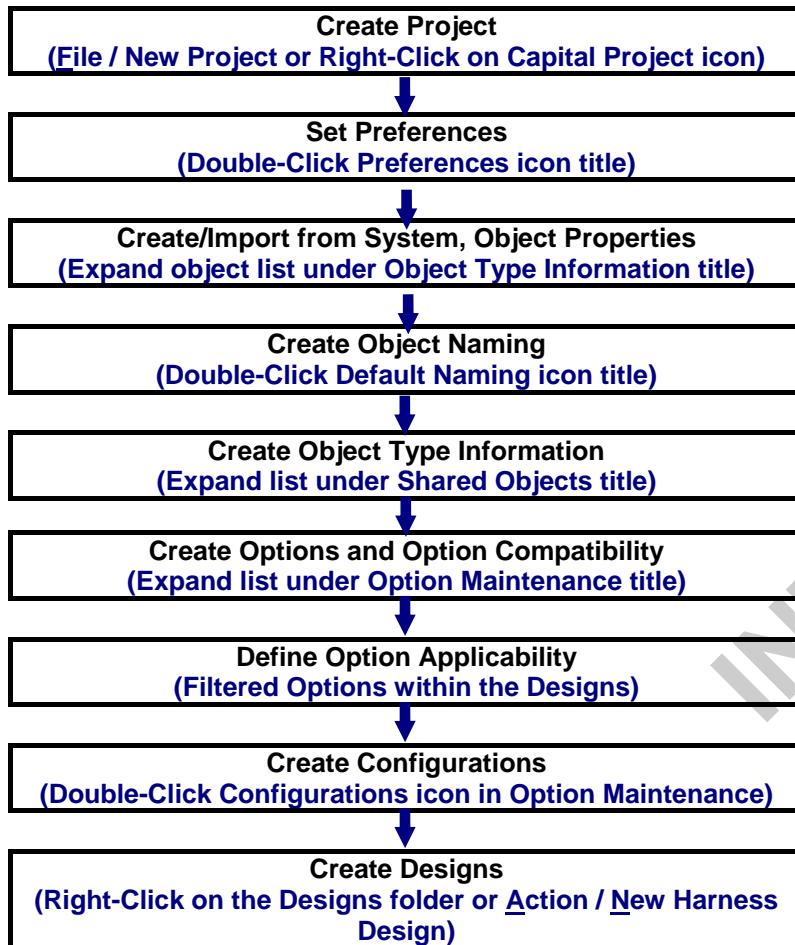
Capital Project enables projects and designs to be created, amended, copied and deleted. It enables options, release levels, preferences (e.g. naming conventions, symbol standards) and shared objects to be defined at project level and then rolled-out to the designs.

Capital Project allows administrators to define and implement maturity lifecycle management for designs ultimately locking released designs from further modification.



Process

The following process flow summarizes the activities to define a project:



Appendix 1

Details methodologies of the Capital tool set.

Getting Started with Capital Project

Create a Project

The electrical system of the project will typically be defined by many designs. Select the Capital Project item in the browser menu or execute the **File / New Project** menu command.

- ⇒ Enter the **Name** of the project
- ⇒ If desired, enter a **domain**
- ⇒ Specify a project folder if desired

The Domain is a method of restricting access to certain projects. Each user is given access to specific domains and therefore can only then access projects with that domain (or when the domain is left blank). Domains will be created later in the course.

Project Scoping

To restrict component part selection to specific customers or manufacturing sites, users may assign a scope to the project:

- ⇒ Right click the project name/Edit
- ⇒ Click Properties/Scope
- ⇒ Select the required scope and assign to the Available area

Project Preferences

System and project based preferences are used to define preferred settings for things such as audit trail requirements, location paths for images, default processing values, error reporting processes, project level design checks. Users must be assigned the permission to modify system and project based preferences.

System wide parameters can be set up and transferred manually to individual Projects by the drag and drop feature.

Note

If no project specific parameters are defined then all available System level parameters can be copied down to the project.

Project Folders

Projects may be placed into folders. This enables users to better organize the project tree

Project folders are created within the edit project information form.

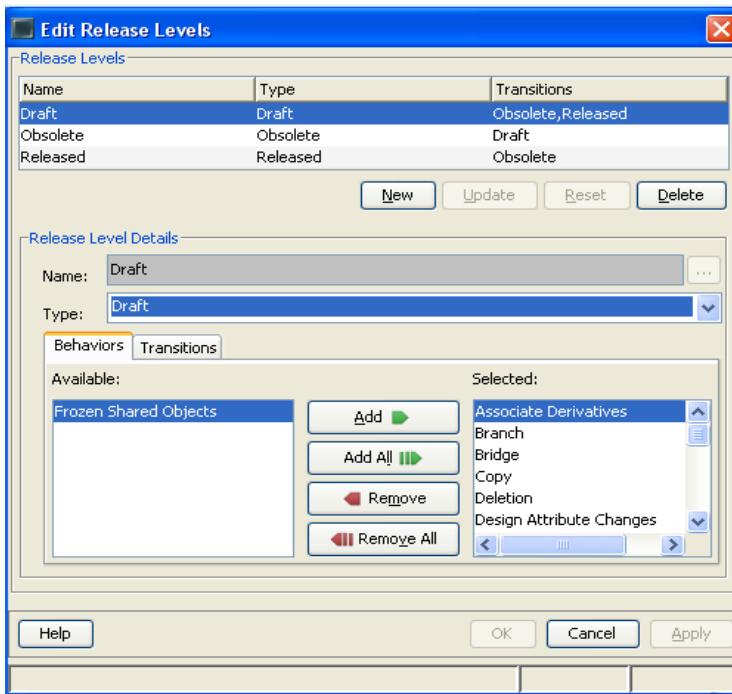
- ⇒ Click on the browser button to the right of the folder field
- ⇒ Right Click the top level project folder
- ⇒ Click New and name the new folder
- ⇒ Sub folders may be created within the new folders in the same manner

The project that is currently being edited will now appear within the defined project folder.

The folder is removed when no project exists within it. To remove a project from a folder:

- ⇒ Edit the project
- ⇒ Click the red cross next to the folder field
- ⇒ The project is now placed back in the top level project list

Define Release Levels



Release levels are a way to track the progress or design stage of a Project or design.

A Design can go through certain stages during its life cycle – these are known as Release Levels. Capital Project has some pre-defined release levels as explained below, however, only users with sufficient access rights will be able to set and change Project and design release levels (set in Capital User):

- ⇒ **Draft** – this is the initial stage of design where no or few restrictions exist, the designs can be updated, modified etc
- ⇒ **Pending** – this is the stage of the design when the final checks are being made, the designs can be updated, modified etc but NOT deleted
- ⇒ **Released** – this is the final stage of design when the designs are complete, in this case the designs can no longer be update, modified etc. They are in read-only mode
- ⇒ **Obsolete** – this implies that the design is either no longer used (or maybe incorrect)

This function allows users to customize the names of the pre-defined release levels by selecting the release level, click **New** and define a new name to associate to the release level type. The **update** button commits this information to the database.

Behaviors

Different release levels can have various behaviors assigned to them to restrict the type of operation that can be performed on the project at that particular release level, i.e. copying and deleting the project, bridging or exporting data out, or creating revisions etc. To assign Behaviors to a release level:

- ⇒ Open the Release levels edit form
- ⇒ Highlight the release level you wish to add behaviors to
- ⇒ Select the Behaviors tab
- ⇒ Remove the behaviors you do not wish the project to have access to

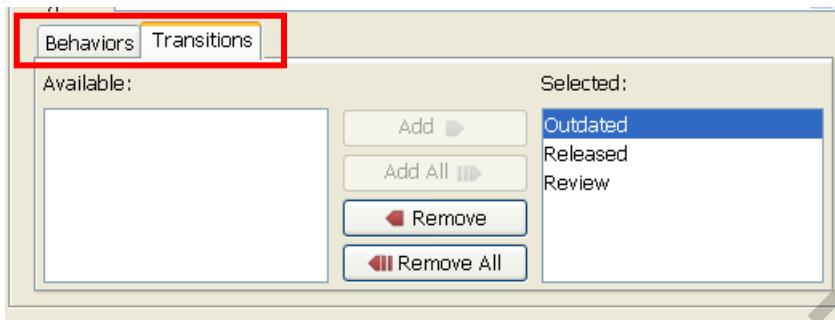
The following are the release levels for Capital HarnessXC. Those shown in **bold** are specific to Capital HarnessXC.

- ⇒ **Associate Derivatives**; ability to associate derivatives to the harness
- ⇒ Branch; ability to create a new revision of a design
- ⇒ Bridge; ability to bridge in MCAD data on that harness
- ⇒ Copy; ability to copy the harness design
- ⇒ Deletion; ability to delete the harness design
- ⇒ Design Attribute Changes; ability to modify the attributes of the design
- ⇒ Design changes; allows design edits, deletion of diagrams and bridging in using web services
- ⇒ Evaluate: ability to use the evaluate function in a design
- ⇒ Export; ability to export the design
- ⇒ **Harness Processing**; ability to run the harness processing on the harness
- ⇒ **Harness XC Connector Editing** – ability to fully edit a connector in XC
- ⇒ **Harness XC Multicore Editing** – ability to fully edit a multicore in XC
- ⇒ **Harness XC Overbraid Editing** – ability to fully edit an overbraid in XC
- ⇒ **Harness XC Splice Editing** – ability to fully edit a splice in XC
- ⇒ **Harness XC Wire Editing** – ability to fully edit a wire in XC
- ⇒ Printing; ability to print the harness
- ⇒ Revision; ability to create a revision of the harness
- ⇒ SVG Service; ability to create a view of the diagram via web services

Transitions

Users must define the transitions available for release levels. For example, a project with a release level of draft may proceed to pending, then review, then obsolete however it is only possible to do this if the release levels (pending, review and obsolete) have been created and defined in the **Transitions** tab which is found in the Edit Release Levels form. To edit transitions:

- ⇒ Highlight the release level you wish to assign the transition to
- ⇒ Select the transitions Tab
- ⇒ Select the available release levels and assign by clicking **Add**



Close Project

Closing a project removes the project and its contents from the active Capital Project session.

This action removes the project from the Browser.

Similarly Right Click (Right Mouse Button) click on a project and selecting Close will close the selected project without the close project dialog box.

(Note: All project changes are saved automatically on close with any user interaction).

All values in a given project can be assigned (by means of a drag and drop action) to the System level by a user if they have permissions.

Exercise 1:

Create a Project:

- ⇒ Using information shown in exercise 1 on the exercise worksheet, create the Project as required.

Naming Conventions

Object Type Information

Objects within the Capital application can have properties associated to them and naming conventions created for each specific project.

The names that are associated to the objects can be created in the object type information area or within the harness diagram itself when adding a connector or wire.

1. Using Object type information (properties and names are predefined)

To create a pre defined set of names for a connector:

- ⇒ Open capital project and your project
- ⇒ Navigate through the browser to the object type information folder
- ⇒ Navigate to the connector names sub folder
- ⇒ Right click and select edit
- ⇒ Create a new name within the dialog

The name is now available for use when inserting a connector on a harness diagram.

Default names are available for objects such as bundles and insulations. Default names are defined within Capital project under the default naming folder.

Note

Object type properties and names can be manually transferred from one project to another using the drag and drop facility

Object Properties

An object property consists of a name, a value type and a value constraint type, it is a way of further identifying an object e.g. color, temperature resistance, wire size etc

Name: this is the reference by which the property is selected from a drop down list

Value Type: - this can be one of 3 options:

- ⇒ **Integer** – a whole number only
- ⇒ **Float** – a decimal or whole number
- ⇒ **String** – Alphanumeric characters

Value Constraint Type: This can be one of three options:

- ⇒ **Unconstrained** – A value is applied once the property is assigned to an object
- ⇒ **List** – a list of specific values are applied to a property. When that property is assigned to an object that specific list will become available for use
- ⇒ **Range** – A numerical range can be created with an upper and lower limit, the range is applied to an object property and if a user enters a value outside of that range a warning is given

Note

The system will prevent you from entering a value that is not in the list or range.

Projects and designs may also have properties created and assigned in the same way as object properties are created.

Appendix 2

Details the process involved in creating unconstrained, list and range properties. Users will have the opportunity to view how these are used within a design later in the course

Exercise 2:

- Take a look at the pre-defined object names defined in the system variables and follow the exercise defined in the worksheet.

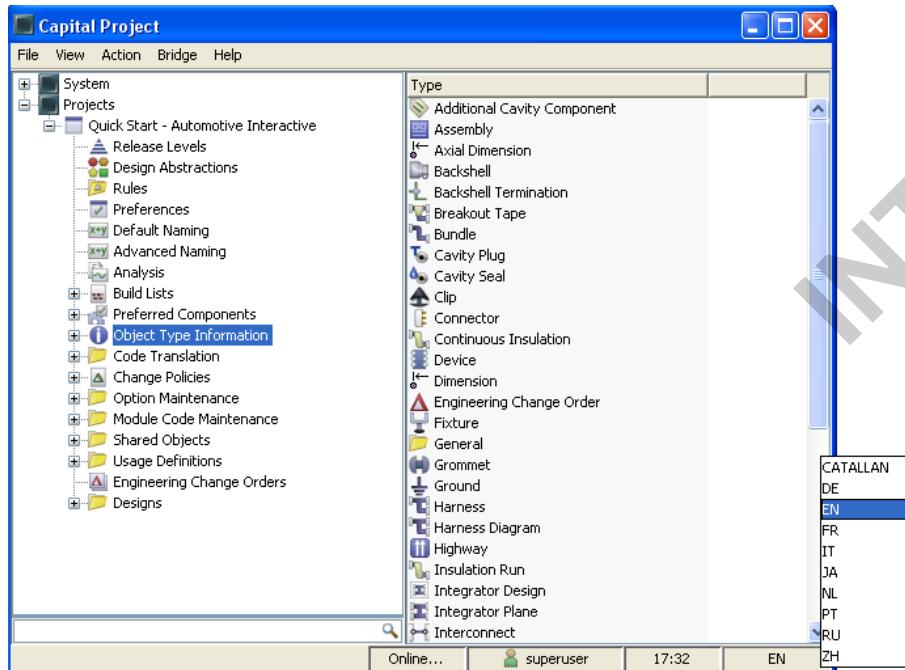
Language Translation

A language dictionary is available to help users translate design information into different languages. This is useful for the production of service documentation.

To load the language dictionary:

- ⇒ Open Capital Project
- ⇒ Select File/import language dictionary
- ⇒ Select the csv file to import
- ⇒ Choose the merge or replace radio button
- ⇒ Click Import

The bottom right hand corner of the active window indicates which language is currently active in the tool.



To change the selected language:

- ⇒ Left click the language identifier
- ⇒ Select the required language from the available list

Any design specific information (description, property, object type information) can be translated using the language dictionary. Text that can be translated is highlighted in grey: circuit diagram

When the language is switched, any changeable text will automatically read in the chosen language, assuming that language has a translation for the original text. Objects without an available translation will remain unchanged.

The language dictionary can be updated at any time (by users with the appropriate permissions) to add or remove translation information. When entering property values that do not currently exist in the language dictionary, users can manually add the new values to the dictionary:

- ⇒ Right click the property value
- ⇒ Select Add to dictionary
- ⇒ The currently selected language will be active
- ⇒ Enter the quick code to be used for this value

Each translatable term must have a quick code, this is what identifies the text as translatable.

If a translation is available for the quick code, users must enter the relevant translation in the dictionary file for each available language.

When project or design information is passed between systems, the language the data is exported in will become the default. If the string identifier and the code are the same on the target and source system, the text will be translated.

If target and source translation information does not match, the intended value (that of the exported data).

Chapter 4

Harness Diagram Creation

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Creating Designs

Create a Design

Select **File / Open Project...** or the icon  to open the project list and select the required project from the list.

Users may wish to configure the tool so that upon opening, one of 3 things occur:

- 1: The most recently viewed project is opened
- 2: The available project list is displayed
- 3: The standard window is displayed and users decide which action to take.

To create a design:

- ⇒ **Right Click** on the design folder in Capital Project or the project name in Capital HarnessXC
- ⇒ Enter the **Name** used to define the type of harness e.g. engine
- ⇒ Enter the **Part Number, Revision and Short Description**; these three fields are used to uniquely identify the individual harnesses across the system

The Domain is used to control access to the harness. The system administrator creates several domains and allocates access to the users. This domain is then set against the designs so if the user has not been allocated access they cannot modify the design.

- ⇒ Define the diagram details, a default style set is assigned automatically, this depicts the look and feel of the diagram and may be changed if users have additional styling licenses – style sets will be discussed later

Note

By selecting to seal the harness, every connector placed on the harness will automatically request to be sealed and plugged and every splice on the harness will require insulation (tape or heat shrink sleeve).

The combination of Name and Revision is unique within the project.
The combination of Part Number and Revision is unique across the system.

Modify Harness Details

To enable the processing and use of correct customer data, it is required to modify the customer details of the harness.

Design Scopes

Projects, individual components, and designs may have scopes assigned. A scope is created in the Library codes folder and is assigned to components. A design can have a scope assigned to ensure that part selection is restricted to those components with a matching scope.

To assign a scope to a design:

- ⇒ Edit the design in the browser (right click/Edit)
- ⇒ Select the properties button
- ⇒ Select the desired scope and assign it to the Available field
- ⇒ Click Apply

Exercise 1:

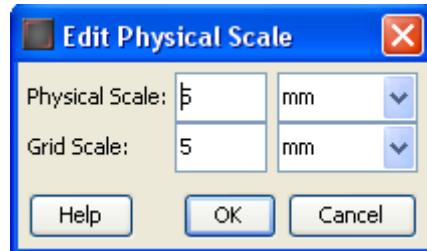
Create a Design:

Using information shown in exercise 1 on the exercise worksheet, create the design as required

Scaling Preferences

When creating a diagram the grid reference is used to help you scale your drawings. In order to set up the scaling in Capital Harness XC

- ⇒ Select **Layout / Diagram**
- ⇒ Enter a value for the physical scale



Physical Scale unit/Value – this specifies the amount by which bundle lengths will be incremented when stretched from 1 grid point to another – this is only enforced when placing a dynamic bundle. This setting may also be defined at system or project level under preferences. If changed directly in the diagram, this will override the project preferences

Grid Scale - This setting combined with the grid spacing unit will affect the size of the objects on a diagram when printed

Grid Defaults

The pin grid and drawing grid settings can be defined under **Layout / Grid / set grid defaults**

Major Grid Interval – This defines the interval between minor grid points and major grid points

Maximum Drawn Points – This defines the maximum number of grid points that will be displayed on the diagram when users zoom out, beyond the defined number no grid points will be displayed.

Drawing Grid Percentage

The drawing grid is used for any objects to snap to and is the snap between each grid point.

For example the setting could be a draw grid spacing of 10% and a print scale (refers to the grid spacing) of 5mm, thus when printed with a scale of 1:1 there will be a distance of 5mm between each grid point. As the print scale changes the number of grid points within a border will change but the size of the border will remain the same.

Note

Once the diagram has been created, any changes to the spacing will not be applied on that diagram, it will only be applicable to new diagrams.

Exercise:

Modify your diagram scales as desired.

Management of Multiple Diagrams

It may be a requirement to have more than 1 diagram for the same design data, for example if users wish to display the information in a different way in another diagram. To create additional diagrams within the same design:

- ⇒ Right Click the design name in the project browser
- ⇒ Select **New Diagram**
- ⇒ Define a name, style set and reference diagram for the new diagram
- ⇒ Define the grid spacing and physical layout of the new diagram

Note

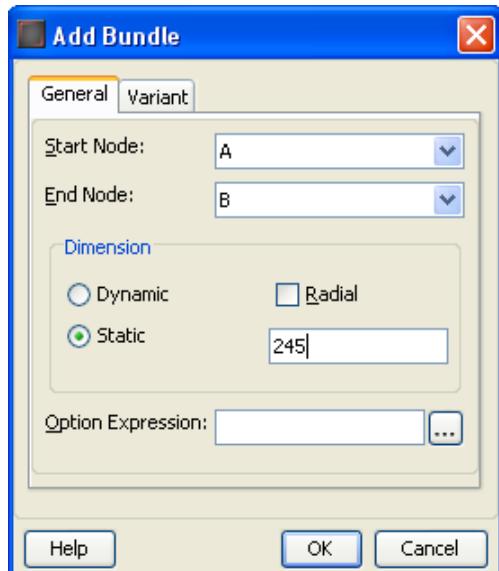
Users should note that the new diagram will retain a link to the original diagram , thus any changes made in the new diagram will be reflected in the original diagram,

Creating the Harness

Creating the Branch Layout

There are 2 main methods to create a bundle.

- ⇒ Adding the bundle directly. Once each node is placed there is an edit box to enter the length
- ⇒ Defining the bundle. The bundle details are entered first via a dialog then the nodes are placed on the diagram



- ⇒ Select Add / Bundle and choose the entry method OR
- ⇒ Select the add bundle icon

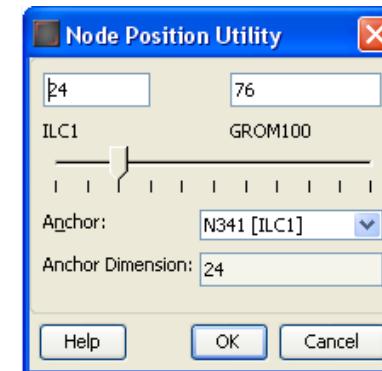
Note

For each method the bundles can be entered dynamically (scaled on the diagram) or statically (user defined length irrespective of diagram length). Bundles can be proportionally stretched and shrunk, this is activated using the proportional stretch icon

Node Position Utility (NPU)

Where 2 bundles intersect or when a component is to be placed on a bundle a Node Position Utility dialog will pop up to prompt the user to define the intersecting point:

- ⇒ Select to add a component or new bundle
- ⇒ Placing the cursor where you wish to insert the new bundle or object
- ⇒ Define the intersecting distance



To specify the distance from each node

- ⇒ Enter the specific value for one side and the other side will correct automatically OR
- ⇒ Use the slider to place the node

The **Anchor Node** will be the node on the bundle that the new node will be referenced from. If the main bundle length is modified, the length between the new node and its anchor node will not change.

Note

Any nodes that are at a bundle end (connector or junction) are **structure nodes**. Any nodes that are added using the NPU are **reference nodes**. Rules for reference nodes are:

- ⇒ They are always referenced from another node (anchor node)
- ⇒ The bundle cannot bend at a reference node

Exercise 2:

Create branch layout:

Using information shown in exercise 2 on the exercise worksheet, create the branch layout

Adding Components

For each component there is

- ⇒ Core data
- ⇒ Properties
- ⇒ Additional data
- ⇒ Additional components

Core Data is core data required for that component and is displayed on the General Tab.

Properties are the standard way of assigning user defined properties to the component. These may be translated to different languages if the language translation file is loaded and up to date

Additional data is data specific to that component that is not core e.g. connectors can have routes assigned and this is available through an extra tab

Additional components allows users to add or remove additional components that are associated with the main component.

Adding a Connector:

- ⇒ Select **Add / Connector** OR select the add connector icon
- ⇒ **Locate** the connector node

The core data includes:

- ⇒ the terminal material each of the cavities in the connector should use
- ⇒ the datum point of the connector if different than the default
- ⇒ if the connector should or should not have seals
- ⇒ IDC connector definition

Library part numbers can be associated to connectors via the library part field.

Adding a Ring Terminal:

- ⇒ Select **Add / Ring Terminal** OR select the add ring terminal icon
- ⇒ **Locate** the ring terminal node

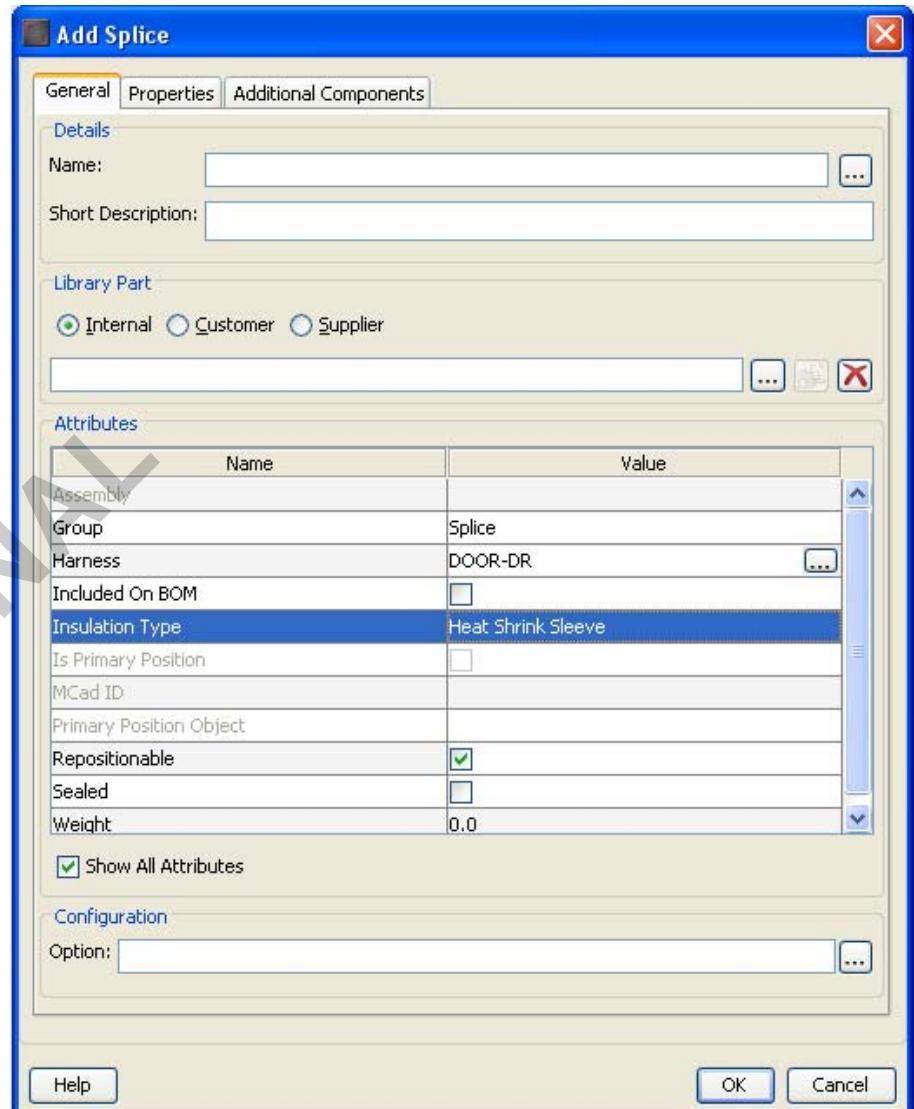
The core data includes:

- ⇒ the terminal material for each wire associated to the ring terminal
- ⇒ the datum point of the ring terminal if different than the default

If a single cavity connector exists on the diagram, it can be converted to a ring terminal:
⇒ Select Actions/convert to ring terminal

Adding a Splice:

- ⇒ Select **Add / Splice** OR select the icon
- ⇒ **Locate** the splice node



The core data includes:

- ⇒ the type of splice (Ultrasonic Weld, Splice, Solder Sleeve)
- ⇒ the type of insulation required (Heat Shrink Sleeve, Spot Tape)

Note

Although possible, it is not necessary to specify a Splice Part Number, as splice parts can be automatically selected following the criteria specified in the components database.

Clips and Grommets

To Add other components such as a clip or grommet:

- ⇒ Select **Add / Clip** OR use the add clip icon
- ⇒ OR **Add / Grommet** OR use the add grommet icon
- ⇒ Locate the node

Clips and Grommets may be repositioned on a bundle by using a 'flip symbol' function – Users must right click and select **flip symbol**.

Clips have a 'set view from direction' utility which enables the user to specify from which direction the clip symbol should be viewed from. This will help harness manufacturers interpret the correct clip orientation when placing it onto a harness.

To define a clip view:

- ⇒ Right click the clip
- ⇒ Select set view from direction
- ⇒ Define the direction (left or right) from which the clip should be viewed

Note

The style set assigned to the diagram controls the display of component symbols, names, part numbers and cavity tables etc

Exercise 3:

Adding the components:

Using information shown in exercise 3 on the exercise worksheet, add all of the components

Adding Wires

Select **Add / Wire** OR the add wire icon



Wire Name	From	From Cavity	To	To Cavity
2N-SPKR-LR-1-1319	ILC1	16	SPKR-LR	2
2N-SPKR-LR-2-1321	ILC1	24	SPKR-LR	1

The wire dialog is different to all other dialogs due to the nature of wires. It is used to define the 2 end locations of a wire. The Edit button can be used to enter the wire information. The edit wires form is divided into 3 tabbed areas that enable users to assign details to the wire such as color, spec, part number, length changes and wire marker information.

The **Additional Info** tab is used to specify any centre strips on the wire and/or any wire markers.

The **Details** tab enables users to define a property or attribute against the wire.

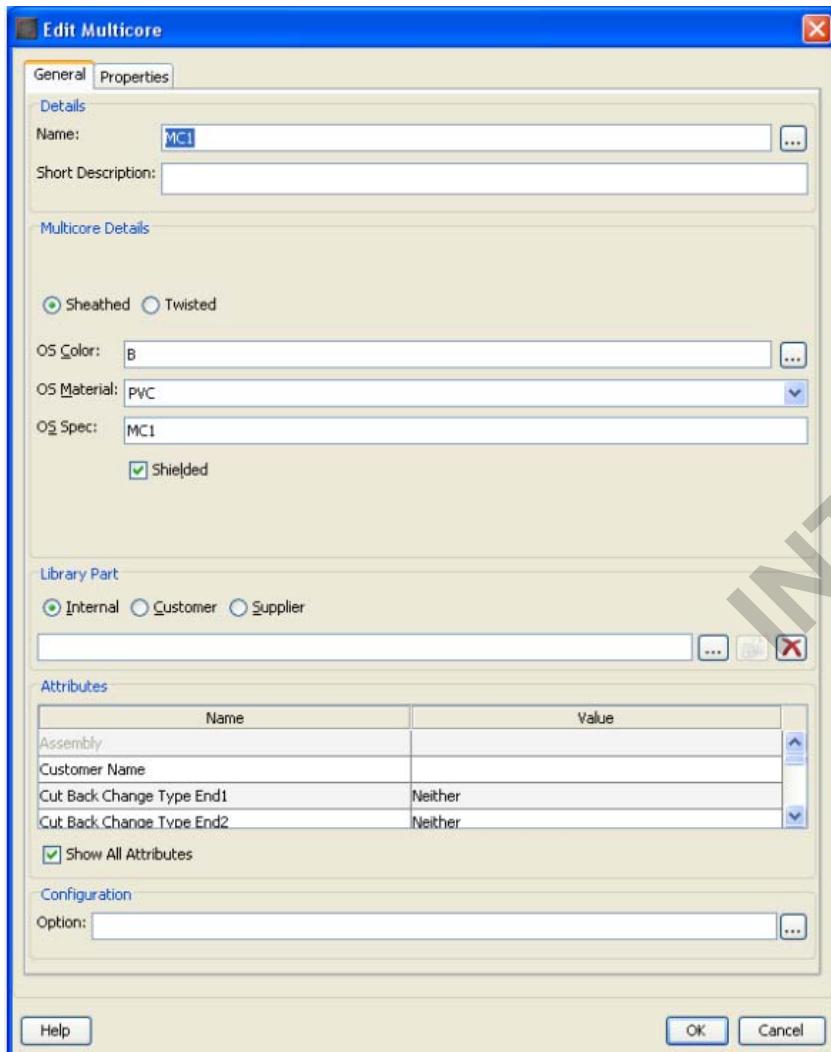
The **General** tab is used to define wire color, spec, material etc

Note

Wire details can be edited at any point during the design process.

Adding Multicores

- ⇒ Select **Add / Multicore** OR select the add multicore icon
- ⇒ Select Add new multicore
- ⇒ Select the Edit button to enter the Multicore Details



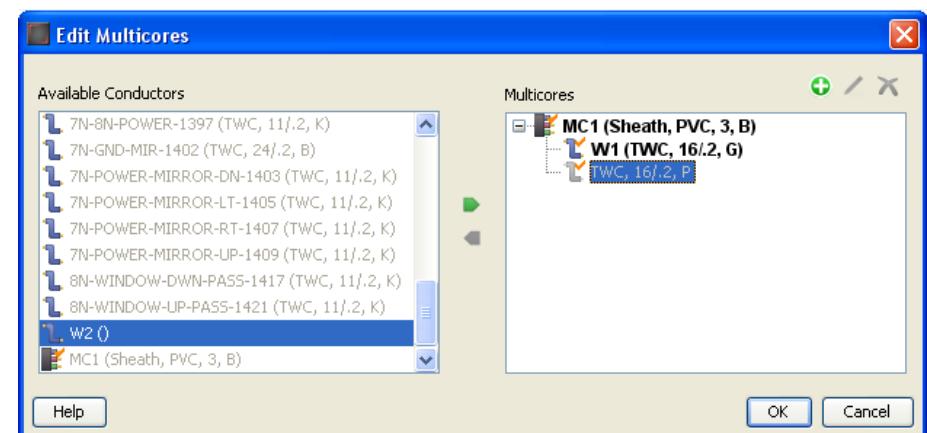
Adding by library part:

If the library part is known the user can assign the part number to the multicore:

- ⇒ Click the ellipse button within the library part area of the Edit Multicore form and search for a specific part number

Once the library part has been added the user must associate wire(s) to the relevant inner core. Only those wires with matching details (color, spec etc) will be available for selection:

- ⇒ Select the multicore reference in the edit multicores form
- ⇒ Open the multicore details in the selected area
- ⇒ Highlight the inner core to be assigned in the **Selected** area
- ⇒ In the **Available** area the system will display possible matching wires
- ⇒ Highlight the required wire and click **Add** to move it to the **Selected** area, thus associating it to the multicore library part
- ⇒ Repeat the steps to assign other innercores
- ⇒ If an innercore remains unassociated the system will prompt the user to accept an auto wire generation. This will create a new wire and associate it to the unassociated innercore.



Adding by part details:

When a part number is not known the system can select the correct part during processing using the details provided.

- ⇒ Select **New**
- ⇒ Define the multicore base details
- ⇒ Assign wires to the multicore by moving them from the **Available** area to the **Selected** area (this is not limited at this point as no library part has been chosen)
- ⇒ During processing the system will search the library for a part number that matches the details of the multicore, if no part can be found a warning will be populated in the processing log file

Exercise 4:

Create wires and multicore wires:

Using information shown in exercise 4 on the exercise worksheet, create the wires and multicores.

Overbraid application

To add additional protection to a bundle an overbraid may be assigned. To add an overbraid to a bundle:

- ⇒ Click the Insert overbraid icon
- ⇒ Click **New** to create the new overbraid reference
- ⇒ Click **Edit** if you wish to change the overbraid name
- ⇒ select the wires or multicore the overbraid is to be assigned to and click **Add** to move the selected wires from the available area to the selected area

The overbraid is referenced in the diagram browser

Assemblies

Two types of assembly can be added to a harness design:

- In-house assembly where all subcomponents are purchased individually and brought together to create the assembly
- Standard assembly where the assembly is purchased with all sub components already included.

Note

An in house assembly is not to be confused with a standard assembly. In house assemblies are only used for non electrical components, typically components that are used for packaging, and that would have a group name of 'other'. When the library part has the group 'in house assembly' the BOM attribute will remain set against each sub component.

The bill of materials (BOM) for the 2 types of assembly will differ. The BOM for the in house assembly will include all sub components whereas the BOM for the standard assembly will only report the assembly part number, no sub components will be considered.

To assign an assembly to a harness diagram:

- ⇒ Ensure a library part exists in the Library tool containing all assembly sub component information
- ⇒ In the harness diagram select Edit/Assemblies or use the assemblies shortcut icon
- ⇒ Click New and insert a name for the assembly or use the default
- ⇒ Select the components to be inserted in the assembly from the Available list and click Add to assign them to the assembly
- ⇒ Click OK

The assemblies folder will now be populated in the design browser.

To assign a part number to the assembly:

- ⇒ Right click the assembly in the design browser
- ⇒ Select Edit assembly
- ⇒ Type a part number in the Library part field or browse for a suitable library part

Properties may be assigned via the properties tab.

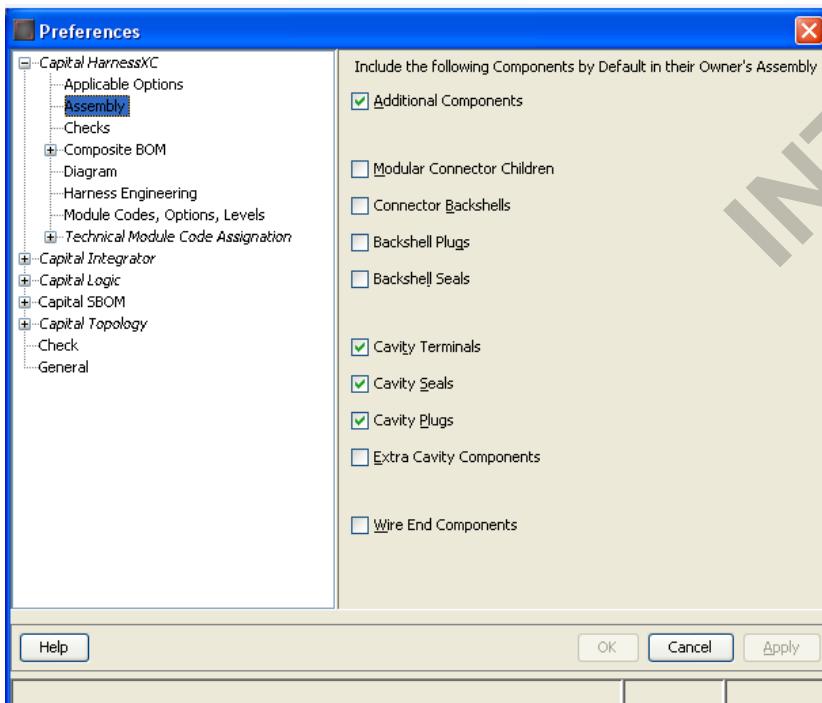
The 'include on BOM' attribute for all sub components will be removed if the library part has group 'assembly'.

A DRC is available to highlight when assembly contents in a diagram do not match those in the library part.

An assembly table may be inserted on the harness diagram to identify the assembly and its contents.

Project Preferences and Assemblies

Objects included in an assembly may have additional parts and associated components (terminals, seals) assigned to them. It may or may not be the intention of users to include these additional parts in the assembly configuration. A set of project preferences exist that will identify whether a sub component is automatically considered as part of an assembly:



When the boxes are unchecked, those associated objects are not included in the Assembly, nor are they considered in the Assembly BOM. When checked, users will need to edit the assembly details to include the required sub components. All subsequent Assembly BOM reports will display the included sub components.

Design Rule checks exist to validate the assembly configuration.

Connector assemblies

In the case of a connector that is to be assigned the status of assembly there are a number of different steps to follow:

- ⇒ Ensure a connector assembly has been created with the group name assembly in Capital Library
- ⇒ Insert the connector on the harness diagram
- ⇒ In the add connector dialog check the connector assembly box
- ⇒ Enter the number of cavities the connector will have (this should match the assembly part in the library)
- ⇒ Assign a library part – either type the known part in the available field or browse for an assembly part
- ⇒ Click insert , then ok

In this case the connector is considered to be the assembly and the connector assembly part number will be listed in the BOM.

Insulation

Introduction to Insulation

Insulation is the protective covering over the bundle of wires where tubes and tapes are generally used.

The location in the vehicle where the insulation is required would determine the type of materials and applications of tape/tube.

E.g. if waterproofing was necessary (engine of a car) then the tape would need to be made of a waterproof material (PVC) and the application should ensure that no water can get to the wires e.g. overlapping the tape.

Correct calculation of insulation length and type is dependent on their accurate definition on the harness. For example there are many types of tape and tube and their lengths or selection will vary depending on the diameter and length of the bundles they cover and, in the case of taping, the method of application (overlap, spiral etc). The system will also take information defined in the system engineering preferences to aid in the selection and calculation of insulation:

- Tubes may have a selection tolerance and/or a tube selection order (cut exact, from reel, cutshorter) defined.
- Tape may have a percentage add on defined and the calculation method set to consider either node ends or bundle diameter.

There are two main types of insulation:

- ⇒ **Continuous Insulation Runs:** insulation runs between two points (continuous taping, tubes, space tape)
- ⇒ **Node Insulation:** insulation is restricted to a single node (spot tape and breakout tape)

Tape Applications

Continuous taping can be:

- Spiral tape with no gap
- Overlapped tape
- Spiral tape with gap
- Space Tape

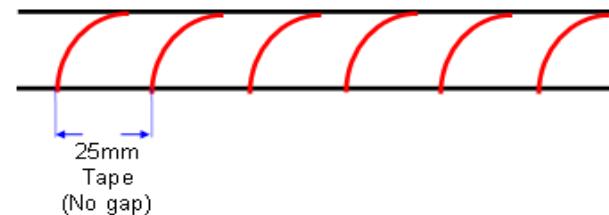
Number of Turns:

Enter the number of overlaps at the start and end of the branch to be taped.

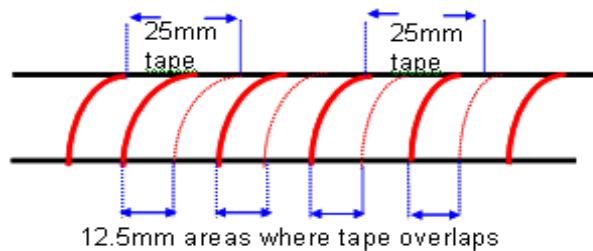
Distance Between Items:

The distance between items for tape applications represents the distance from the left hand side of the first piece of tape to the left hand side of the second piece of tape. E.g.

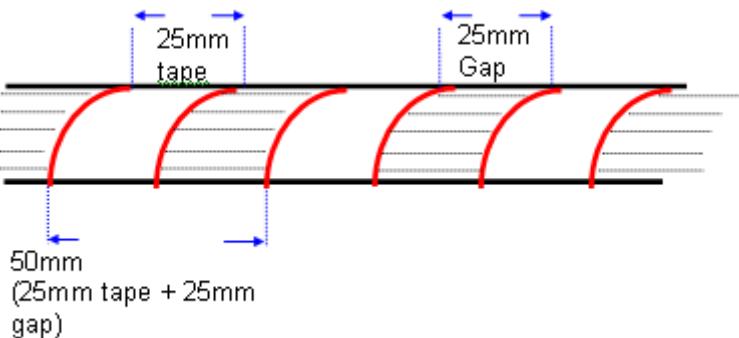
For **spiral tape with no gap**, the distance in percentage would be 100% or in relation with the width of the tape would be 25mm (where the width is 25mm)



For **overlapped taping**, the distance is the tape width minus the length of the overlap. For example the 'distance between items' for 25mm tape with an overlap of 50% would be 12.5mm in relation with the width of the tape.



For **spiral taping with a gap**, the distance is the tape width plus the desired gap. For example the “distance between items” for spiral tape 25mm wide with a gap of 25mm would be 50mm or 150%.



Space tape:

Space tape is multiple applications of spot tape.

Distance Between Item:

The distance between items represents the distance between pieces of space tape e.g. 100mm.

Distance From a Connector:

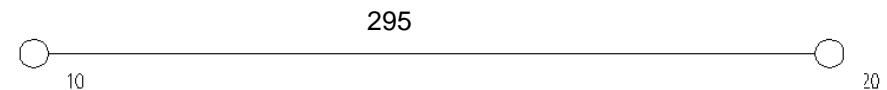
The distance between a connector and the first piece of tape e.g. 25mm.



Longest Distance Allowed:

The maximum permissible untaped section of the branch. This is used if the distance from the last piece of tape placed and the end of the branch is less than the distance between items.

An example of when this would be used is if there is a distance between items of 100, and branch length is 295mm.



In this case, the first piece of tape would be 100mm away from Node 10, The next piece 100mm from that. There would then be a 95mm distance of untaped branch. If 75 were entered into the ‘Long Dist.’ field, then a piece of tape would be added 75mm away from last piece of tape.

Number of Turns:

The number of overlaps at the start and end of each application (or in the case of spot tape, the number of turns per application)

Note

By default the calculation for the amount of tape required on a bundle uses the diameter at the start and end nodes for that tape application. If users prefer the system to use the bundle diameter this must be specified in the **Harness Engineering parameters** before calculations are run (overlap/spiral tape calculation method). A tape usage percentage add on may also be defined here.

Node taping can be:

Spot tape -

Spot tape is one application of tape at a node.

Number of Turns:

The number of overlaps of tape for this application.

Breakout tape -

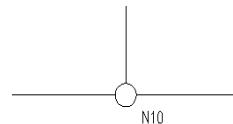
Breakout tape is the application of tape at a junction.

Turns per Break:

The number of overlaps at the breakout node.

Turns per Branch:

The number of overlaps per branch in breakout.



In the example shown here; N10 would be the breakout node, and there are 3 branches in the breakout.

Adding Continuous Insulation

There are two methods of specifying which bundles the insulation covers:

⇒ **Selecting each bundle**

⇒ After selecting all bundles, hold the shift key whilst double clicking in free space (to end the selection) and the system will apply the insulation as one continuous piece. If the user selects connected and unconnected bundles, the system will apply the insulation as separate continuous runs.

⇒ Without holding the shift key, the system will apply the insulation to each individual bundle, unconnected.

⇒ **Selecting the start and end node**

The system will apply the insulation between the two points and treat multiple bundles connected between these points as one continuous run. The user can only select one run at a time.

Using this method does not require additional structure nodes as the NPU can be used to create a reference node along a bundle.

⇒ Select **Add / Insulation Run /** or select the create insulation icon

⇒ Select **Bundles OR Start and End**

Insulation can be a single layer of insulation or multiple layers. The layer closest to the bundle of wires is displayed at the top of the list.

Insulation may be added manually or automatically using an insulation code.

To add insulation manually:

⇒ Open the Edit Insulation run form

⇒ Click **New**

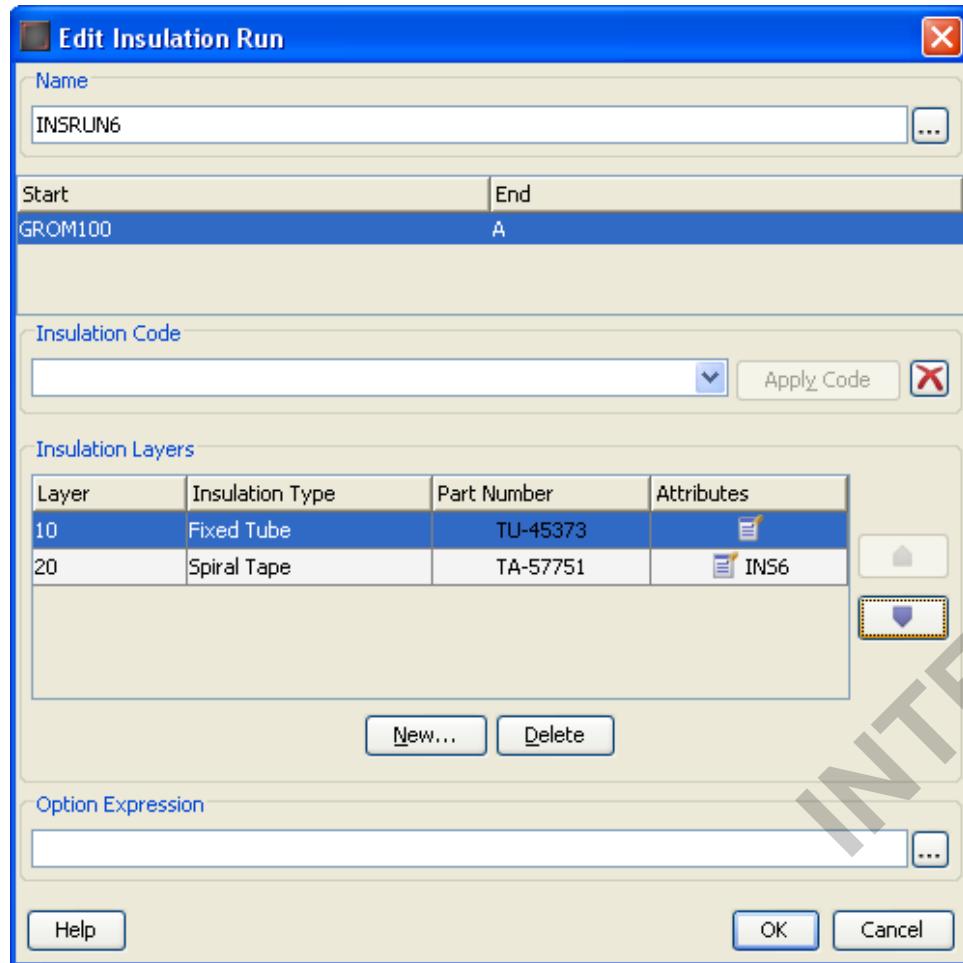
⇒ Select the insulation type

⇒ Define the insulation Library Part/Details/no. of turns etc

⇒ Click **OK**

⇒ **Repeat the steps to add another layer**

To edit any of the layers users can click on the Attributes button to the right of the specified layer.



If no library part is used, the system shall select the correct part using the information supplied in the attribute table (color, material etc).

Relevant fields will be made available when the insulation type is chosen.

Insulation Codes

To save re-entry of the same insulation, insulation codes can be used for ease of use when authoring the harness.

These codes are created in **Capital Project** under the **Object Type Information** for **Insulation Run**, **Spot Tape** and **Breakout Tape** and refer to one or more (insulation run) layers of insulation. They can be created as a system or a project code.

The code is then assigned to a branch (or branches) on the harness and the relevant layers will be displayed.

To add the insulation code to a branch:

- ⇒ Select the branches to which you wish to add the insulation as described earlier
- ⇒ Select the pre-defined insulation code from the drop down box
- ⇒ Click **Apply Code**

If the insulation code is changed in Capital Project, users must manually regenerate the insulation run in order to ensure that the most up to date insulation information is displayed on the diagram.

- ⇒ In the XC diagram browser right click the insulation run that has been changed
- ⇒ Select **regenerate insulation run**

Note

The insulation name is an arbitrary reference.

An insulation code can contain one or more layers.

The layer attributes of the insulation can be modified as previously described using the attributes button next to the specified layer.

Exercise 5:

Creating Insulation Codes:

Using information shown in exercise 5 on the exercise worksheet, create the insulation codes as required and add to the harness as shown.

Adding Node Insulation

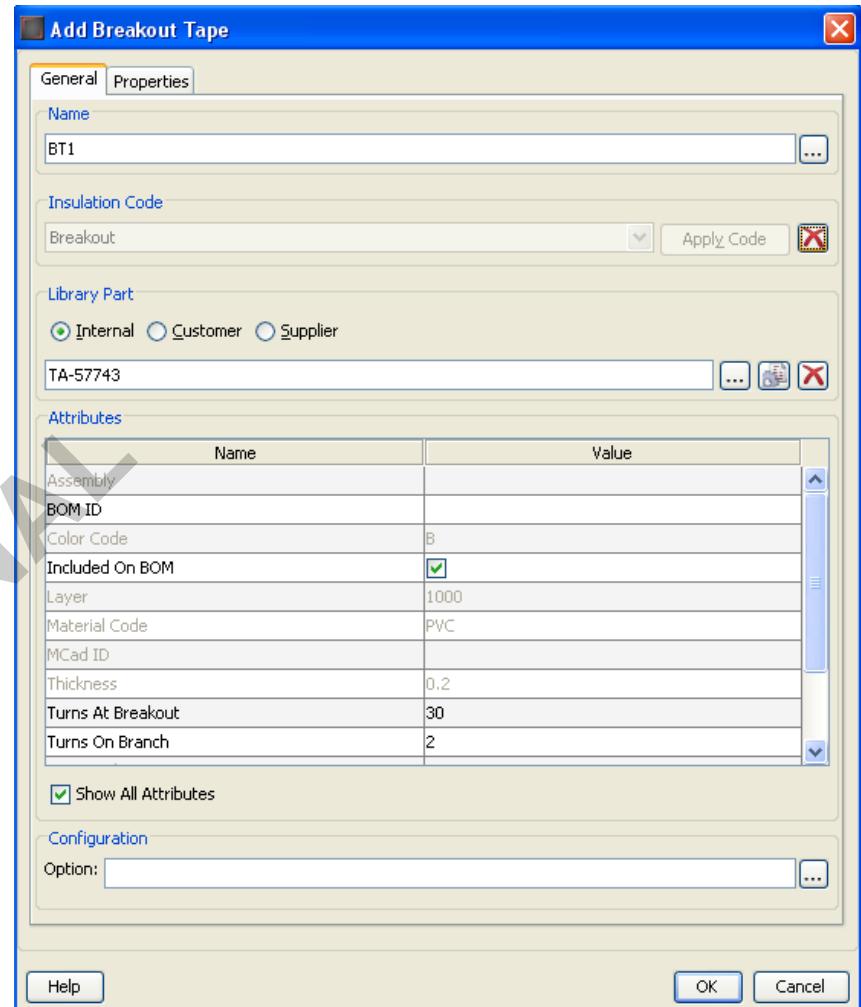
Node Insulation can be Spot Tape or Breakout Tape.

Adding Spot Tape

- ⇒ Select **Add / Node Insulation /**
- ⇒ Select **Spot Tape**
- ⇒ OR select the icon
- ⇒ Select a reference node or use the NPU to **locate** the spot tape
- ⇒ Enter the library part OR the tape details for the required tape or the appropriate code
- ⇒ Enter the number of turns

Adding Breakout Tape

- ⇒ Select **Add / Node Insulation /**
- ⇒ Select **Breakout Tape**
- ⇒ OR select the icon
- ⇒ **Locate** the breakout tape; it can only be placed at a junction of 3 or more branches or select the appropriate code



- ⇒ Enter the library part OR the tape details for the required tape
- ⇒ Enter the number of turns per break and per branch

Note

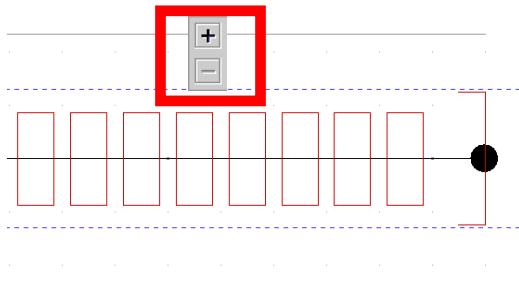
The layer defaults to 1000.

Modify Linestyle Graphics

Resize Linestyles

It is possible to alter the linestyle graphic by resizing either side of the linestyle.
Either:

- ⇒ **Right Click** on the insulation graphic OR
- ⇒ **Right Click** on the insulation run in the tree browser



The blue dotted line above represents the original width of the linestyle (this does not appear on the diagram). By using the + and - keys the width of the linestyle can be increased or decreased.

Note

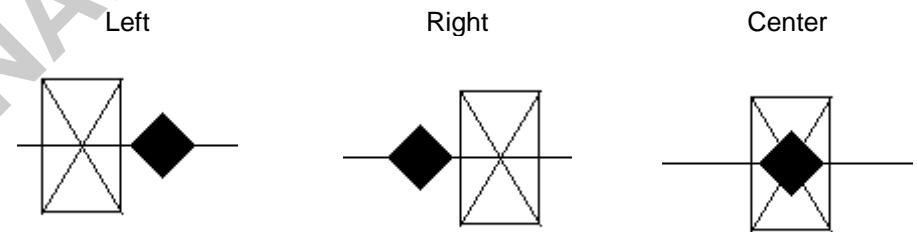
The linestyle cannot be decreased more than the original width.
Where there is more than one layer, the outer most layer determines the **maximum** width the linestyle can be and the inner most layer determines the **minimum** width the linestyle can be.
If a branch is angled and the angle is less than 30° the insulation linestyle will not be rendered.

Move Spot Tape Symbol

It is possible to move the symbol with respect to the node.
Either:

- ⇒ **Right Click** on the spot tape graphic OR
- ⇒ **Right Click** on the node insulation in the tree browser
- ⇒ Select to **Align Spot Tape symbol**

Selecting to move the spot tape (Move Left, Move Right, Move Center) will move the spot tape symbol with respect to the node.

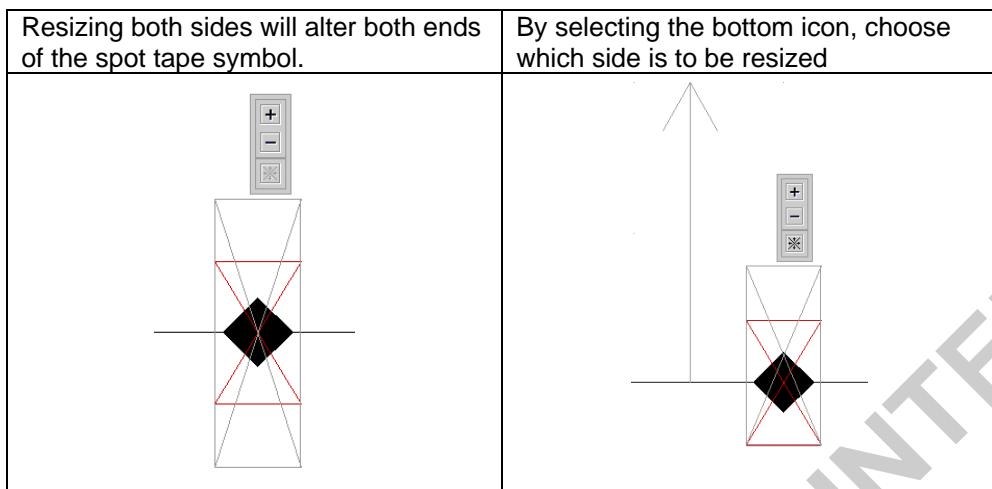


Resize Spot Tape

It is possible to resize the height of the spot tape.

Either:

- ⇒ **Right Click** on the spot tape graphic OR
- ⇒ **Right Click** on the node insulation in the tree browser
- ⇒ Select to **Resize** the spot tape symbol



Manipulation of linestyle displays

The way in which the insulations are displayed on a diagram can be changed via the style sets. For example some users may not wish to display all of the insulation layers belonging to a particular insulation definition along a bundle. It may be sufficient to only display the outer layer, in this case users may do the following:

- ⇒ Open the style set assigned to the drawing (Edit / style / style sets)
- ⇒ Navigate to the Insulation Run folder
- ⇒ Select **Shape**
- ⇒ Check the 'hide insulation layers under run?' box
- ⇒ Click Apply and OK to close the form
- ⇒ Select the diagram objects and right click, apply style

The bundles with insulation should now only display the outer insulation layer to indicate that an insulation run exists on that bundle. To view the individual layers of the insulation, open up the insulation run in the diagram browser.

Override graphic style

- ⇒ Right Click the linestyle/override styling/Edit graphic style
- ⇒ Select the desired shape
- ⇒ Click OK

The linestyle will change to the selected shape however this will revert to the defined style when style sets are re-applied.

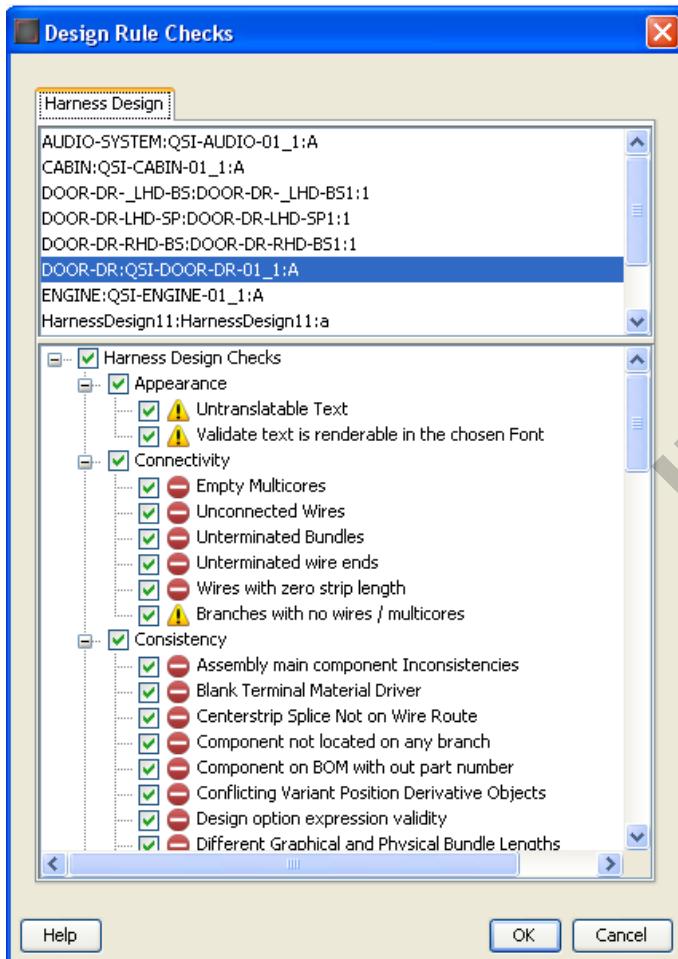
Exercise

If time allows the instructor will talk you though how to change the default style set to change the display of the bundle insulation – Remember on a network installation only 1 person at a time will have access to the style set.

Design Rule Checks (DRCs)

To help ensure that the harness is being designed without problems, the DRCs can be run to highlight potential problems or errors. By running the DRCs regularly throughout the design process, errors can be reduced. The DRC severity and frequency of application can be set in Capital Project under Preferences.

⇒ Select Tools / Design Rule Checks

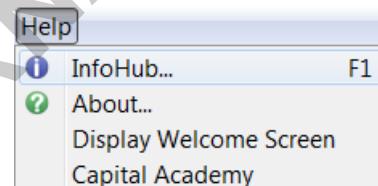


- ⇒ Select which checks are to be run
- ⇒ The System will present the results in the command bar

The DRCS are divided into sections:

- ⇒ Appearance DRCS
- ⇒ Connectivity DRCS
- ⇒ General
- ⇒ Inclusion
- ⇒ Library Consistency
- ⇒ Naming

The infohub gives detailed explanations of all DRCs



The following are examples of DRCS that are relevant to the Harness XC design:

- ⇒ **Component not located on any Branch:** a component does not exist on a branch
- ⇒ **Minimum Bend radius:** Checks for bundles with an arc that has a bend radius lower than the specified minimum
- ⇒ **Components housing definition validity:** checks that additional components at the node (cavity or node components) exist in the housing definition of the main node component
- ⇒ **Inappropriate Terminal Material or Type for wire:** informs the user if the terminal material / type of the selected terminal doesn't match the wire terminal material or type defined in Cavity Component Management
- ⇒ **Unsourced part number (no supplier numbers):** Part has no preferred supplier defined

View DRC Results

To view the results of the DRC checks expand the information bar (underneath the diagram window). Any warnings will be displayed.

Harness Design C01-6677:B:Simple

- ✖ Status of library part for [CLIP101](#) is 'new', not current
- ⚠ Splice [W10](#) has no library part

By selecting the underlined harness object the system will hyperlink to the object in the diagram; the node will be highlighted for node objects such as splices, clips, connectors etc; the route will be highlighted for routed objects such as wires and insulations.

Exercise:

Run the DRCs on your harness and discuss any messages.

Save Design

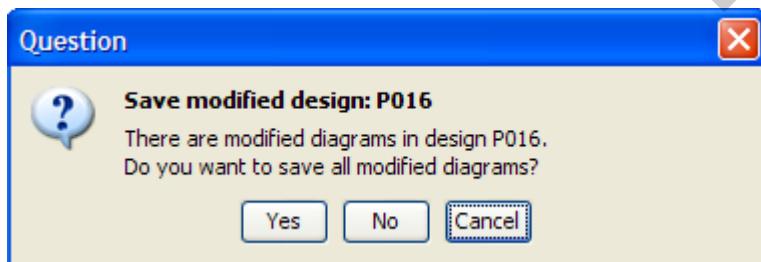
The design can be saved at any point during the creation of the design in any of the following ways:

- ⇒ Select **File / Save** or the save icon. This will save all modified designs in that project OR
- ⇒ **Close the Project using File / Close project** or the close icon. You will be prompted to save all modified designs in that project.



OR

- ⇒ **Close the design** by Right Clicking on the design in the tree browser and selecting Close. Alternatively use the red x at the top right hand corner of the diagram you will be prompted to save that specific design if it has been modified



Style sets

It is often the case that several diagrams of the same harness are produced, each with a different representation of the harness information. Suppliers may require a simple stick diagram while customers or management will require a detailed diagram with all component symbols, part numbers and company specific linestyles displayed.

Several different style sets can be prepared and the desired style set can be assigned to individual harness diagrams to render the required information.

Style set creation is a topic that will be discussed briefly later in the course.

To apply a style set to a diagram that is currently active:

- ⇒ Right click the diagram name and select Edit
- ⇒ Select the desired style set from the style set field
- ⇒ Select all diagram objects (CTRL + A)
- ⇒ Apply the style set (Shift + A)

Note

If the diagram has already been rendered with a style set and changes have been made, it is advisable to select individual areas of the diagram and apply the style only to that area.

To modify a style set:

- ⇒ Right click the diagram select Style/ Edit style

This will display the edit style set dialog where users have the correct permission. If the style set is currently being edited, users will be informed that the style set is locked by another user.

If there are several designs in a project that are to have the same style set applied, these can be multi selected and the style set applied to each design in 1 step. Designs within a build list may also have a style set applied in the same way. Restrictions exist where designs have been released or have a domain applied.

If applying a style set to multiple designs in the browser, users can make a choice regarding forcing the apply style as well as updating the border properties and/or re-selecting the border symbol to match the border in the selected style set.

INTERNAL

Chapter 5

Harness Processing

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Harness Processing – Set up

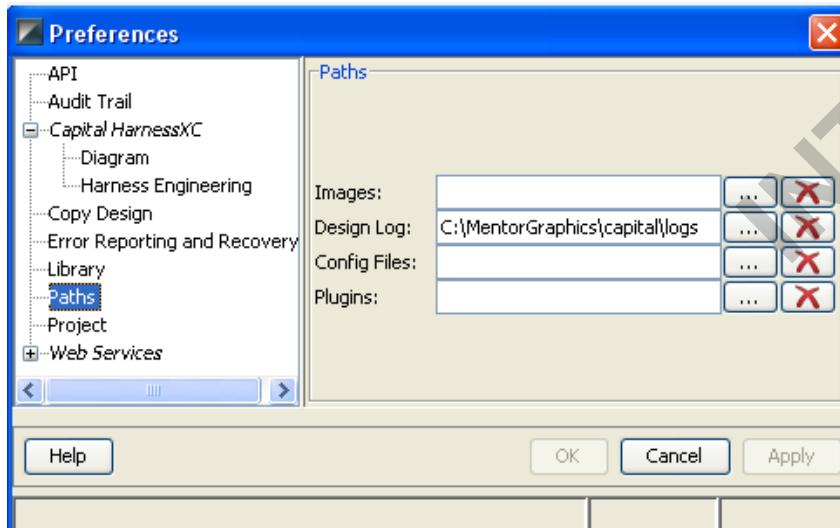
Harness Processing is the method used to calculate exact quantities of components and select correct parts using parameters supplied by the harness and library.

Prior to running the harness processing within CHS a number of paths and engineering parameters must be set:

Set Paths

In Capital Project it is necessary to set the path for the log files so that it is possible to view any errors that may occur during processing e.g. unselected terminals.

⇒ Select **System / Preferences / Paths**



⇒ Set the **Design Log** file path

The design log file is generated during processing and contains information regarding warnings and errors that may have occurred during processing.

Harness Engineering Parameters

There are many parameters used during the processing of a harness, such as

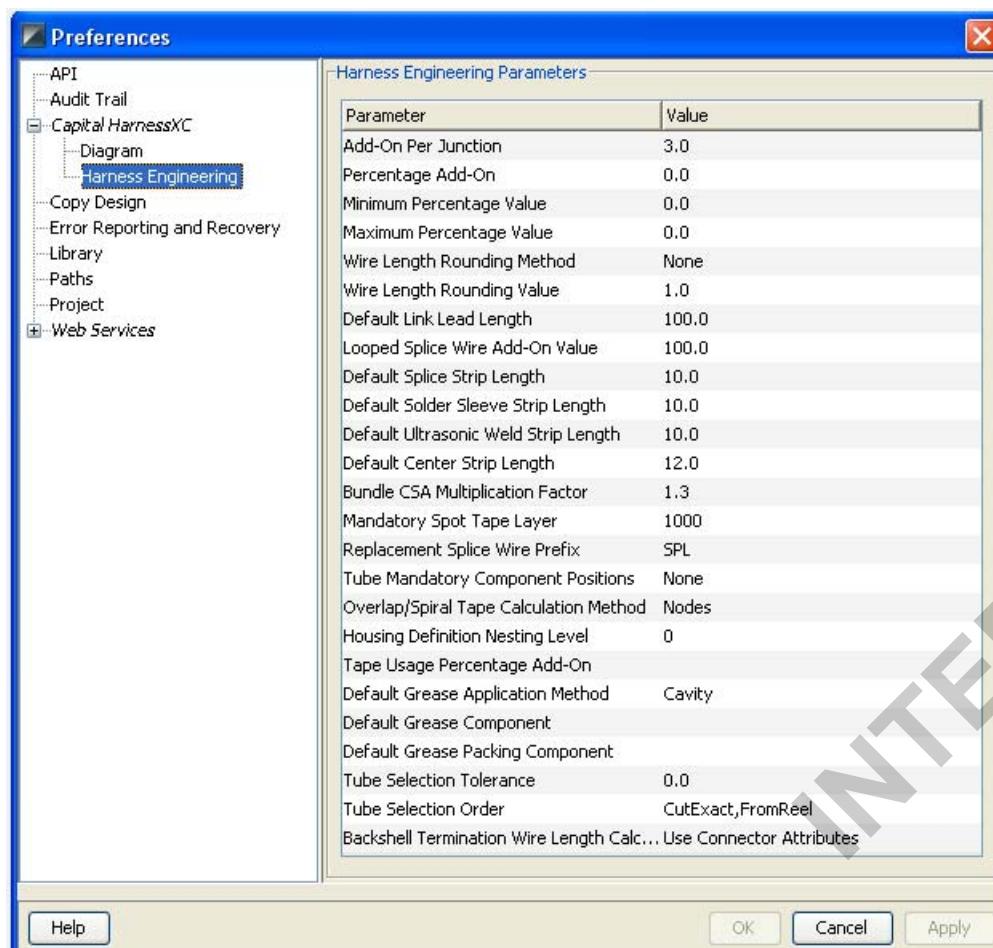
- ⇒ wire length calculations
 - junction add ons
 - rounding methods and values
 - ...
- ⇒ Bundle calculations
- ⇒ Strip lengths
 - Wires
 - Splices

These must be set up at the system level

⇒ Select **System / Preferences / Harness Engineering (in Capital Project) or File/Project preferences within harness XC**

Clicking on the relevant parameter value cell will allow users to define the required value.

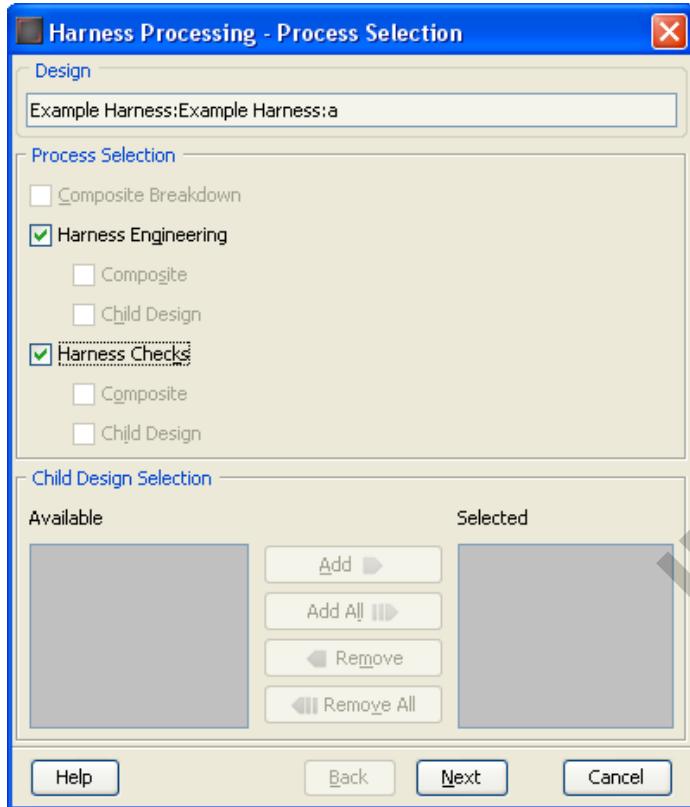
It is important that the settings here are accurate as they will affect calculations for every harness on the database.



Harness Processing

Initiating Processing

⇒ Select Tools / Harness Processing...



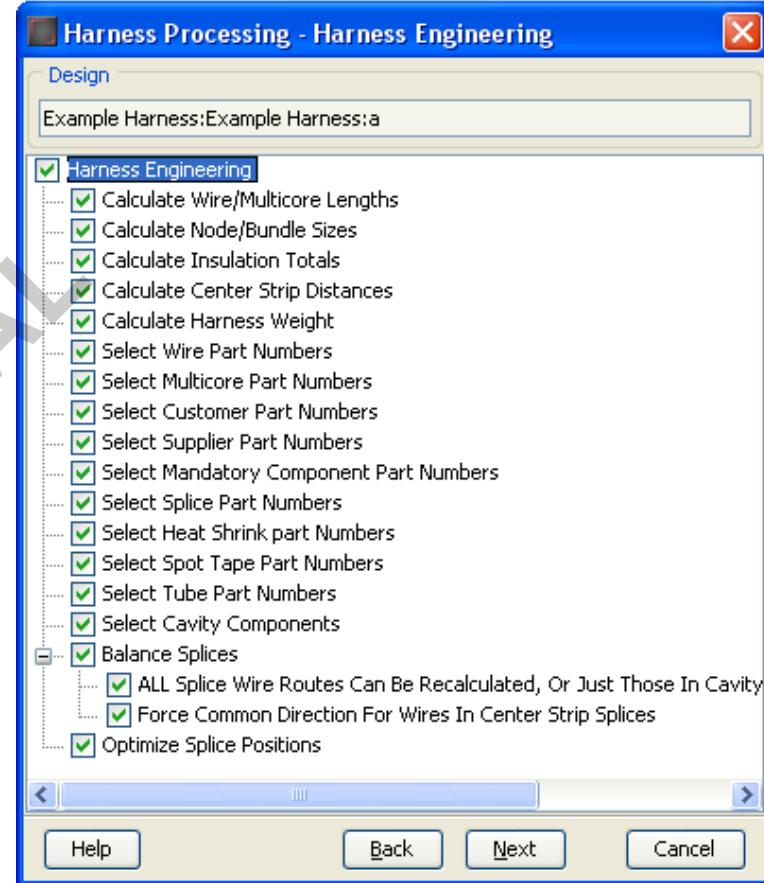
The processes available will be presented.

Note

If all of the processes are chosen, the system will present each of the following in turn, prompting the user to select 'Next'.

When the selected design is not a composite harness, the composite and derivative options are not available. See later chapter.

Harness Engineering



Choose the calculations to be performed

The following explains the options:

- ⇒ **Calculate wire/multicore lengths:** calculates exact wire lengths, using the previously set up parameters in System Preferences amongst other things

- ⇒ **Calculate node/bundle sizes**
- ⇒ **Calculate insulation totals:** calculates lengths required using the bundle size calculations. Remember that the system parameter in Capital Harness Classic must be set to ensure the system calculates tape requirements using either start and end node diameters or bundle diameter
- ⇒ **Calculate center strip distances:** calculates the position of a center strip on a wire cut sheet
- ⇒ **Calculate harness weight:** calculates the overall harness weight using the library details
- ⇒ **Select wire part numbers:** selects valid wire part numbers
- ⇒ **Select multicore part numbers:** select valid multicore part numbers
- ⇒ **Select customer part number:** finds the corresponding customer part number for the given internal part number
- ⇒ **Select supplier part number:** finds the corresponding preferred supplier part number for the given internal part number
- ⇒ **Select mandatory component part numbers:** selects all mandatory items found in the housing definition of any harness component.
- ⇒ **Select splice/heat shrink sleeve/spot tape part numbers**
- ⇒ **Select tube part numbers:** using the calculated bundle sizes and the defined tube, the system finds the tube part number to fit the bundle run
- ⇒ **Select Cavity Components:** selects any of the cavity components that have been set for the system to automatically select. Any of these components which have been **manually** entered using cavity component management will be retained
- ⇒ **Balance Splices:** requests that the system relocate splice/weld wires that are located in cavity X to the correct side due to the location of the other end of that wire.
 - ⇒ Calculate ALL Wire Routes will re-route all wires located at a splice/weld. If some wires have been manually forced into the cavities **Left** or **Right** this option will also re-route these wires
 - ⇒ Force common direction
- ⇒ **Optimize Splice Positions:** enforces splice positioning constraints on the diagram where splices are set to allow repositioning

Hide empty cavities

At this point, any cavities that are empty can be removed from the connector cavity tables – this is applied under the style sets:

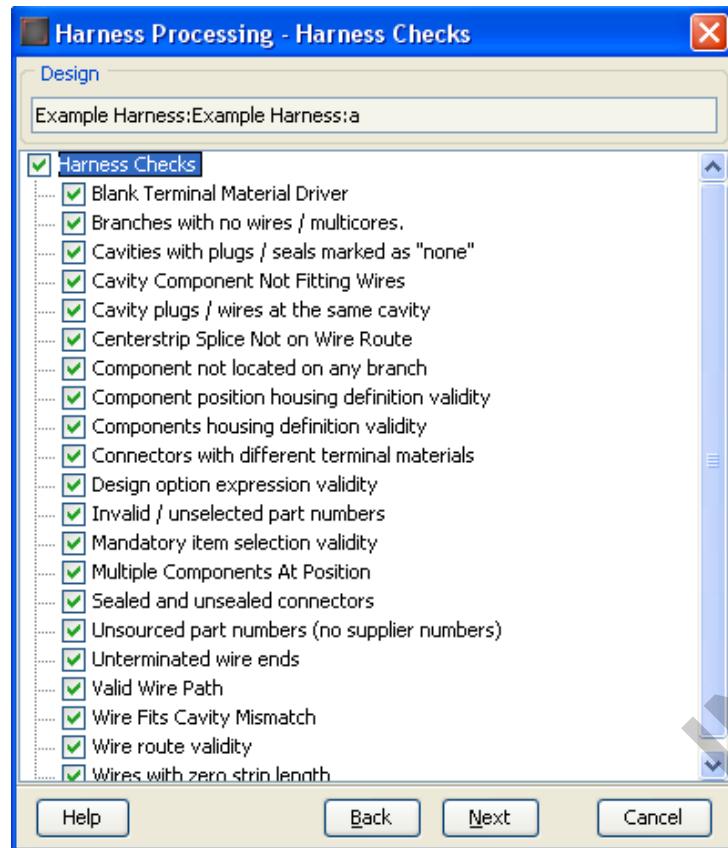
Select **Edit / Style Sets** / select the correct style set
 Highlight **connectors / Decorations / Tables**
 Select the **cavity wire table cavity chart**
 Select **Edit** and check the **Hide empty cavities** box

Scoping of housing information

When registering the harness in XC it is possible to set the scoping against the harness (under the customer harness details tab) – this depicts scope setting i.e. customer and/or manufacturing site and is used by the system when running calculations.

If the customer is set then the system searches for housing components set against the defined customer and scoped in capital Library, if a part associated to the customer cannot be found then a warning will be given in the log file but a suitable terminal will be applied from the standard connector housing. Where manufacturing site is also implemented then again the system will try to match component housing information using manufacturing site associations from the library, again if nothing can be matched a suitable housing will be assigned and a warning given in the log.

Harness Checks



The **Harness Checks** routine is a powerful tool used to highlight any harness definition inconsistencies. It must be run after the **Harness Engineering**. No calculations are performed with this process.

Appendix 3

See Appendix 3 for examples of each harness check and some DRCs detailing the various warnings produced, and a method of amending them if appropriate. Also an explanation for how the exact wire lengths are calculated.

Individual processes

It is possible to process sections of the harness as you are editing. This information will be presented in the browse area and not in the design logs. Selecting **Actions / Processing** allows you to select individual processes to run as follows:

- ⇒ Calculate Wire/Multicore Lengths
- ⇒ Calculate Node/Bundle Sizes
- ⇒ Calculate Insulation Totals
- ⇒ Calculate Centre Strip Distances
- ⇒ Calculate Harness Weight
- ⇒ Balance Splices
- ⇒ Balance Splices (Force Entry Direction)
- ⇒ Part Selection (All or individual component groups)

Exercise:

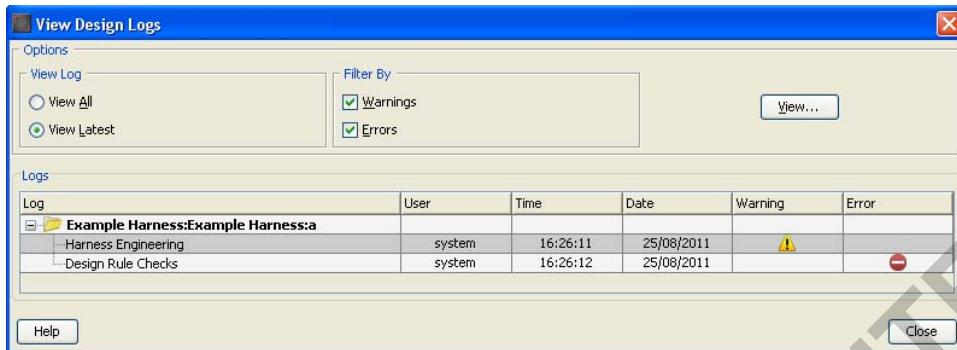
- ⇒ Run Harness Engineering against your harness
- ⇒ Run Harness Checks against your harness

Log Files

Each process creates its own log file to report any problems that may have occurred. These files are stored in the directories set up previously (structured log files and job controller files).

Within the structured log file directory the system creates a folder for each harness that is processed and an xml file for each process is created. These can be viewed in the Design log.

⇒ Select **Tools / Design Log / View**



- ⇒ **View All/Latest:** choosing to view latest will display the latest report for the processes performed on that harness. If a process has been run more than once it will use the xml files created and display the latest one. Viewing all will display all reports
- ⇒ **Filter By:** filters the reports to show only the required entries; Warnings show problems with the harness that can be resolved by modifying the harness or the library; Errors show problems that need to be sent to the support team for help
- ⇒ **Preview:** allows the user to view the selected report
- ⇒ Within the log it is possible to filter the view as necessary. These settings can be per-defined by selecting **Tools / Design Log / Options**

Harness Reports

Basic reports may be generated against the harness to display a record of component or wire or insulation held within the harness. To create a report:

- ⇒ Select Tools / Reports
- ⇒ Select the type of report you wish to generate

The system will automatically generate and open the report.

The types of report available within Harness XC are:

- Harness Component BOM
- Harness Tape BOM
- Harness Tube BOM
- Harness Wire BOM
- Multicore List
- Wire Cutting chart
- Wire List
- Configured Wire List

The reports are all in a pre-defined format but may be saved in an html format and printed.

A more detailed report may be generated using the Capital Enterprise Reporter tool which is an add-on tool.

Appendix 4

Gives more details about the Capital Enterprise Reporter tool and its functions

Chapter 6

Creating Symbols

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Creating Symbols

Capital Symbol allows the creation and management of object symbols to be used with the Capital drawing tools. A symbol is a graphical representation of a harness object.

Several types of symbols are used in the tool suite:

- ⇒ Schematic Symbols (Devices and Grounds)
- ⇒ Plug/Pin Map Symbols
- ⇒ Comment Symbols
- ⇒ Border and Skeleton Symbols

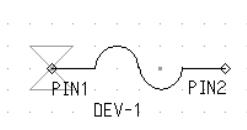
Capital Symbol manages libraries of symbols to help with managing and accessing the symbols.

A **Schematic Symbol** is used to graphically represent the function of the device/component/part. This type of symbol is placed (instantiated) directly and is used for specific understanding of the diagram being captured.

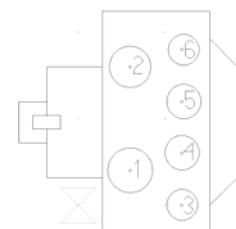
A **Plug/Pin Map Symbol** is used to capture and display a relative physical layout of a device/component/part's points of electrical connection (pins). This type of symbol is not typically placed on a schematic diagram; it is typically used as a reference for choosing and locating the pins to that device/component/part.

A **Comment** symbol is used to graphically represent a harness component (connector face view, clip, grommet etc).

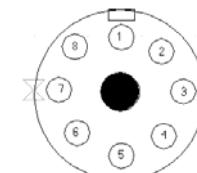
A **Border** symbol depicts the zone area and title block information and helps define the available drawing area for the diagram.



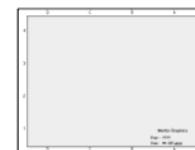
Schematic Symbol



Plug map Symbol



Comment Symbol



Border Symbol

In this chapter we will be looking at Comment Symbols. Comment symbols are the only type of symbol visible on a harness diagram.

Launch Capital Symbol using the Launchpad; Capital Symbol is found within Design Solutions.

Create a Symbol Library

A **Symbol Library** is a collection of symbols grouped according to a categorization scheme to aid in the management, access, and processing of symbols.

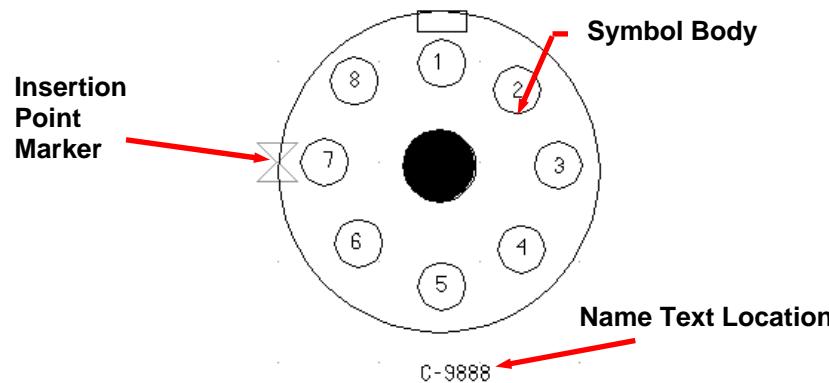
All symbols must be stored in a library. Capital Symbol supports the creation and management of multiple user defined symbol libraries.

To create a new symbol library:

- ⇒ Select **File / New Library**
- ⇒ **Define a name for the library**
- ⇒ **Select the library type (symbol or border)**

Elements of a Symbol

For a Comment Symbol:



There are six main elements to a symbol:

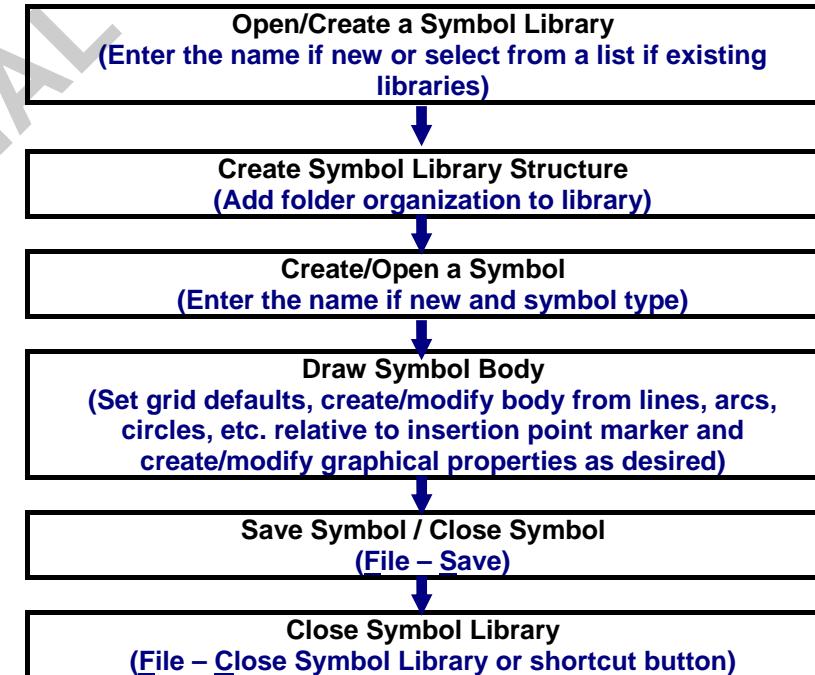
- ⇒ **Symbol name** is a string of characters used to identify the symbol. This name appears in the Symbol Brower Tree. For a comment symbol, if this is the internal part number it will appear with the component when placed.
- ⇒ **Symbol body** (shape) is the graphical image of the symbol.
- ⇒ **Graphical Properties** are graphical attributes of the symbol body. These properties control the appearance of the symbol.
- ⇒ **General Properties** are the characteristics of the symbol that are not easily represented graphically. These properties are in the form of name/value pair text.
- ⇒ **Insertion Point Marker** indicates the point at which the cursor will appear when the symbol is to be placed on a diagram. The insertion point marker is not movable within the symbol diagram; the symbol must be created relative to the location of the insertion marker.
- ⇒ **Name Text Location** is a placeholder for the text name that represents the specific instance of the symbol when the symbol is placed on a logic diagram.

Note

Comment symbols can be assigned to one or more library parts, alternatively, one library part can have more than one symbol assigned. If a symbol has been assigned to a library part then this will be used and takes precedence over a comment symbol named with an internal part number.

Create a Symbol

A symbol must be created within a symbol library. The outline process is as follows:



To create a new symbol:

- ⇒ Right click the library name and choose **New Symbol**
- ⇒ Enter the name of the symbol and select the symbol type

For Harness symbols, the type must always be **Comment**.

- ⇒ Define the symbol settings (physical scale and non resizable)
- ⇒ Click OK

The scale enables users to specify the scale to be used for the symbol, this would typically match the scale used in the harness diagrams to ensure that symbols are of the correct size when inserted on a diagram.

If symbols are defined as non-resizable users will not be able to change their size in the harness diagram.

To change a symbol from non-resizable to re-sizeable:

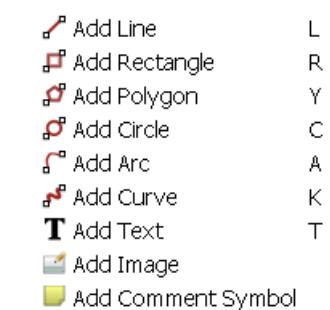
- ⇒ Open the symbol
- ⇒ Select Layout/Grid/Set Grid Defaults
- ⇒ Uncheck the non-resizable box

To set all comment symbol library contents to non-resizable:

- ⇒ Ensure all symbols in the library are closed
- ⇒ Right click the library name and select edit symbol library
- ⇒ Check the non-resizable box
- ⇒ Define if this setting is to be applied to all existing symbols

Define the symbol body

The symbol body would be generated using the drawing commands.



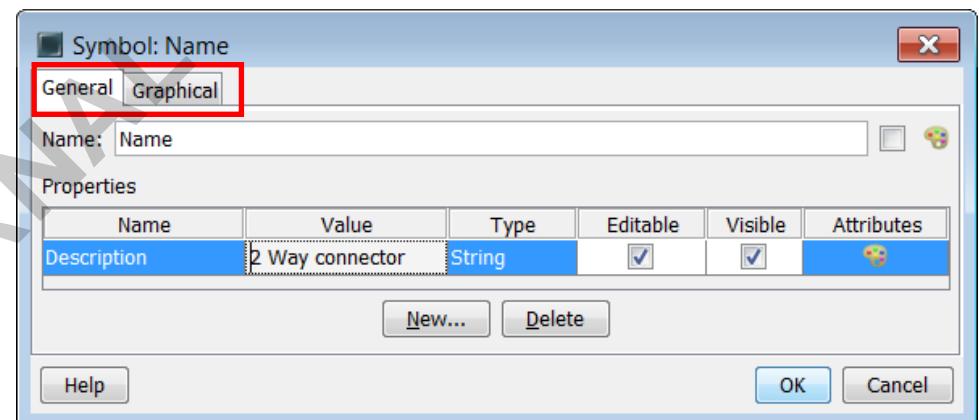
Where several symbols are brought together to create a composite symbol, or where several shapes are used to define a simple symbol, users may implement the 'Bring to Front' or 'Send to Back' functions to specify the overlap preference.

Add Symbol Properties

To add properties on the symbol diagram:

- ⇒ Unselect all objects within the diagram and execute **Edit / Properties** or **popup menu: Properties**.

This will display the **Edit Properties: Symbol** dialog box.



- ⇒ Click New and enter the desired property and value

- ⇒ Click Accept. The property value gets automatically placed on the diagram

Property values can be translated using the switch language feature. This assumes that the language dictionary is loaded and contains the relevant translation for the selected text.

Adding Name text place holders

The Name text placeholder is placed on the symbol drawing and will take the default name assigned to the symbol when it is placed on the drawing

To place the Name text, click **Add** in the menu bar.

Copying Symbols

Symbols can easily be copied to a new library or within the current library using the duplicate function.

- ⇒ Right Click the object to be copied
- ⇒ Select **Duplicate**
- ⇒ Select the new location for the object – if it is to be placed within the same library users will be prompted to define a new name

Save a Symbol

To save the symbol select **File / Save** or the save icon.

Note

It is possible to create a symbol in which parts of the symbol, or blocks, can be instantiated separately. The type of symbol will be covered in the advanced course.

Ensure that you refresh the symbol library in the XC tool whenever an update is made in the symbol tool:

- ⇒ **File / Refresh / Symbol Libraries**

Assigning Symbol Libraries to Project

When placing a component in XC, if the component has a library part number assigned and there is a symbol available in the symbol library with the same name as the library part then that symbol will be added automatically.

In Capital Harness XC users may also assign symbol library sets to a project. These library sets may be used to assign multiple symbols to components.

- ⇒ **Open XC**
- ⇒ **Select Edit / Style / Style Sets / Design / General Settings / Library sets**
- ⇒ **Select Add** and define a library set name
- ⇒ **Click OK**

Now you will need to assign libraries to the library set:

- ⇒ Highlight the library set name
- ⇒ On the right hand side of the form click **Add**
- ⇒ Select the symbol library you wish to assign
- ⇒ **Click OK** and repeat the process until all required libraries are assigned
- ⇒ Use the up and down arrows to change the hierarchical position of the libraries

Note

The order of the libraries determines the hierarchy of symbols to be used. The system will search for an associated symbol in the first library in the list and then move onto the next and so on.

Placing the component symbol in the XC Diagram

The symbol will be placed on the diagram in accordance with the defined style set.

If users choose to change the symbol at any time this can be done by using the following steps:

- ⇒ Right Click the Connector symbol
- ⇒ Select **styling / override symbol**
- ⇒ Choose the required symbol from the available symbol libraries
- ⇒ Click **OK**

Note

A comment symbol can have an option expression assigned and therefore be displayed on the correct derivative after processing.

Manually adding a Symbol to a diagram

It is possible to manually add a symbol to a diagram, this may be required if users wish to add an additional piece of information to the diagram without using style sets:

- ⇒ Select to insert a symbol **Graphics / Add comment symbol**
- ⇒ Select the required symbol
- ⇒ Select the node/area to associate the symbol
- ⇒ Click on the diagram to locate the symbol

Note that the symbol can be re-sized if the user has not opted to check the non resizable box upon creation of the symbol

Exercise 1:

Create symbols:

Using information shown in exercise 1 on the exercise worksheet, create the library and symbols as shown

Management of Symbol Libraries

Users are able to create folders within symbol libraries to aid the organization.

To create folders within libraries:

- ⇒ Right click the library reference
- ⇒ Select **Edit Folders**
- ⇒ Create folders or sub folders within the library. Right Click the library reference and click **New**
- ⇒ Once folders are created symbols may be transferred between them

Moving symbols between libraries

A symbol may be moved from one library to another – this is NOT a copy function and will physically transfer symbols from one library to another.

To move symbols to other libraries:

- ⇒ Right Click the symbol you wish to move
- ⇒ Select **move**
- ⇒ Select the library and folder you wish to move the symbol to
- ⇒ Click **OK**

Assigning a print region to a comment symbol

Sometimes it is desirable to print symbols on individual sheets directly from the harness diagram. If a print region has been established on the comment symbol it will be possible to print the individual symbol directly from the harness diagram it exists in:

- ⇒ Go to File/Print Regions /Add
- ⇒ Define the print region by instantiating a box around the symbol
- ⇒ Right click and select commit
- ⇒ Save the symbol

When the symbol is inserted on a harness diagram it will have its own print region defined and can be printed separately to any other print regions in the diagram.

Advanced symbol creation:

In addition to the standard symbol creation, users may wish to create symbols containing more detailed information. The advanced functions available within Capital Symbol allow users to define datum points which can relate to wire location points within connector and splice cavities, wire table locations in relation to diagram borders, connector datum points and attribute datum points .

Datum points

Datum points can be divided into 3 sections;

- RED Datums**
- Engineered Datums**
- Attribute Datums**

RED (Related Entity Decoration) Datums

This type of datum act as a locator for symbol information, for example:

- **Connector/Splice cavity datum** – acts as a locator to indicate where the connector/splice cavity name on a symbol should be placed if it has been specified in the style set.
- **Cavity wire datum** – this locator indicates where the wire information should be placed in relation to the connector /splice cavity, and also, what type of wire information should be displayed.
- **Design Connector datum**
- **Design Splice Datum**

To place the datum against a symbol:

- ⇒ Open the symbol in the symbol library
- ⇒ Ensure nothing is highlighted in the symbol area
- ⇒ Select **Add / Add Datum**
- ⇒ Choose the type of datum you wish to add to the symbol

If users choose to add a connector cavity datum, they may then select that datum, right click and add an associated datum / wire datum.

Engineered Datums

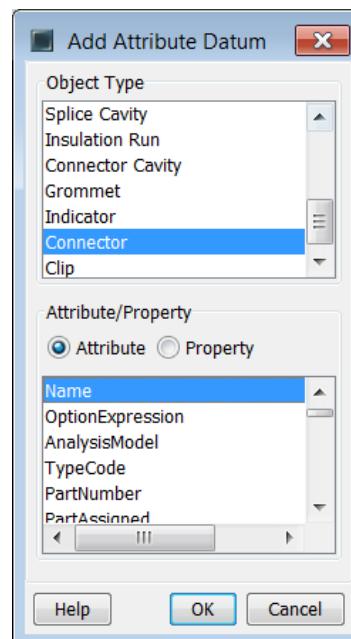
This type of datum point is used on connectors to identify the location of the back, tip or front datum. This information can then be placed against a symbol and used for locating the symbol in a drawing, typically this would be done for a form board drawing.

To place this type of datum on a symbol:

- ⇒ Open the symbol
- ⇒ Click **Add / Engineering datum**
- ⇒ Select the required datum
- ⇒ Place the datum on the symbol

Attribute Datums

This type of datum can be assigned to a symbol representing a specific object type such as a connector or splice. The attributes or properties available for the selected object type will be displayed in the attribute datum form and users may choose which one to assign to the symbol:



This attribute or property will then be displayed on the diagram once the symbol is placed.

Exercise 2:

Create the connector symbol and add datums as directed in the exercise worksheet

Constraint box

Where components with multiple additional components are added to harness diagrams, users may wish to display the additional components on the drawing as well. It is not often necessary for the additional components to be displayed alongside the main node component, users may prefer for the additional components to be displayed in a separate table to the side of the design.

The constraint box may be added to a symbol in Capital Symbol and will act as a boundary for any multi decorations associated to a main node component, the multi decorations will be placed inside this box, until it is full and then another box will be created if needed.

To add a constraint box:

- ⇒ Define a boundary symbol
- ⇒ Add a constraint around this symbol
- ⇒ Click **Add / Add datum / constraining box**
- ⇒ Define the size of the constraining box.

Users will now need to define this constraint box against a connector in the style set for the XC design. – see the styling chapter for an explanation of how to do this.

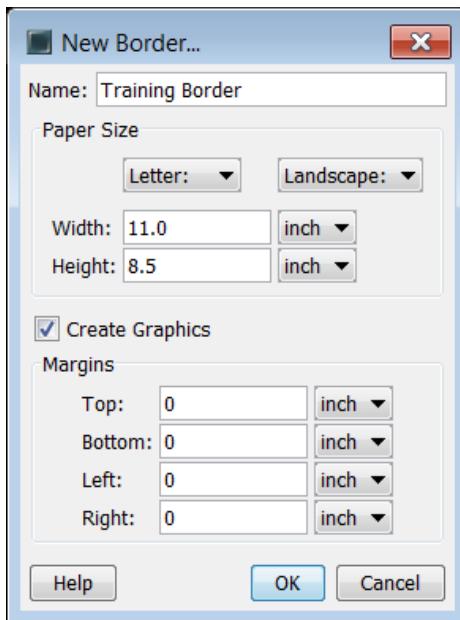
Borders

Borders provide a template for printed diagrams according to company standards. They often contain a title block area where information such as Name and Owner can be specified. A border also defines the page size used when printing a diagram. Borders are managed within a library, which means you can set up different libraries for different customers, vehicles, craft etc. Borders also use the grid preferences set up in project (see earlier).

Creating a Border

In Capital Symbol, choose **File / New Library** and choose to create a new library for **borders**.

Once the library has been created (or opened) you can now create a new border by right clicking on the library name and selecting **New Border**.



From here choose the details of the border – these can be modified later if needed:

Paper Size

From here choose the paper size and type (landscape/portrait) that the diagram should be printed onto. If you choose custom you must then specify the width and height that the paper should be

Create Graphics

This allows you to set up the margin details and controls whether the outside extent of the border is represented in Capital Symbol.

Margins

The margin will prevent any automated processes placing entities within the specified area.

Note

The co-ordinates of the center of the border are 0,0.
Ensure that your border margins reflect the margin settings of your default printer.



Above is an example border.

Dotted Rectangle: the dotted outer line is the paper size and will not be seen in Capital HarnessXC.

Rectangle: is the outside extent of the border taking into consideration of the margin details

Inner Grey Rectangle: the inside rectangle is the area where the zone references are displayed about and the area of the design that is editable.

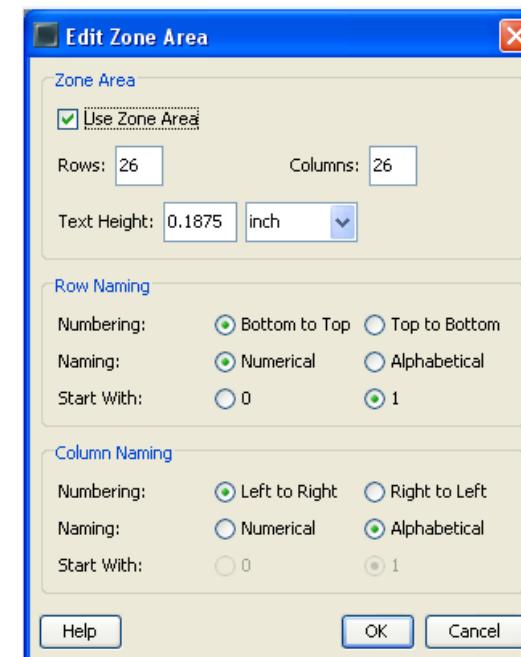
Using the standard drawing tools as introduced earlier, you can create a title block as required.

Adding Zones to a Border Symbol

Zones can be added to enhance the border appearance. Border zones provide a grid.

After you have defined the border size, you can add zone definitions.

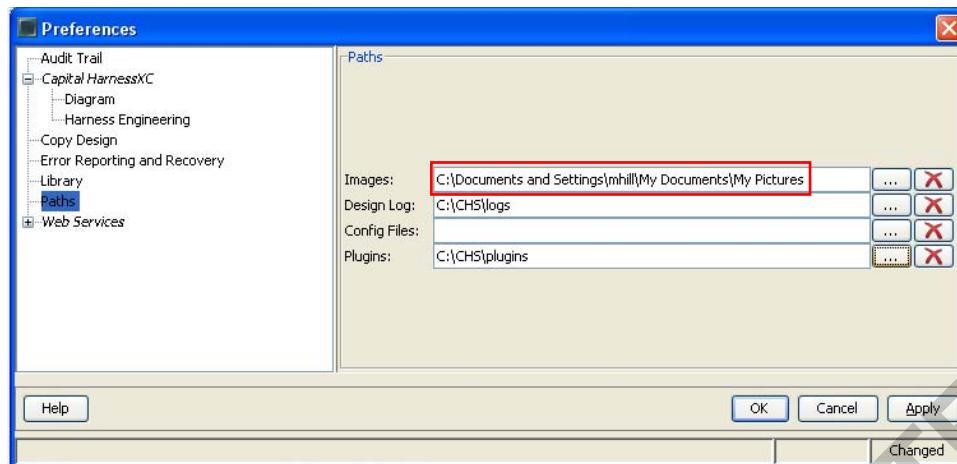
- ⇒ **Edit / Modify Zone Area** opens the edit zone dialog. There are 3 sections to this dialog, **Zone Area**, **Row Naming** and **Column Naming**.
- ⇒ **Zone Area:** define the number of columns and rows desired. Also define the text height. This is the zone indicator that is placed between the outline of the border and the internal drawing area.
- ⇒ **Naming Area:** both rows and columns can be named with alphabetical or numeric characters. You can choose any combination to suit your needs.
 - Choose an ordering scheme for both rows and characters.
 - If you use numeric for column or rows, select whether you want to start at 0 or 1.
- ⇒ Click **OK** to complete. The border view will show the zones defined.



Inserting Images

It is possible to insert an image onto the drawing.

The path to identify the location of the symbols must be defined before an image can be inserted. This is done in Capital Project under the system preferences **System/Preferences/Paths**



- ⇒ Select **System**
- ⇒ Select **Preferences**
- ⇒ Define location for images

To insert the image:

- ⇒ Select **Add / Image** or choose the icon.
- ⇒ Draw the area that the image will be displayed
- ⇒ The image will then appear

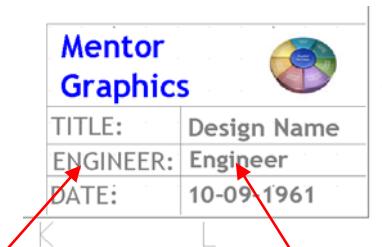
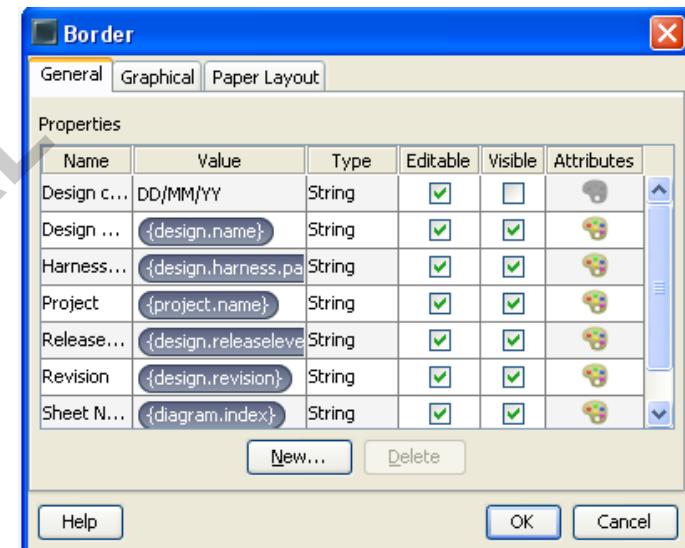
Note

An image can be inserted into any of the graphical tools.

Properties

By right clicking on the border in the tree area, you can choose to add properties to the border.

These properties can be modified when used in Capital HarnessXC for example, the name of the design, the date, the revision level etc. A tagged property value can be translated using the language dictionary should users wish to change the language selection on a diagram.



Text added to the title block

Property value added to the title block

Intelligent Text Placement

It is also possible to add placeholders for specific project, design and diagram information to the border symbol. The value field will be automatically populated with more meaningful information once the border is placed in the diagram.

The property is added to the border symbol by right clicking the border reference in the tree browser and selecting properties. A name and value is given to the property. The value for the placeholder string needs to be entered between braces - { } and must also follow a specific convention:

Property reference	Placeholder string
Name of design to which diagram belongs	{design.name}
Release level of the design	{design.releaselevel}
Revision of the design	{design.revision}
Long description for the design	{design.description}
Short description for the design	{design.shortdescription}
Number of diagrams that exist for the design	{design.diagramcount}
Diagram index of the diagram	{diagram.index}
Name of the diagram	{diagram.name}
Name of the project	{project.name}
Description of the project	{project.description}
Part number of design	{design.harness.partnumber}

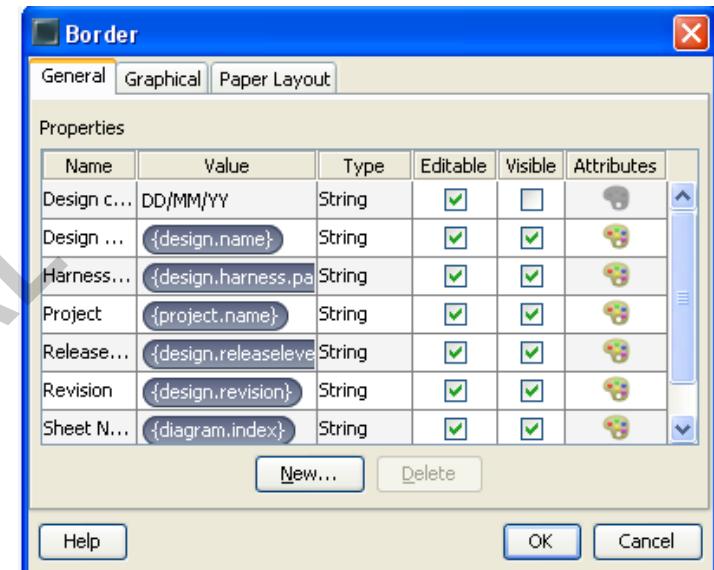
The {diagram.index} and {design.diagramcount} properties can be combined to indicate the order held of the currently open diagram out of the total number of diagrams existing for that particular design i.e.:
{diagram.index} of {design.diagramcount} : on the 2nd sheet of 5 design diagrams would read: 2 of 5

Once the property and value have been created the value can now be placed on the title block.

Example: Add property of title to reflect the harness part number:

When adding this border to the harness diagram the design.harness.partnumber value will take on the value assigned to the harness diagram.

Also notice that the value for the intelligent text is highlighted, indicating that a translation of this text is possible.



Exercise 3

In this exercise you will create a new border library and create a new border.

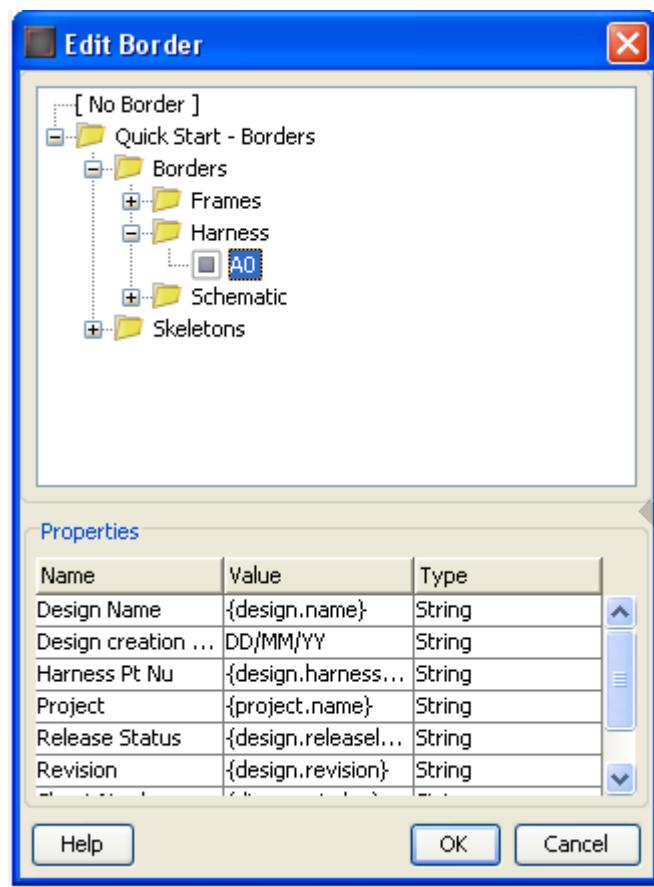
→ Create the new border as explained in the exercise worksheet

Placing the Border in XC

A border can be added to the diagram at any point during the diagram creation.

- ⇒ Select **Graphics / Border**
- ⇒ Choose desired border from the list
- ⇒ System will present **properties** associated to the selected border

- ⇒ Where the value shows something in brackets {} the system will update this information automatically
- ⇒ Where the value shows plain text the user can edit this information
- ⇒ On pressing **OK** the system will place the border



Chapter 7

Design Changes

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Create Design Revisions

Revision Management of Project Designs is accomplished by creating Revisions of Designs within the Project. Each Design Revision can be accessed (with appropriate Release Level) within the Project and has the following information:

- ⇒ **Name** (cannot be edited)
- ⇒ **Part Number** (any alpha-numeric string)
- ⇒ **Design Revision** (any alpha-numeric string)
- ⇒ **Short Description** (any alpha-numeric string)
- ⇒ Description – Long Description field
- ⇒ Release Level

Note

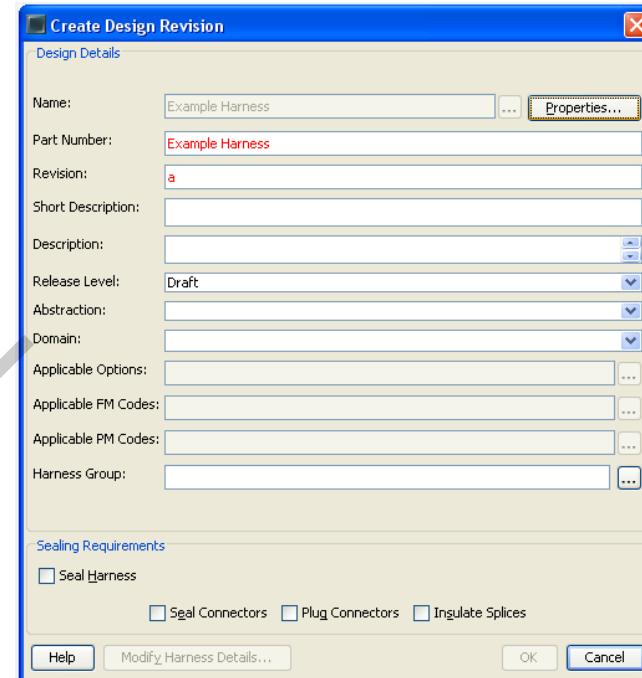
The combination of **Name**, **Revision** and **Short Description** must be unique when creating a design revision.

The combination of Name, Revision and Short Description must be unique within a project.

The combination of Part Number and Revision must be unique across the database.

To create a Design Revision:

- ⇒ Right-mouse click on the **Design Name** in the Project Browser.
- ⇒ Select **Create / Design Revision** from the menu:
- ⇒ The **Create Design Revision** form will be displayed:



- ⇒ Enter the new **Part Number** or **Revision** value (or both).
- ⇒ Select the **Release Level** for the new Design Revision.
- ⇒ Select **OK**. A new Design will be created in the Project Browser, and all of the data and graphical information associated with the original Design will be copied to the new Design Revision.

Note

Design revisions keep their history. This means that you can see the state of the design at any point in time.

Copying a design breaks the link back to the original design and all its objects. Copying is used if you are trying to save time and start a new design that is similar to the original.

Any derivatives must be edited to create a unique harness design.

Design Change Tracking

Engineering change orders (ECO) help to record design change information and identify the impacted designs.

Change orders can be managed from within the Capital Project or Logic tool, assuming users have the appropriate user permission to handle ECOs. Each change order is held within a category folder.

To create an engineering change order:

- ⇒ Open the project
- ⇒ Double click engineering change orders (in project) or go to Edit/Engineering Change Order (Harness XC)
- ⇒ Click  to add a new change order category
- ⇒ Define a name for the category
- ⇒ Within the new category click  to create a new ECO
- ⇒ Rename the ECO as required
- ⇒ Assign descriptions and properties as required
- ⇒ Add notes for the ECO as required
- ⇒ Add the impacted designs

Adding impacted designs:

- ⇒ Click 'Add new associations' in the impacted designs area
- ⇒ Select the impacted designs (use the filter if necessary)
- ⇒ Click OK to close the ECO dialog

To Delete an ECO:

- ⇒ Open the ECO dialog
- ⇒ Highlight the relevant ECO
- ⇒ Remove all designs from the impacted designs area
- ⇒ Click  to remove the ECO from the highlighted category

Categories that no longer contain an ECO will automatically be removed upon closure of the dialog

Exercise 1:

In this exercise, you will create Design Revisions.

Using information shown in exercise 1, create the revision and ECO

Multiple Location Components

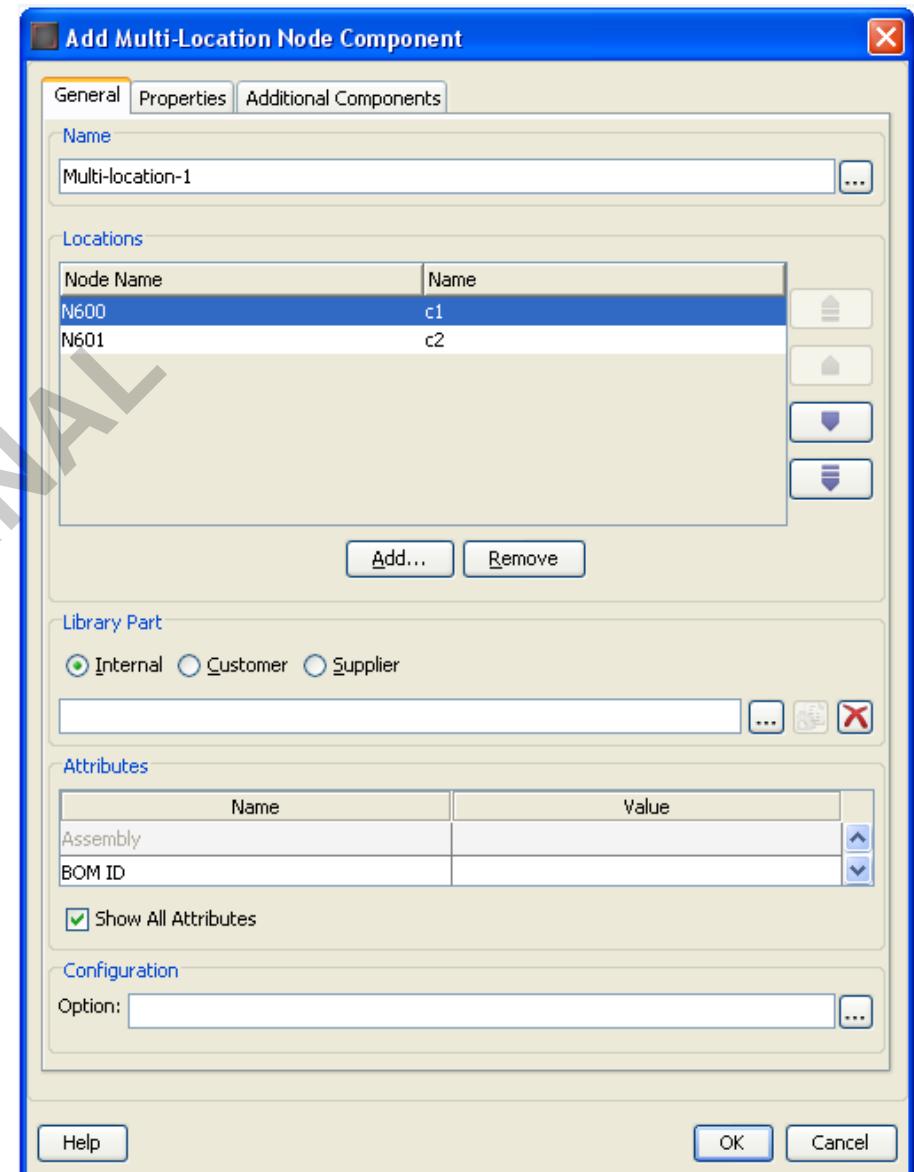
Multiple location components are components which span more than one node, cavity or wire end.

- ⇒ An example of a component spanning several nodes would be a bracket that may hold several connectors together
- ⇒ An example of a component spanning several cavities maybe a busbar which electrically connects a wire with several cavities
- ⇒ An example of a component spanning several wire ends is a label

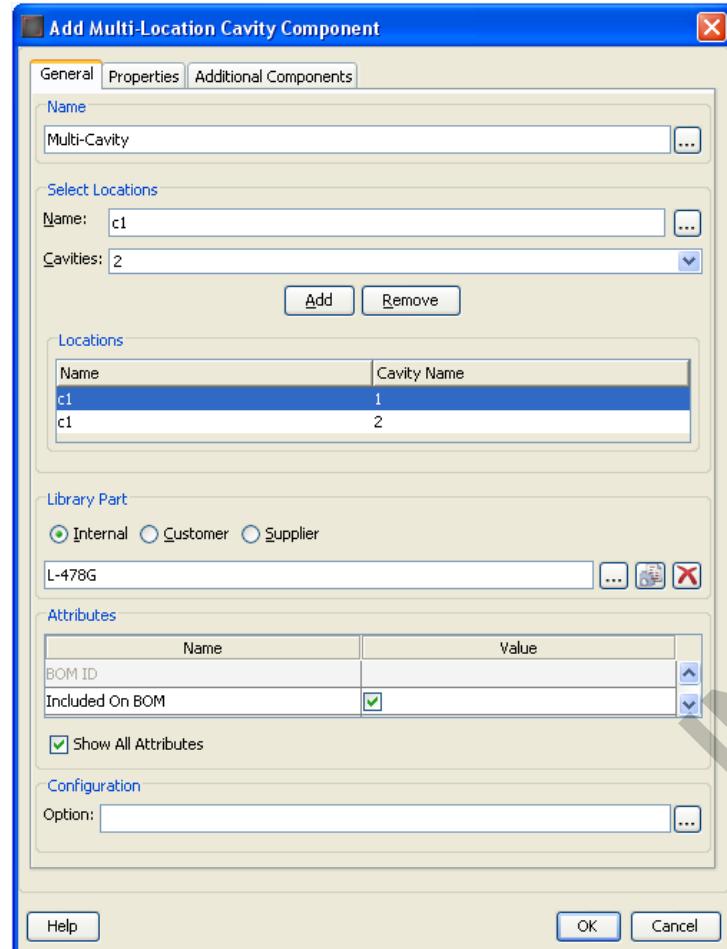
Select **Add / Multi Location** / and select from the list which type of multi location is required.

Multi Node Location

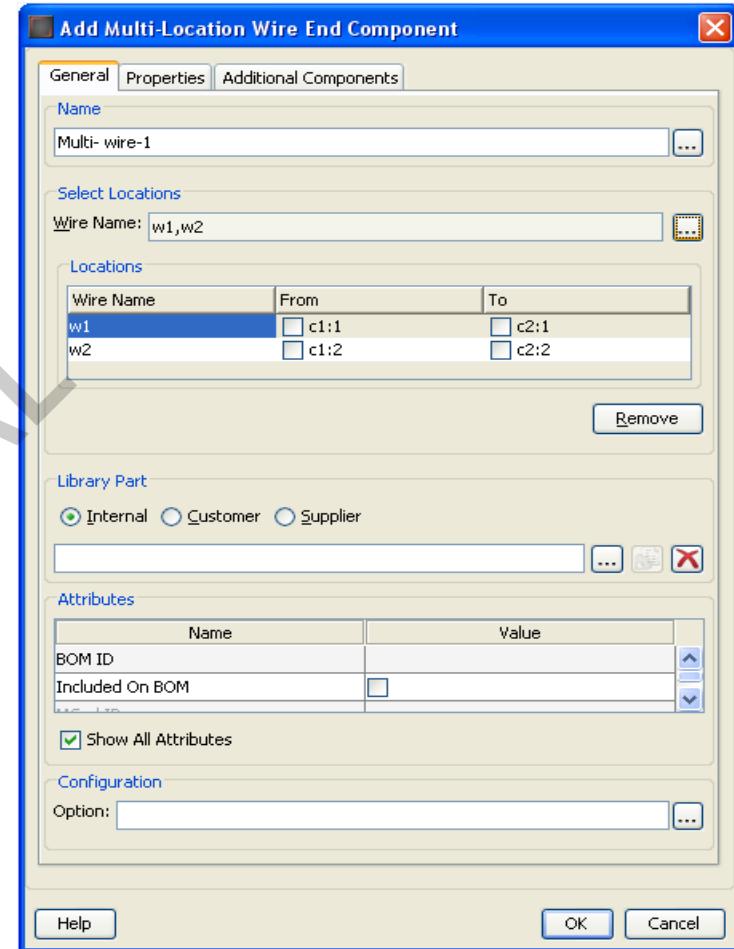
- ⇒ Select a **Node** from the pull down list
- ⇒ Add the **node** to the list
- ⇒ **Repeat** as necessary
- ⇒ **Select OK**



Multi Cavity Location



Multi Wire End Location



- ⇒ Select the **Node** from the list
- ⇒ Select the **cavity** from the pull down list
- ⇒ Add the **cavity** to the list
- ⇒ **Repeat** as necessary
- ⇒ **Select OK**

- ⇒ Select the **Wire** from the list
- ⇒ Select which **wire end**
- ⇒ **Repeat** as necessary
- ⇒ **Select OK**

Other Functions

Wire Length Changes

Often it is necessary to alter the length of a wire for manufacturing purposes.

Wire length changes are made:

- ⇒ To properly define link wires (both wire ends in 1 connector)
- ⇒ For re-routed wires in a balanced splice
- ⇒ For wires looped back and taped-in
- ⇒ To adjust tight or slack wires

There are two types of length changes:

- ⇒ Absolute: calculated length is replaced
- ⇒ Offset: calculated length is increased or decreased

Using the **Wire Attributes** tab the user defines the length change type and value as required.

Attribute Table	
Name	Value
Included On Cut Chart	<input checked="" type="checkbox"/>
Length Change Type	Offset
Length Change Value	100
MCad ID	

Configuration

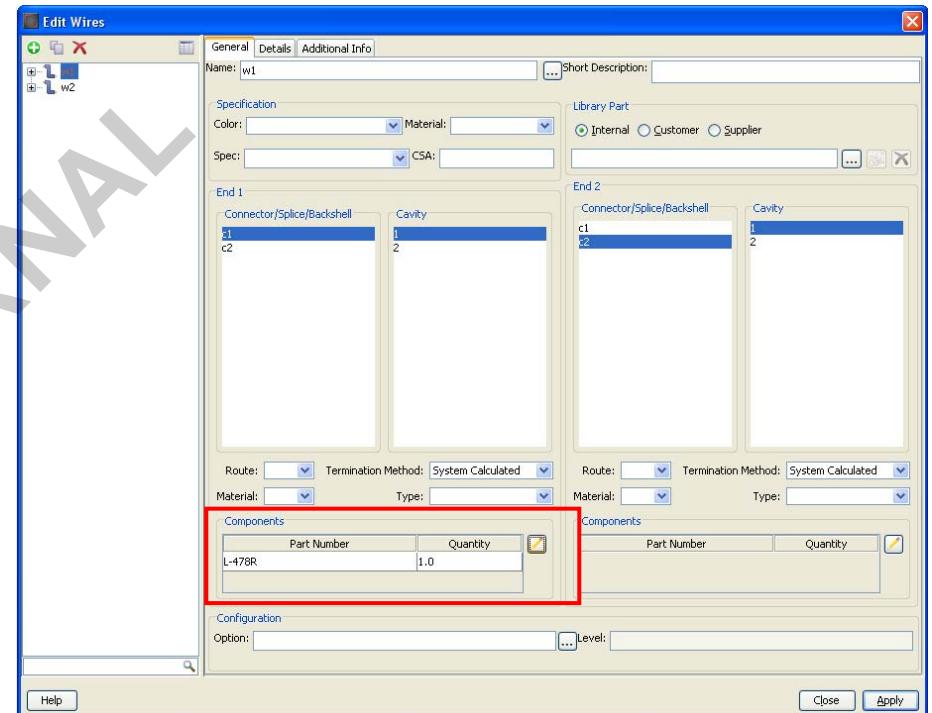
Module: _____

Level: _____

Option: O1

Adding extra components to wire ends

Wires have their own wire end details form that is used to define the wire end locations and in addition can also be used to define any extra components that need to be assigned to one or other of the wire ends such as a label. The form can be found within the Edit wires form under the General tab.



To add a component to a particular wire end:

- ⇒ Open the edit wire form and select the wire in question
- ⇒ Select the General tab
- ⇒ Identify the wire end to which you wish to assign the extra component
- ⇒ Click the edit wire end components icon to add a component and select a library part number from the library form that pops up
- ⇒ Click **Insert** and the component is now assigned to the wire end

Adding Extra Components

There are components that are not classified with their own group type such as identity labels, brackets, covers, mechanical components etc.

To add extra components:

- ⇒ Select **Add / Other Component**
- ⇒ Navigate to the position on the harness where the component is to be added and click on the location using the **NPU** if the location is on a pre-existing bundle
- ⇒ Select the library part required and click **OK**

Centre Strip Wires

Where both ends of a wire are located it may be necessary to electrically connect the wire to one or more splices. A centre strip (window wire) in a wire is a way of making this connection. By stripping back the insulation and exposing the copper stranding this copper stranding can then be connected to a splice. Each wire can have multiple centre strips.

To assign a centre strip on a wire:

- ⇒ Select **Edit Wire**
- ⇒ Select **Additional Info** tab
- ⇒ Select which splice is to have the centre strip

Moving Components

It is possible to move a component (not a connector) either along the same branch or to a different branch by using the NPU.

- ⇒ **Right Click** on the selected component
- ⇒ Select **Move**
- ⇒ Using the cursor, locate along the bundle for the new position of the component
- ⇒ Use the **NPU** to determine the exact location

Note

It is possible to move multiple splices to the same location.

Moving Connectors

It is possible to move a connector to an empty structure node.

- ⇒ Select the connector you wish to move
- ⇒ **Right Click** and select **Move**
- ⇒ Select the node you wish to move the connector to. The connector and all its wires will now move to the new location. It will be necessary to delete the old node.

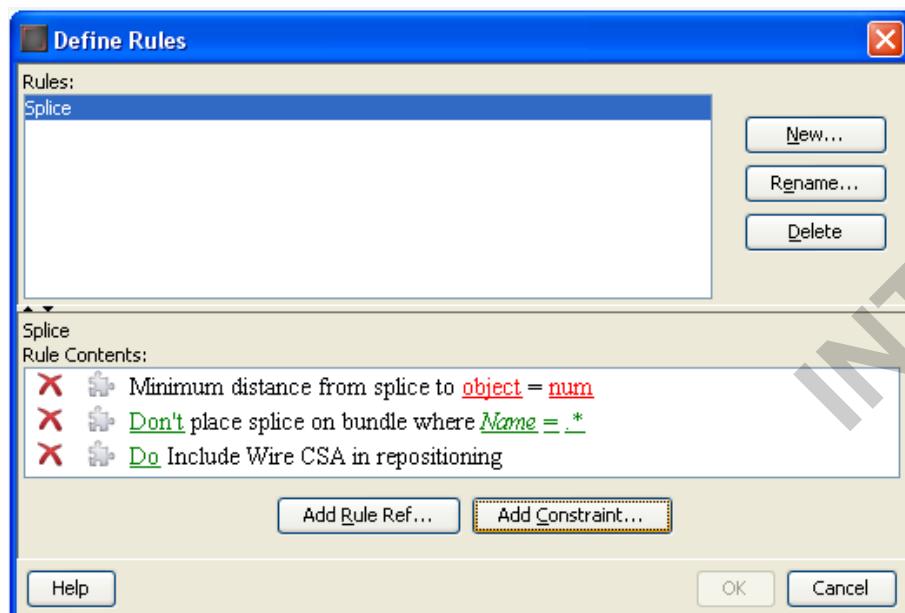
Splice Repositioning

Splices can be repositioned according to a number of constraints set against the harness diagram. When placing or editing a splice, users must check the 'repositionable' box to ensure that when constraints are applied, the splices are repositioned.

The splice repositioning constraints use bundles, separation from other objects and wire csa information in order to optimally reposition a splice:

To add a constraint:

- ⇒ Click Edit/Rules/New
- ⇒ Add constraint and select the desired constraint



To re-position a splice:

- ⇒ Select the splices to be repositioned (if no splices are selected all will be repositioned)
- ⇒ Select Actions/Processing/Optimize Splice Positions

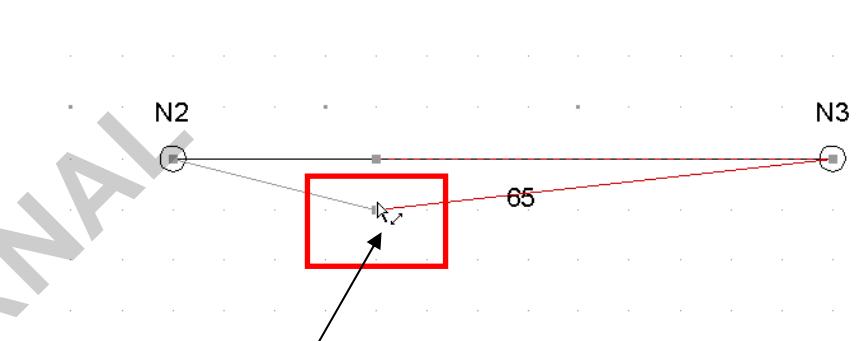
Harness Engineering will also apply splice positioning rules if the check box is selected

Grip points on Bundles

Grip points allow the bundle to bend mid-branch.

To add a Grip Point

- ⇒ **Right Click** on the bundle
- ⇒ **Select Grip Point / Add**
- ⇒ Using the cursor locate the required grip point



- ⇒ With the grip point selected drag to create the bend

To Delete a Grip Point

- ⇒ **Right Click** on the bundle segment with the unwanted grip point
- ⇒ **Select Grip Point / Delete**
- ⇒ Click on the grip point to be deleted
- ⇒ The bundle is automatically straightened by the system

Note

To select the complete bundle i.e. all the segments, hold the CTRL key down whilst selecting the segments or select the bundle from the tree browser.

Harness Rotation

To re-position bundles on a harness, typically users would click on a node or grip point and use this to re-locate the bundle, however, this does not take into account the positioning of other bundles that may be attached to the bundle that is being moved. Users may prefer to use the Rotate function to rotate the bundle and any components or bundles associated to it in one go.

To perform the **Rotate** function:

- ⇒ Open **Layout / Rotate section of harness**
- ⇒ Select a point on the bundle around which you wish to insert a rotation (this would typically be a node or waypoint)
- ⇒ Select a second point on the bundle close to the waypoint you wish to rotate around – this will denote which section of the harness to be rotated i.e. to the left of point 1 or to the right of point1
- ⇒ Begin the rotation

Points to consider:

Users may do a full rotation of the selected section or a rigid rotation whereby only the main bundle is rotated and any attached component or bundles will retain their original position.

To switch between rigid or full rotation, use the **T** key

When the rotation function is initiated it is assumed that the angle of rotation will be measured from the current angle of the section selected. i.e. if there is a selected section at 45° and you wish to rotate it 80° you will need to rotate until the tool indicates 35°. If users were to press the CTRL key the angles are displayed with reference to the global horizontal axis, in which case rotating the angle to 80° is done by rotating until users see the increase to 80°.

Exercise

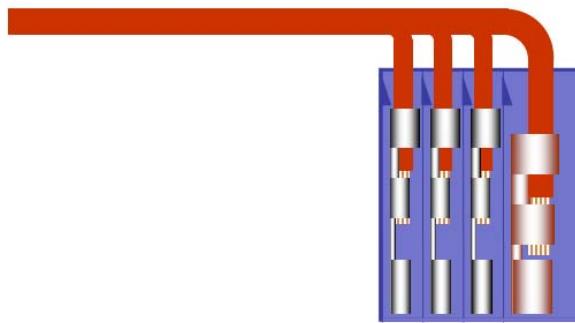
Users may try the rotation function now using the steps described above to rotate a section of the XY-P017 harness Revision B.

Routed Connectors

Overview

When a connector is to be orientated in such a way that a cavity position will affect the length of the incoming wire, it is necessary to establish the dress/route. The route applies to that particular dressing / orientation or that particular connector.

Simple Route



The above example shows one particular route for the wires. The wire going to the first cavity will be shorter than the wire going to the last cavity. If a different route were used then the wire lengths would be different.

The route of the connector could be determined by the following:

- ⇒ Manufacturing may need the wires / branches to be routed a particular way for ease of build
- ⇒ It may be necessary to use different routes due to how the harness is to be fitted

To add the route to the connector:

- ⇒ **Insert** the connector
- ⇒ Go to the **Routes** tab
- ⇒ **Select** the correct route from the list

Note

The system will only present the routes that have been set up in Capital Library.

Setting up the Add-ons for the Connector

A connector can have more than one route or be used with no route. If the latter is implemented then the Add-on or Knock-off details would then be taken from the Component's Base Details.

The Tab 'Dressed Routes' allows entry of the add-ons associated with each individual cavity depending on the route used.

The screenshot shows a software interface titled 'Component Details'. The 'Dressed Routes' tab is selected. A table lists a single route entry:

Route	Cavity	Add On	Knock Off
1	1	5.000000	0.000000

Below the table are several buttons: Delete Route..., Copy Route..., New, Update, Reset, and Delete. At the bottom, there are input fields for Route (1), Cavity (1), Add On (5.000000), Group Name (Connector), Knock Off (0.000000), and a note field.

Note

Routes can be applied to any connector.

Select by Attribute/Property

It is possible to search for objects that have the same attribute or property value assigned to them.

The objects found will be highlighted in red on the diagram:

- ⇒ Select Edit/Select by Attribute/Property



Design Comparison

A comparison of designs can be performed in both Capital Project and/or Harness XC.

To compare a design with any other design within the same or any other project, users can run the **Compare Designs** function.

With the first design currently active:

- ⇒ Select **Tools / Compare Designs**
- ⇒ Select the project and relevant design you wish to make a comparison with
- ⇒ Select the required comparison attributes
- ⇒ Click Compare and view the results in the differences area of the dialog
- ⇒ The show differences button will highlight the differences on the open diagram(s)

Exercise 2-3:

Create the changes as described in the worksheet and perform a design comparison between revisions A and B of your XY-P017 design

Copy Designs

Copy a Design within a Project

It is possible to copy a design or a project within the current project or to a different project.

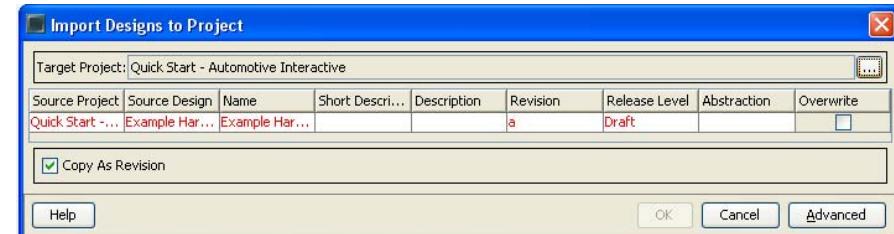
- ⇒ Right Click on the design
- ⇒ Select **Copy / Within Current Project**

A design has to be unique within a project and therefore the part number must change.

Advanced Copy of a Design

This will allow users to copy a design into a different project.

- ⇒ Right Click on the design
- ⇒ Select **Copy / Advanced**
- ⇒ Select the **target project** to **copy** the design(s) to
- ⇒ Click Next



- ⇒ Select the advanced button to define further copy details:

- ⇒ **Design Details.** The combination of part number, revision and short description must be unique at system level. Selecting to **Copy As Revision** will retain the name and keep the connection with the source design

Other copy features within the tabbed form are not required.

- ⇒ Click Back to return to the original dialog

INTERNAL

Chapter 8

Composite Harnesses

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Composite / Derivative Harness Relationships

Composite Families Explained

Composite Harness

(Also referred to as the family or master harness). This harness will not typically be built.

Derivative Harness

(Also referred to as the child or variation). This is a real harness in all aspects. Because it is similar to one or more other harnesses, it is managed via a composite entity.

Option

The relationship between the composite and its derivatives is described in terms of Options. An option is a **label** with which to link wires and components to derivatives.

Table 1, Derivatives / Options

Composite Harness: CompHarn, Issue D

Options > ↳ Derivatives	Option LHD	Option RHD	Option ABS	Option Airbag	All Derivatives
Der1Harn, Issue C	✓			✓	✓
Der2Harn, Issue C		✓	✓	✓	✓
Der3Harn, Issue D	✓		✓		✓

Note

Within a composite harness options are usually associated with wires, but can also be registered against branches, components and insulations.

Components and attached branches will be deleted if no wires are present at or pass through the node.

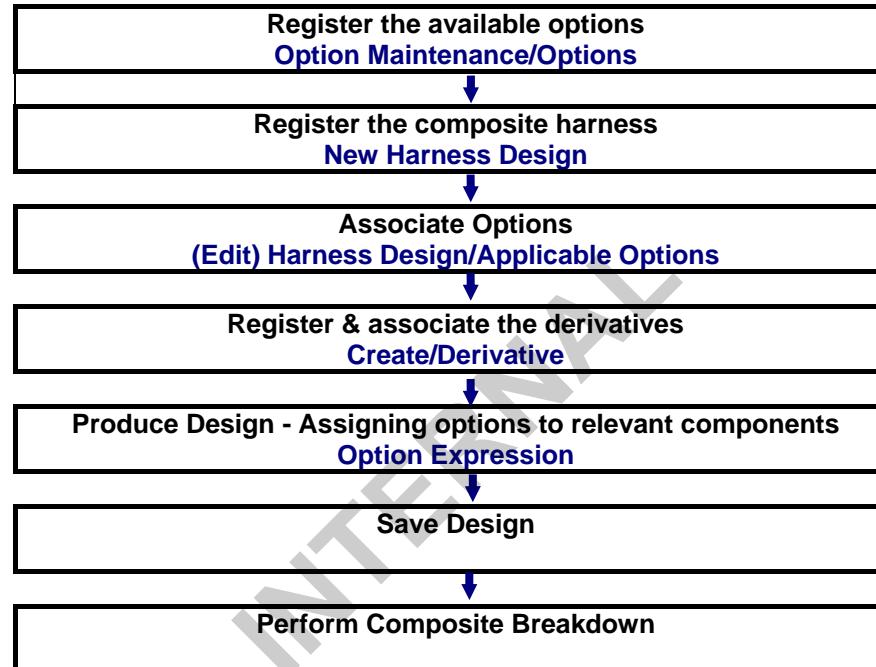
Table 2, Wires / Options

WIRE	COL	MAT	CSA	Option
1	B	TWC	0.5	LHD
2	W	TWC	0.5	LHD
3	LG	TWC	0.35	RHD
4	R	TWC	0.35	ABS
5	Y	TWC	2.0	AIRBAG
6	U	TWC	1.5	ALL

In the example above, wire 6 belongs to Option ALL and therefore will appear in all derivatives. All other wires are included in derivatives containing the corresponding option.

Composite Input Procedure

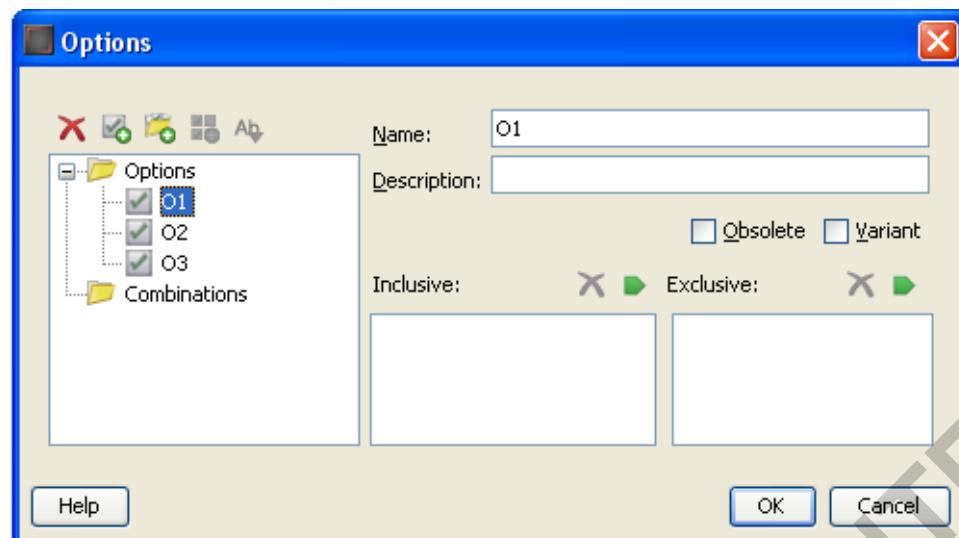
Inputting & Processing a Composite Harness requires the following steps:-



Creating a Composite

Creating Options

Options may be created in Capital Project or Capital Harness XC.



To create an option:

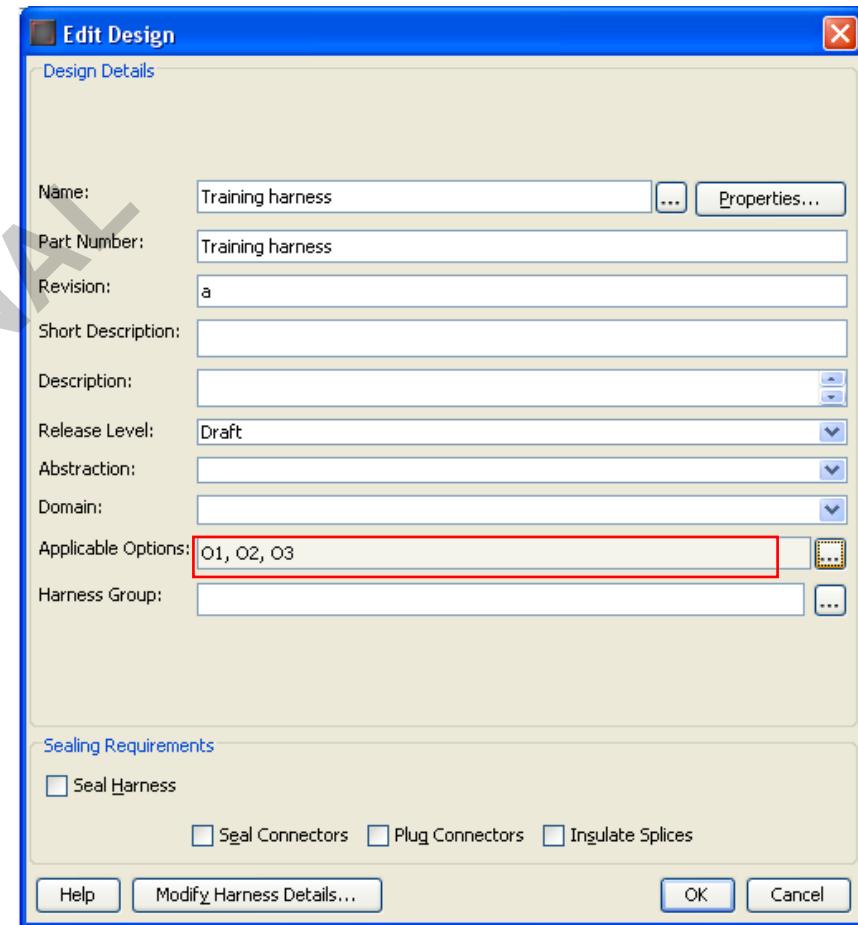
- ⇒ Click the create new option icon
- ⇒ Define a name and description for the option
- ⇒ Repeat the steps to create a new option
- ⇒ Use the icon to alphabetically sort options

To define option inclusivity or exclusivity:

- ⇒ Highlight the option in question
- ⇒ Click the green arrow above the inclusive or exclusive table
- ⇒ Select the option(s) you wish to include/exclude with the currently highlighted option and click **Add** to move it to the selected area
- ⇒ Click **OK**

Options Associated with the Composite

The relevant options from the project must be associated with the composite design if they have been created in Capital Project.

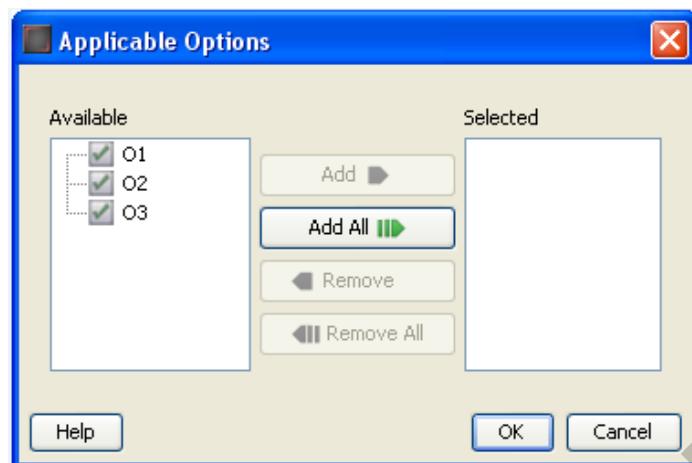


Only options related to the composite need to be associated.

Creating the Derivatives

To create the derivatives:

- ⇒ **In Capital Project** Right Click on the composite design
- ⇒ Select **Create / Derivative**
- ⇒ **Enter** the derivative harness details
- ⇒ **Assign** the applicable options for that derivative via the **Applicable Options** field



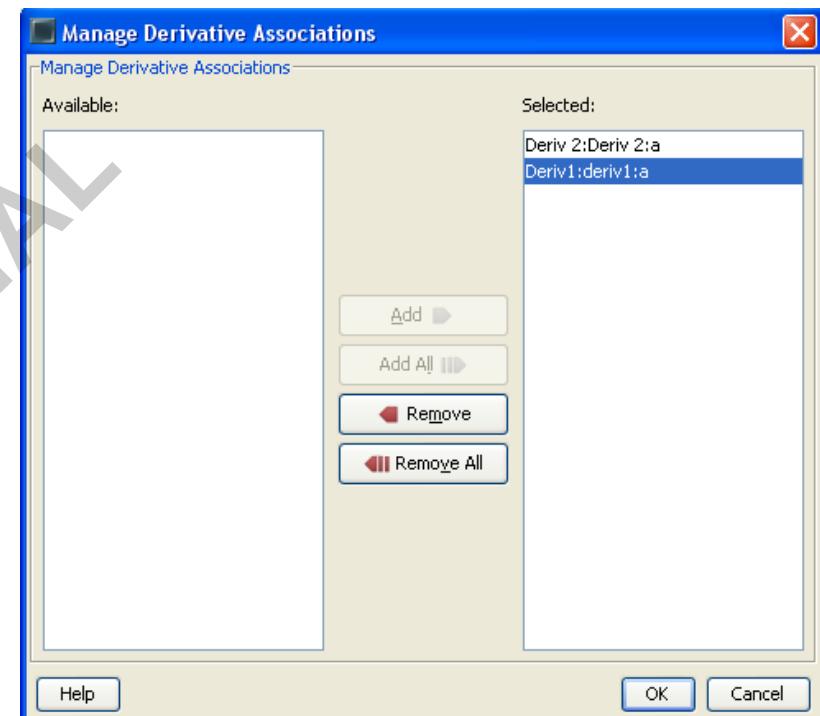
Note

Only the options associated with the composite will be available.

Managing Derivative Associations

It may be necessary to add a previously created design or remove an existing derivative from the composite design. To do this in Capital Project:

- ⇒ **Right Click** on the composite design
- ⇒ Select **Manage / Derivative Associations**
- ⇒ **Choose** the required derivative harnesses for that composite



Note

Only designs with one or more options common to the selected composite will be available for selection.
This is the only place derivatives can be removed from a composite. If a derivative is deleted from the project tree browser it will be removed from the system completely.

Options against Components

Wires are the most important harness property to consider when determining which aspects of the composite harness should be assigned to which options.

By detailing which option a wire has, the system is able to determine which derivative harnesses that wire should appear on.

When the wires are removed to form a derivative harness, any unpopulated connectors can be automatically removed, as can any branches or nodes containing no wires (during the breakdown process users must ensure that the 'Evaluate blank option expression as All' box is unselected).

As well as being able to place options against a wire, it is also possible to have options against components, additional components, branches, insulation and symbols where the occurrence of that object differs from derivative to derivative.

Options are placed using the ellipses button for the Option Expression section found on every dialog.

Exercise 1:

Create derivatives:

Using information shown in exercise 1 on the exercise worksheet, convert the harness into a composite harness; create the options and derivatives as required. Add the required option to the wires.

Inserting Composite tables:

The harness diagram may display a composite table, detailing the derivatives and options. The assignation of the table to the diagram is dependent upon the style settings you have assigned (see later chapter for style sets), however there are a couple of points you should check before applying the style:

- 1) Ensure that the table has been defined and is visible in the style set (**Edit / Style / Style Set / Harness** / (select style set) / **Design / Decorations / Tables / Harness Composite table**)
- 2) Ensure that you have updated the border which by default is used to set the wire table and other tables. (**Actions / Update / Border**)

To insert the table on the drawing:

- ⇒ **Edit / select all**
- ⇒ **Edit / Apply style**

Options are displayed on the table in alphabetic order, if you wish to re-order the option positions:

- ⇒ Close the design
- ⇒ Right Click the design **Edit**
- ⇒ Click the button next to the Applicable options field
- ⇒ Highlight the selected options and use the up/down arrow to re-order
- ⇒ Click **ok** to close the dialogue
- ⇒ Re-open the diagram and the options will now be re-ordered

Composite BOM Table

A BOM table may be placed on the composite diagram without users having to first run the composite breakdown.

The Composite BOM table may include a listing of the total number of wires, connectors etc, as well as individual component information held on the diagram. Again, the assignation of the table to the diagram is dependent on the chosen style set

To apply the settings for the composite BOM table:

- ⇒ Open Capital Project
- ⇒ Select **preferences**
- ⇒ Select Composite BOM and /or BOM id naming
- ⇒ Define the settings you require for the table
- ⇒ Click OK and close

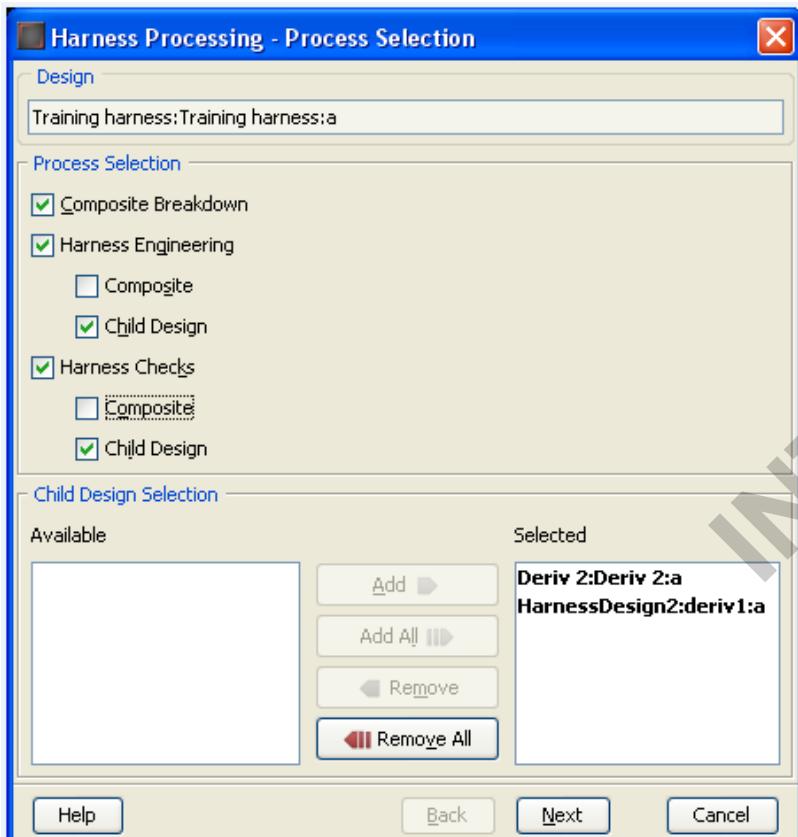
To assign the table to the Composite diagram:

- ⇒ Ensure the border is set and updated (**Actions / Update / Border**)
- ⇒ Ensure that the table is made visible in the style set – ((**Edit / Style / Style Set / Harness / (select style set) / Design / Decorations / Tables / Harness Composite BOM table**)
- ⇒ Select all diagram contents
- ⇒ Select **Edit / Style / Apply Style**

Processing the Composite Harness

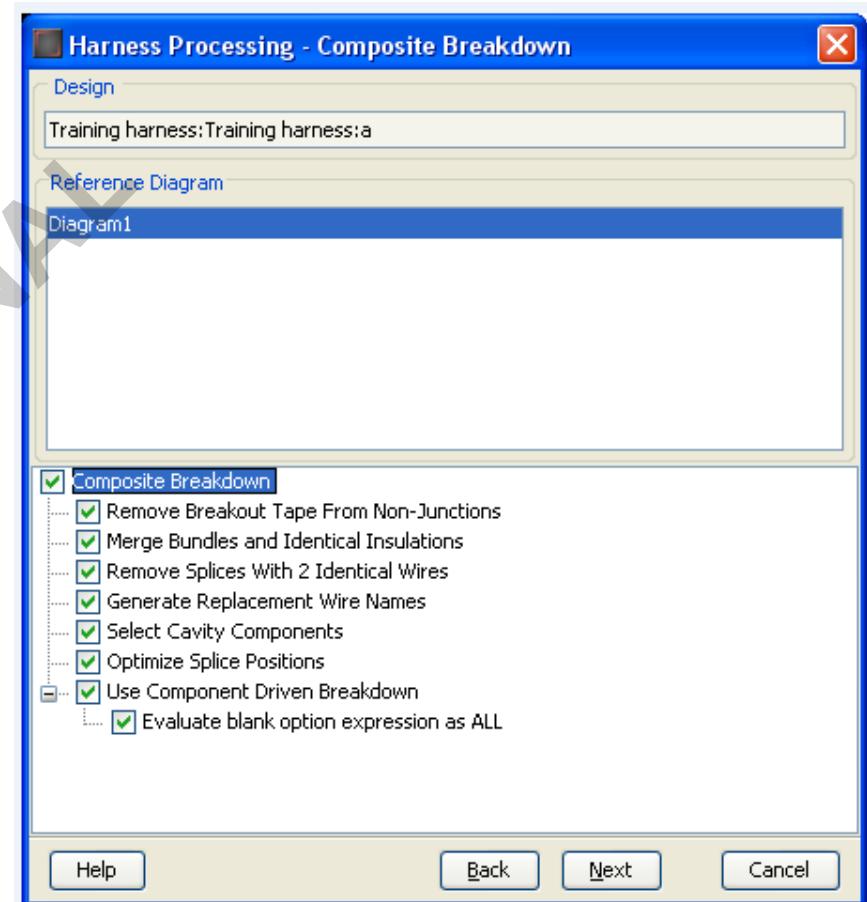
Processing a composite is initiated in the same manner as if it were a single harness.

⇒ Select Tools / Harness Processing...



- ⇒ The system presents all processes
- ⇒ The system presents all available derivatives for that composite
- ⇒ For Engineering Calculations and Harness Checks, choose to run the process on the composite harness or the derivative harnesses

⇒ Selecting the **Composite Breakdown** automatically produces the derivative harness details, distributing all of the relevant data using the option expressions.



After the distribution of all the data has occurred, the system can perform various calculations on each derivative. The following explains each of the calculations:

- ⇒ **Remove breakout tape from non-junctions?** Although a composite harness may have a junction with 3 or more branches, a derivative may have only 2 due to component distribution. In this case it may not be necessary to use breakout tape. This option requests that the system remove the breakout tape in this situation.
- ⇒ **Merge Bundles and Identical Insulations?** When junctions have been removed leaving 2 bundles on a derivative, the system will merge the 2 bundles and merge any identical insulations into one continuous run
- ⇒ **Remove splices with 2 identical wires?** Removes any splice where only 2 wires remain and the wires are identical in color, material and spec. The splice will be replaced with one single wire with that color, material and spec.
- ⇒ **Generate replacement wire names?** Allows the user to specify the prefix to the wire names that replace the splices (see previous option). The prefix will be followed by a numerical value (n+1)
- ⇒ **Select Cavity Components?** This is the same as with Harness Engineering (chapter 5) but for each derivative, overriding any automatic selections made on the composite (if it had been performed at composite level)
- ⇒ **Optimize splice positions?** uses the splice position definition set within rules in the XC tool to define where the splices are to be located on each derivative
- ⇒ Use Component driven Breakdown uses option information assigned to components
- ⇒ Evaluate blank option expression as ALL when checked assumes objects without an option will have option 'All' – users should be aware that checking this option will leave unused connectors and bundles on the derivatives even when no wires exist in the connector

Once complete users may open each derivative diagram individually.

Note

As the Composite Breakdown is automatically producing the derivative harnesses, they also should have the 'Harness Engineering' run against them.

De-Populate Redundant Objects

Often when importing data from another source, e.g. Capital Integrator, there can be data on the composite harness that doesn't exist on any derivative and is therefore redundant.

This will be removed during composite breakdown, alternatively you could choose to remove these redundant objects from the composite.

Select **Actions / De-populate redundant objects** and all redundant objects will be removed.

Exercise 2:

Run the composite breakdown and harness engineering and harness checks per derivative. View the results and fix any problems that have occurred.

Use of Logical Operators

- ⇒ Logical operators can be used against Options in branch configuration, branch insulation and wires. They are:

&& (and)

|| (or)

! (not)

(and) (parentheses)

Table 1, Derivatives / Options					
Composite Harness: CompHarn, Issue D					
Options > Derivatives	Option LHD	Option RHD	Option ABS	Option AIRBAG	All Derivatives
DERV001, Issue C	✓			✓	✓
DERV002, Issue C		✓	✓	✓	✓
DERV003, Issue D	✓		✓	✓	✓

Table 3, Logical Operators	
WIRE	Option
1	LHD
2	LHD RHD
3	LHD&&AIRBAG
4	LHD&& ABS
5	AIRBAG !ABS
6	AIRBAG&&ABS&&!RHD

In the table above a number of logical operations has been placed. Following these rules the wires will be placed in derivatives as follows:

Wire 1 – All derivatives containing the LHD option.
DERV001 and Derv003

Wire 2 – All derivatives containing the LHD **OR** RHD option
DERV001, Derv002 and Derv003

Wire 3 – All derivatives containing LHD **AND** AIRBAG options
DERV001 and Derv003

Wire 4 – All derivatives containing LHD **AND NOT** ABS
DERV001

Wire 5 – All derivatives containing AIRBAG **OR NOT** ABS
DERV001

Wire 6 – All derivatives containing AIRBAG **AND ABS AND NOT RHD**
DERV003

The exercises over the next few pages are examples of logical operators.

Note

The system works out the formula using a standard mathematical algorithm.

Exercise 3:

Discuss Logical Operators.

Create Composite:

Using information shown in exercise 2 on the exercise worksheet, create the options, composite and derivatives as required. Create the harness.

Variant positioned connectors and splices

A connector or splice may need to be positioned at different locations on a harness, depending on the derivative design, for example, in the case of a connector a headlamp switch may be in 1 location on a left hand drive vehicle and at another location on a right hand drive vehicle. To address this need, the XC tool provides a variant position connectors (vpc) function.

To instantiate variant connector positions:

- ⇒ Insert a new bundle for the alternate connector position
- ⇒ Edit the connector you wish to variably place
- ⇒ Select the **Variant positions** tab
- ⇒ Click **New** and select the node you wish to place the variant connector at
- ⇒ Select the variant option you wish to use for the connector placement
- ⇒ Click **OK**

The connector will now be duplicated and placed at the new variant position as well as at the original location.

The primary instance of the connector will have the attribute 'is primary instance' assigned.

The wires at the variant connector are identical to those in the original connector, however the option expressions on the wires will be a combination of the existing option expression on the wires and the variant option expression held in the variant positioning dialog.

Synchronizing Variant positioned connectors

- If the master connector has any edits made, users will need to manually update the variably placed connector (right click/synchronize variant position connectors)
- Users may wish to assign a different part number to the original connector, this will not affect the part number on the variant connector unless a sync is performed. When a different part number is assigned to the original connector wires are automatically mapped to the new cavities using the same ordering. Both instances of the wire (on the original and

variant connector) are displayed on the wire edit form. The variant instance will have an attribute 'is variant position' assigned

- If the Variant connectors are edited, users will not be able to synchronize back to the primary position connector

Variant positioned splices

To instantiate variant splice positions:

- ⇒ Insert a splice at an alternative location
- ⇒ Edit the original splice and ensure it has a relevant option
- ⇒ Select the variant positions tab
- ⇒ Select the alternatively positioned (variant) splice
- ⇒ Assign an option to the variant splice

The wiring from the original splice will be duplicated in the variant splice.

Synchronization

If a synchronization is performed from the Logic or Integrator tools, the variably positioned component information remains unchanged until a manual sync is performed in XC.

Variant positioned objects

Objects such as grommets, clips and additional components can also be variably placed along the bundle and linked to structure or other reference nodes on the bundle.

To insert variably placed clips on the diagram:

- ⇒ Insert a clip as normal
- ⇒ Assign a node position to the clip selecting the required anchor node
- ⇒ Define a variant position(s) for the clip (click Add variant position and define the variant option, anchor node and dimension)

Variantly placed objects will retain their positioning relative to the defined anchor node.

If a bundle is merged, the positioning of objects will be amended so that the anchor node changes to the next available reference node on the bundle.

In order to view the variant dimensions a variant dimension must be defined in the style set, this is held under Edit/Style/Style set/select the style set/dimension/variant dimension. Users must define what information they wish to see for a variant dimension on the diagram.

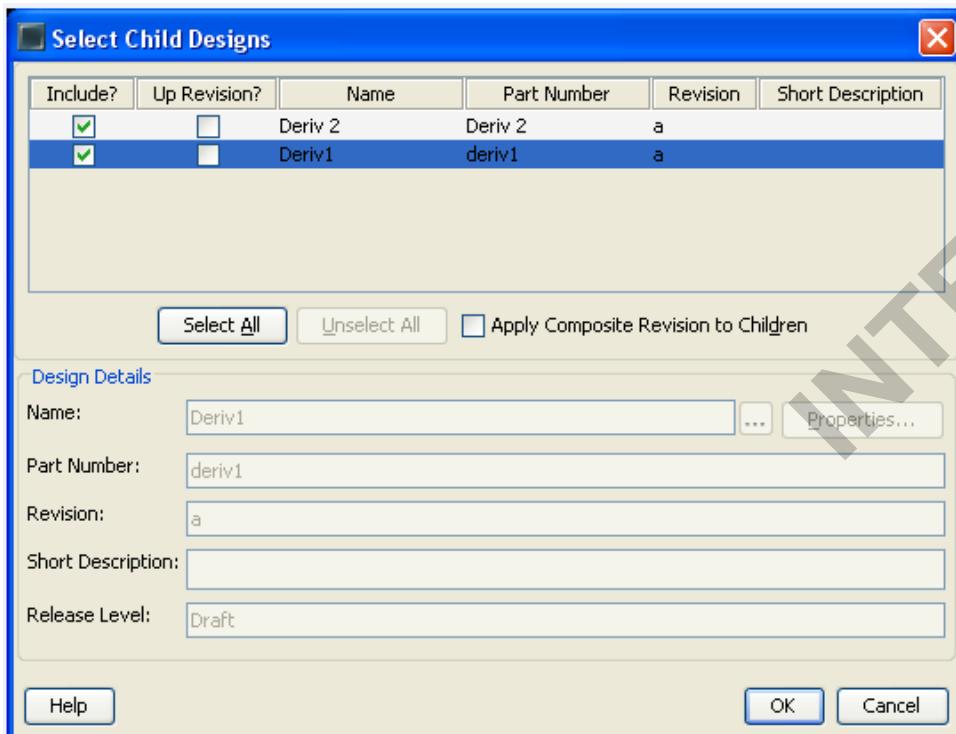
INTERNAL

Creating a Revision of a Composite

- ⇒ Right Click on the composite design and select to create a revision.

As mentioned earlier, the part number and revision combination is unique across the system. Modify as required.

To create revisions of the required derivatives select the ellipse button next to the **Select Derivatives** field.



It is possible from here to state which derivatives are to be revised.

Select All: will allow you to change the revision (or part number) for all derivatives.

Alternatively **Unselect All**, will allow you to manually choose which derivatives are to be revised.

Select the derivative and revision as required. A new derivative will be created underneath the new revision of the composite.

If all derivatives are to be revised and will take the same revision as the composite, users can check the up-revision box for each derivative and check the '**Apply Composite Revision to Derivatives?**' box

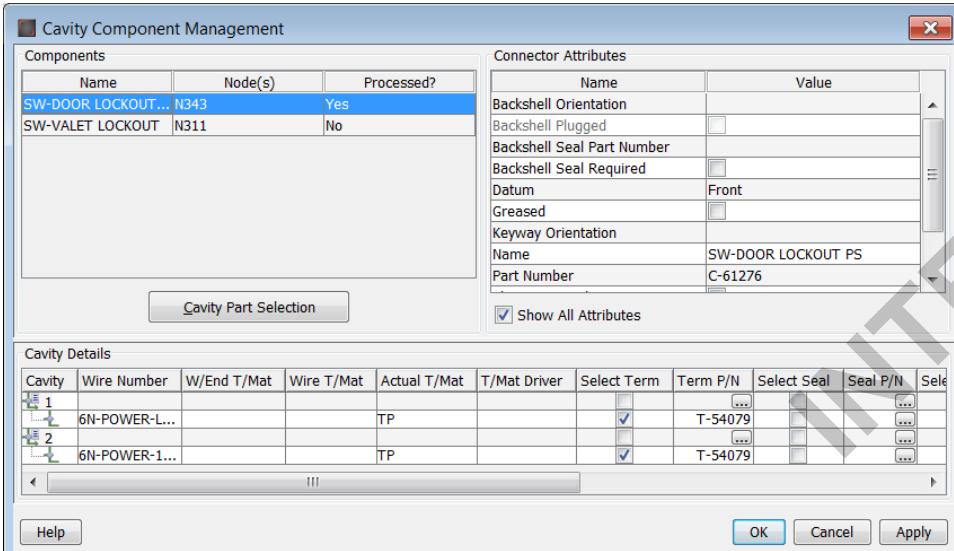
A composite revision table may be placed on the Composite diagram – this may be specified in the style set and is placed by applying the style set – (see earlier steps for placing the composite table)

Cavity Component Management

Using the Cavity Component Management function it is possible to manage the selection for each cavity component in one dialog, including derivative harnesses.

- ⇒ Right Click in free space on the diagram
- ⇒ Select **Cavity Component Management**

Upon opening this dialog the system will select the cavity components for the selected nodes.



Note

The dialog can also be accessed by Right clicking on a connector (only that connector will be presented for modification).

By selecting a connector all of the cavities, and the terminating wires, are displayed in the browser at the bottom of the dialog.

Connector Attributes

It is possible to set the selections required at the connector level. Setting a connector to be Plugged or Sealed at this level indicates to the system that cavity plugs and or cavity seals need to be selected.

The connector attribute terminal material drives the terminal material of the cavities of that connector.

Cavity Details

This area displays the cavity component details for the chosen connector.

- ⇒ **Wire number:** displays all wires to that cavity. For a composite, the system would display all combinations of wires in the derivatives. This would mean the cavity would be expanded to show as many combinations, not derivatives
- ⇒ **W/End T/Mat:** displays the terminal material defined against the wire end under the wire end tab
- ⇒ **Wire T/Mat:** displays the terminal material assigned to the wire, selected by the user. This is **not** used in selecting a terminal and is used for information only. This is set against the wire attributes
- ⇒ **Actual T/Mat:** Displays the terminal material code that has been selected
- ⇒ **T/Mat Driver:** displays which of the 3 terminal material drivers is being used to auto select a terminal (wire end, wire attributes, connector)
- ⇒ **Select Seal/Terminal:** this checkbox is toggled on when a connector is set to require seals/terminals in the connector attributes section. This checkbox requests the system to automatically select a cavity seal/terminal
- ⇒ **Seal P/N or Plug P/N or Term P/N:** displays the cavity seal/plug/terminal internal part number. If the Select Seal/Plug/Terminal checkbox is on, the part selected by the system is displayed and cannot be edited by the user. If the checkbox is off, the user can manually choose a part by using the ellipses button. All parts in the database will be available for selection. If the part does not exist in the housing definition DRCs and Harness Checks will report it. If the **Sealed/Plugged**

- checkbox in the connector attributes section is **not** selected then a cavity seal/plug part cannot be manually added.
- ⇒ **Backshell Plug P/N:** displays the backshell plug internal part number selected by the system
 - ⇒ **Connector (T/Mat Driver):** displays the terminal material from the connector attributes section.
 - ⇒ **Extra Components:** displays any extra components assigned to the cavity selected by the user
 - ⇒ **DRC Results:** allows the user to see the DRC results specific to cavity components
 - ⇒ **Harnesses:** Displays the derivative name in which the cavity/wire combination exists

Cavity Part Selection

Running the process within the dialog after making changes, the system will search for the correct cavity components using the parameters supplied. All parts selected will be displayed and where a part cannot be found the field will remain empty and the DRC will indicate a problem.

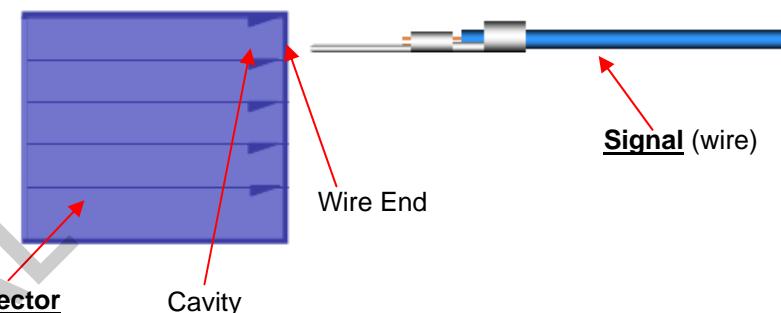
For a composite harness, the system will determine the wire combinations for each cavity on each derivative and display each combination.

If the cavity is empty users can still insert a seal or terminal part against it which can be used for future requirements. If a wire is subsequently inserted in the empty cavity a DRC will highlight if the selected terminal or seal is suitable for the wire size.

Where a composite harness is defined, users are able to define terminals and seals for empty cavities on the derivatives, an extra row per cavity is displayed for each instance of wire with differing options assigned

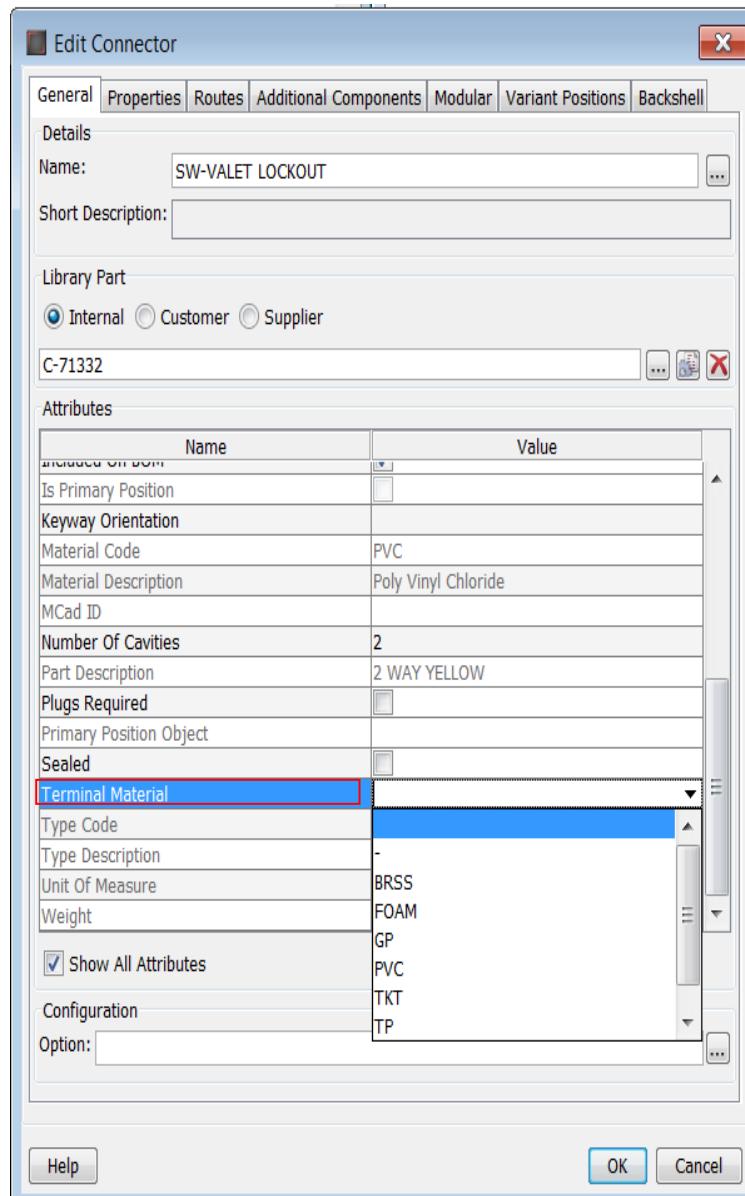
Terminal Material Drivers

A terminal material can be chosen by 2 main drivers, the connector and the signal.



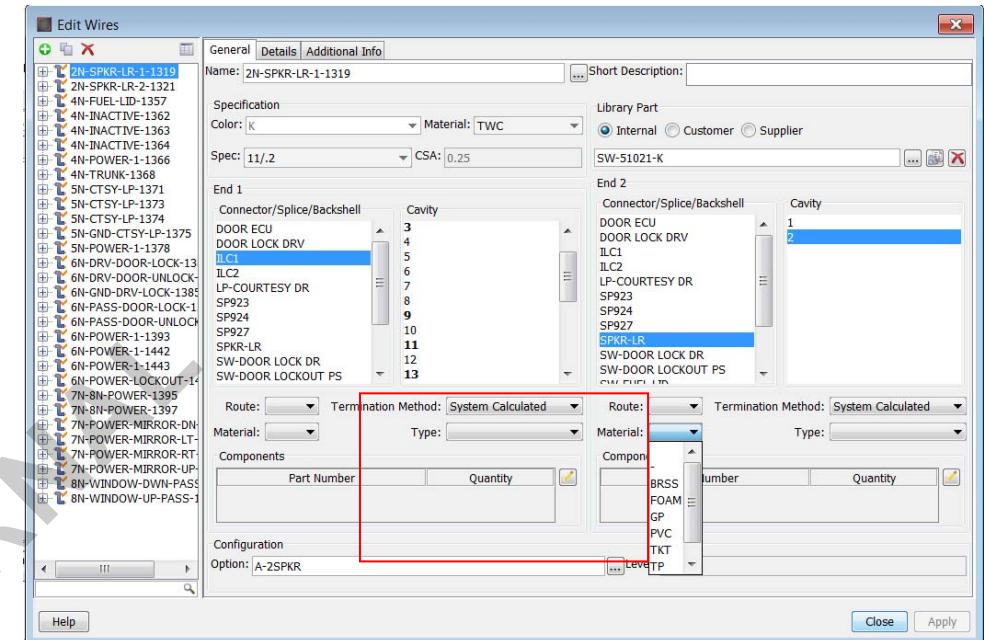
There are 4 places in Capital HarnessXC to define the terminal material driver.

Against the **Connector** on Insert Connector or in the Cavity Component Management form.



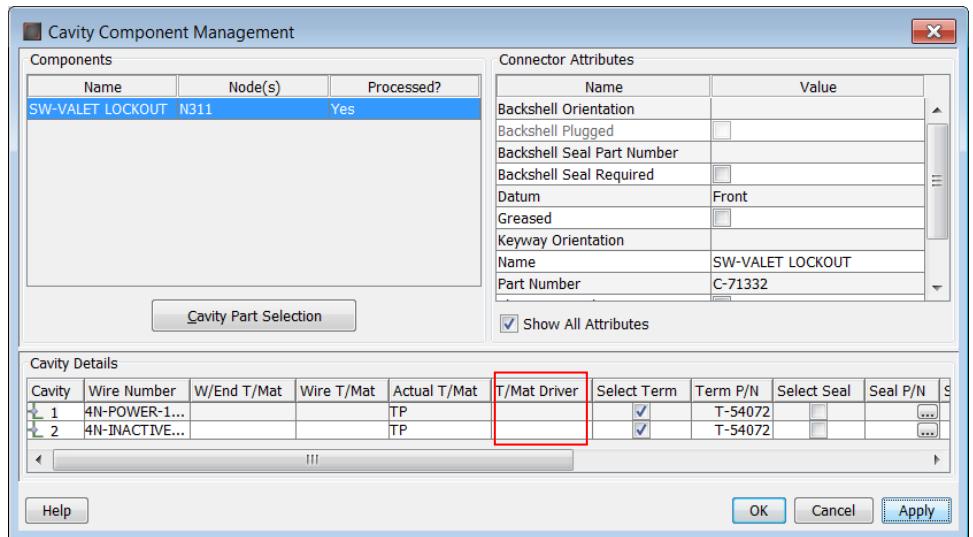
Against the wire end in a cavity in the Edit Wires form. **NOTE: this is actually the cavity not the wire end**

Against the signal (wire) in Edit wires.



Capital Harness XC uses the following precedence to populate the material that will be used to automatically select a terminal. From top to bottom, the system checks if a terminal material driver has been set against that wire end in the Edit wires form under the wire End tab. If one has been set then this will be populated in the T/Mat Driver field in the Cavity Component Management dialog and will be used to automatically select the relevant component. If one hasn't been set then the system will move to the next object in the list (wire driver – set in the wire attributes form) and use that one if no wire driver has been set then the system moves to the connector driver. If none have been set then the system will search for any terminal of any material

- ⇒ Wire end driver
- ⇒ Wire driver
- ⇒ Connector driver



Note

Users should be aware that if a wire is relocated in a different cavity, the terminal driver does not move with the wire. It is essential to ensure that the correct terminal driver is re-assigned to the wire end in order that the processing calculations can select the correct terminal. If this is not specified the system automatically selects the standard terminal assigned to the connector at the node. An extra harness check is available to identify if the terminal driver field is blank or contains a hyphen (-) when a terminal part number is assigned to the cavity.

Exercise:

Using Cavity Component Management set all connectors to use Seals and Plugs

Chapter 9

Through Nodes & Modular Connectors

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Management of Backshell components

Some connectors may require backshells and backshell terminations to be associated to them. If this is the case, and they have not been associated by way of a library part number or brought through via a synchronisation from Logic/Integrator or from the Bridges tool, they may be manually added in the XC tool:

To manage the backshell information on a connector:

- ⇒ Right Click the connector and select **Edit connector**
- ⇒ Select the **Backshell** tab within the edit connector form
- ⇒ Define a backshell name
- ⇒ Associate a backshell library part number to the connector

If the library part number already has backshell terminations defined, these will be displayed in the termination field of the form. Users may set the style set to display the termination information on the diagram relative to the connector symbol – style sets will be covered in a later chapter.

If no termination information has been assigned to the library part it may be manually assigned from within the backshell form by typing in a name in the termination field and clicking Add. In order to produce a correct bill of Materials, users would need to assign a library part number to the backshell at some point.

- ⇒ Edit the wire and select the backshell termination as a wire end location

When running cavity component management users can view the backshell termination detail and any cavity parts selected for the backshell.

When wires are to be measured to the backshell component, users can ensure that the calculation of wire length is amended to the backshell by setting the engineering parameter for backshell termination wire length calculation to use backshell attributes:

- ⇒ System preferences/Harness Engineering

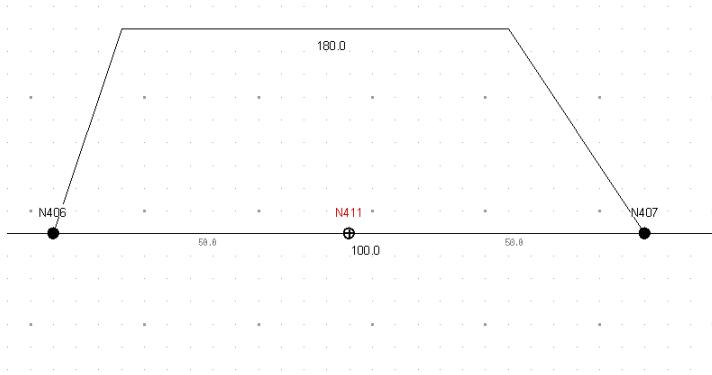
Harness Engineering Parameters	
Parameter	Value
Add-On Per Junction	3.0
Percentage Add-On	0.0
Minimum Percentage Value	0.0
Maximum Percentage Value	0.0
Wire Length Rounding Method	None
Wire Length Rounding Value	1.0
Default Link Lead Length	100.0
Looped Splice Wire Add-On Value	100.0
Default Splice Strip Length	10.0
Default Solder Sleeve Strip Length	10.0
Default Ultrasonic Weld Strip Length	10.0
Default Center Strip Length	12.0
Bundle CSA Multiplication Factor	1.3
Mandatory Spot Tape Layer	1000
Replacement Splice Wire Prefix	SPL
Tube Mandatory Component Positions	None
Overlap/Spiral Tape Calculation Method	Nodes
Housing Definition Nesting Level	0
Tape Usage Percentage Add-On	
Default Grease Application Method	Cavity
Default Grease Component	
Default Grease Packing Component	
Tube Selection Tolerance	0.0
Tube Selection Order	CutExact,FromReel
Backshell Termination Wire Length Calculation	<input type="button" value="Use Connector Attributes"/> <input type="button" value="Use Backshell Attributes"/>

Exercise 1:

Using information shown in exercise 1 on the exercise worksheet, associate backshell information to connectors

Through Nodes

It is possible to specify a particular route for wires and insulation. If no through nodes are specified then the wires/insulation will be allocated the shortest route by default.



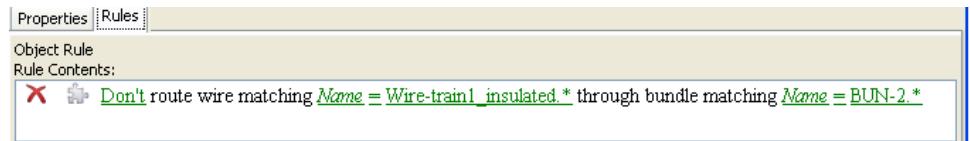
As soon as a loop is created in the branch configuration the system automatically creates a reference node to allow the routing of wires and insulation (N411 in the above example).

To set the path of the wire through a particular node:

- ⇒ Right Click on the wire/insulation run from the tree browser
- ⇒ Select the option to **Set Path**

- ⇒ **Select** the node for the path, double clicking will toggle a through or not through node
- ⇒ **Double click** to commit.

An alternative to using the through node/not through node function is to implement a constraint on a bundle to ensure that certain wires are or are not routed through the bundle depending on specifications made relating to the wire attributes or properties..



Modular Connectors

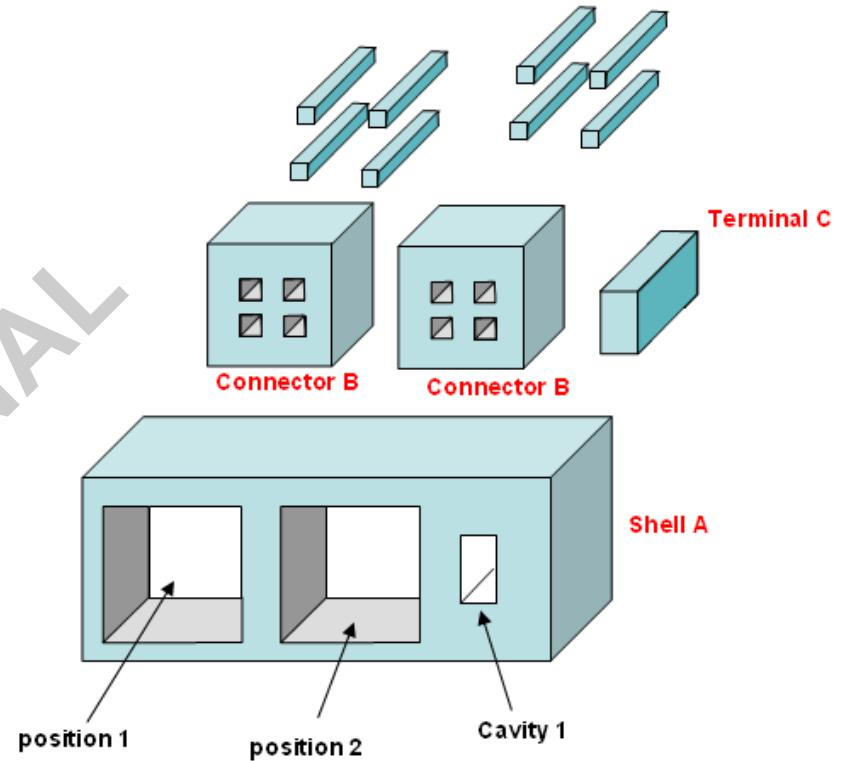
A modular connector is where more than one connector can be combined together to create a single connector by positional inserts.

The diagram shows an example where the modular connector is an assembly of:

- ⇒ Shell A (the main connector)
- ⇒ Connector B
- ⇒ Connector B
- ⇒ Terminal C

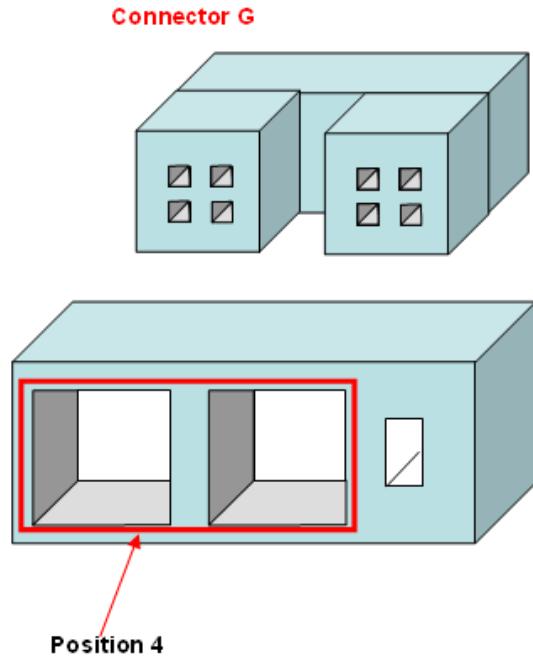
There are several variations in this example:

1. In one harness Connector B may be located at position 1
2. And another harness Connector B may be located at position 2
3. Alternatively Connector B could be used twice on the same harness in position 1 and position 2.



Position Blocking

In the diagram below, Connector G is located in position 4 which would block position 1 and 2 and of course using Connector B in either position 1 or position 2 would block position 4.

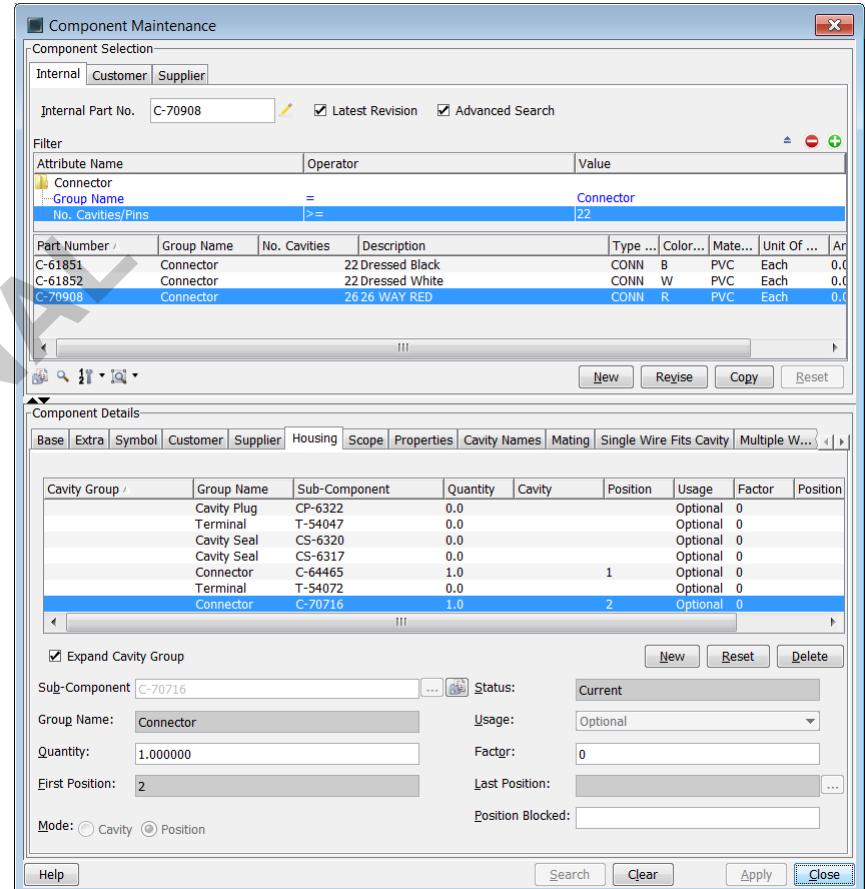


Exercise 2:

The trainer will now present a power point to show examples of real modular connectors; 'Modular conns photos.ppt'

Modular Connectors in Capital Library

The main shell is created with the component group **Connector**. The sub-connectors and position information is created in the **Housing Definition** of the main connector.



- ⇒ Select the **Mode** to be **Position** to assign connectors as sub components to the main connector
- ⇒ Select the **library part** for the sub-connector
- ⇒ Select the **quantity**
- ⇒ Select the **first position** that this connector uses

- ⇒ If any positions have already been defined, select, if necessary, any **positions** that would be **blocked** if this connector were used

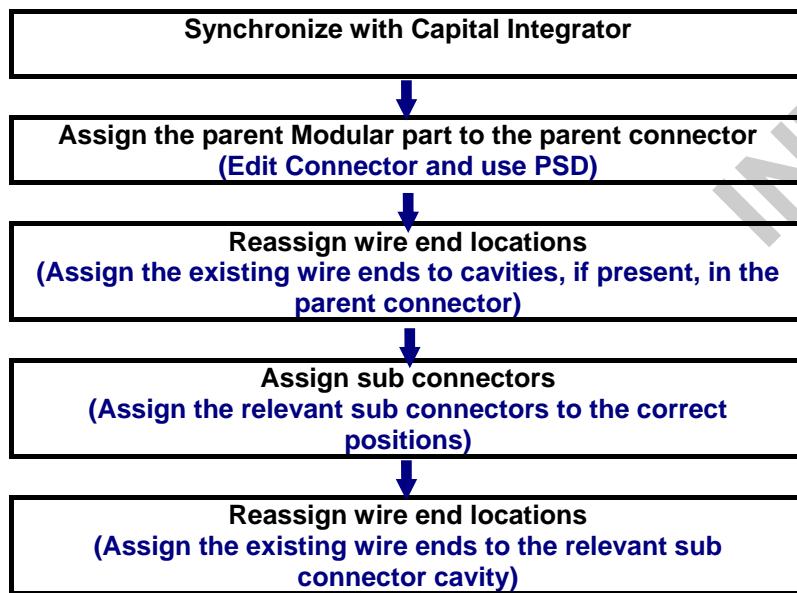
Note

Where a connector may be used in a range of positions e.g. 1-4 then the First and Last Position fields can be used.

To block more than one position, separate the blocked position with a comma.

Assigning a Modular Connector

One flow from Capital Integrator could be where the modular set up is not defined and the wires will be transferred to Capital Harness as if in one big connector with cavities. The sub-modules are defined and wires assigned accordingly. In this case it will be necessary to assign the modular connector part details along with ensuring the wire locations are correct.



Here is an example of what could be imported from Capital Integrator
All wires imported into one connector.

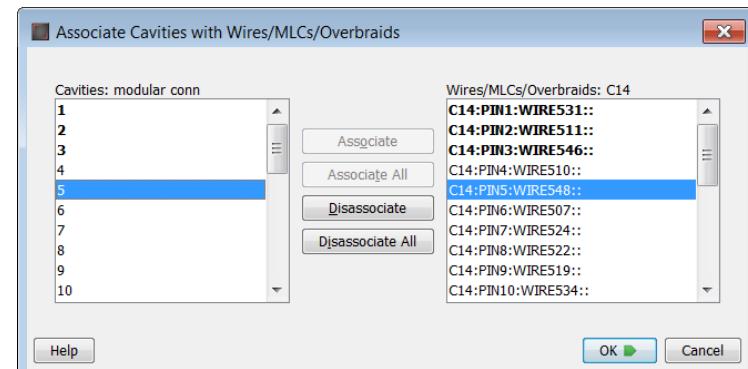
ILC1

Cavity No.	Color	Spec	Type	Option	CSA
2	1N-SPKR-LF-3-925	B	16/2 TWC	A-4SPKR	0.5
1	1N-SPKR-LF-4-927	Y	16/2 TWC	A-4SPKR	0.5
3	1N-SPKR-LR-7-928	R/B	16/2 TWC		0.5
4	1N-SPKR-LR-8-930	R	16/2 TWC		0.5
5	3N-POWER-FUEL-978	G	16/2 TWC		0.5
6	3N-POWER-FUEL LID-979	O/R	12/2 TWC	FuelLid	0.35
7	3N-POWER-TRUNK-982	P/U	16/2 TWC	Trunk	0.5
8	4N-CTS Y LP-1795-988	G	12/2 TWC		0.35
9	4N-GND-CTS Y LP-989	O/R	12/2 TWC		0.35
10	4N-POWER-2-993	G/W	16/2 TWC		0.5
11	5N-GND-DRV LOCK-999	B/K	16/2 TWC	PDrLcks-Dr	0.5
12	-	-	-	-	-
13	-	-	-	-	-
14	-	-	-	-	-
15	6N-GND-MIR-1013	O/R	12/2 TWC		0.35
16	6N-POWER-MIR-1-1015	G/W	16/2 TWC		0.5
17	-	-	-	-	-
18	7N-GND-WNDOW-DRV-10450	16/2 TWC	PWndws		0.5
19	-	-	-	-	-
20	-	-	-	-	-
21	-	-	-	-	-
22	1N-SPKR-LT-11-1093	B/R	12/2 TWC	A-6SPKR	0.35
23	1N-SPKR-LT-12-1095	B	16/2 TWC	A-6SPKR	0.5
24	-	-	-	-	-
25	-	-	-	-	-
26	-	-	-	-	-

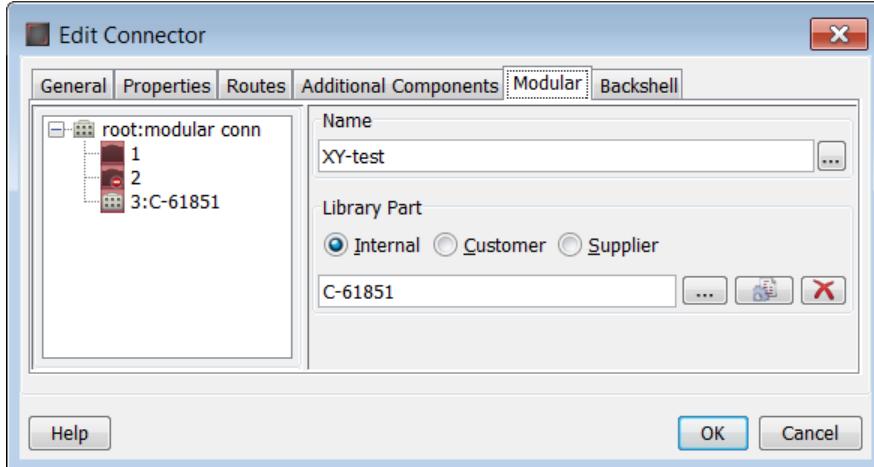
C-60908

The flow could be as follows:

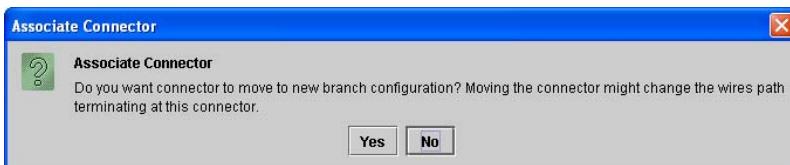
- ⇒ Select to **Edit the Modular Parent Connector**
- ⇒ Using the PSD assign the parent modular connector part number
- ⇒ **Reassign wires:** reassign the parent modular wires to the correct cavities



- ⇒ Go to the **Modular Tab** and select the correct sub modules, using the positions and the PSD. Once selected reassign the wires to the sub module.



- ⇒ When a position is blocked it will not be possible to add a sub connector to that position
- ⇒ Repeat this for all sub-module connectors
- ⇒ The system asks how you want to associate the branches
 - **Yes:** will add a branch of length zero to the end of the node where the parent lives
 - **No:** would leave the existing branch to the sub connector where it is



Associate Connectors

When the sub-module connectors and the parent module connector have been placed onto the drawing separately, it is necessary to associate the sub-modules with the parent.

- ⇒ Right Click on the sub connector and select **Associate Connector**
- ⇒ The system displays the available parent connectors and the available positions for the sub-connector. **Select** the correct **position**
- ⇒ The system asks how you want to associate the branches as previously explained.

Modular Connector from Scratch

If necessary it is possible to create a modular connector from scratch.

- ⇒ Insert the parent modular connector using **Edit Connector** and the **PSD**
- ⇒ Using the **Modular Tab** and the PSD select the position and the system will present the library parts available for that position

Chapter 10

Other Functionality

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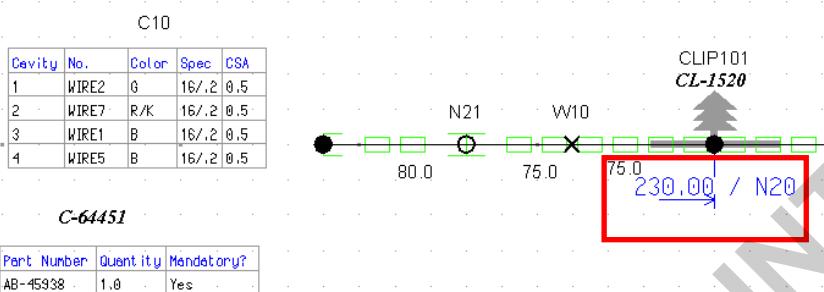
INTERNAL

Reference Dimensions

It is often necessary to ensure that certain objects are dimensioned to a reference point such as clips.

Insert Reference/External Style

- ⇒ Select Add / Dimension / Reference Style or External style
- ⇒ Select object to be dimensioned
- ⇒ Select reference for object to be dimensioned from
- ⇒ Right Click and choose flip to flip the direction of the dimension (above/below the bundle)

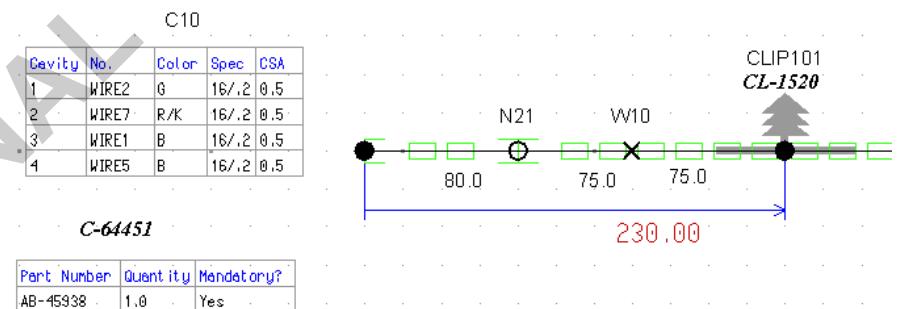


- ⇒ System displays the **combined length** and the **reference node** at the location of the object
- ⇒ If the External style option is chosen the dimensions will be placed externally to the chosen nodes, typically this option is chosen if there is limited space to place a reference dimension



Insert Standard Styles

- ⇒ Select Add / Dimension / Standard Style
- ⇒ Select object to be dimensioned
- ⇒ Select reference for object to be dimensioned from
- ⇒ Right Click and choose flip to flip the direction of the dimension (above/below the bundle)



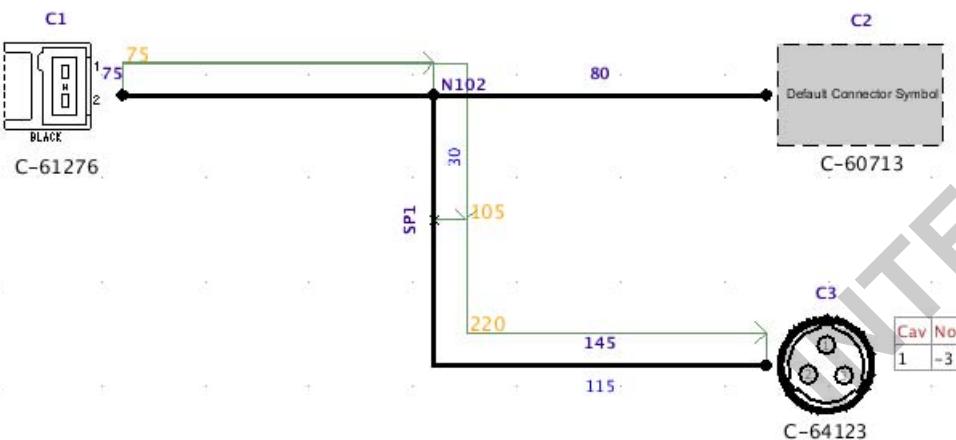
- ⇒ System displays the **combined length** from the reference node

Note

If any of the bundle dimensions are modified this will be reflected in the combined length automatically.

Inserting contiguous style dimensions

- ⇒ Select **Add / Dimension / Contiguous Style**
- ⇒ Select the node to be dimensioned
- ⇒ Select the subsequent nodes that will be dimensioned back to the reference node
- ⇒ **Right Click** and choose flip to flip the direction of the dimension (above/below the bundle)
- ⇒ Double click in a space to apply the contiguous style
- ⇒ System displays incremental dimensions along all selected points



All dimension lines can be moved via drag and drop to allow for best fit within a diagram.

Users can also define a style for each type of dimension graphic, including adding a symbol at the start/end node of the dimension style to indicate the type of dimension it is.

Harness Tables

There are several information tables that can be placed onto the diagram.

The setup of tables and definition of what information will be viewable in the table is done from the styling tool. All tables are placed according to a style set.

The style sets are located in the following menu:

- ⇒ **Edit / Style / Style set**
- ⇒ Select the applicable style set

The generic tables are all held under the **Design** menu

All tables are located within the **Decorations** subset of any menu

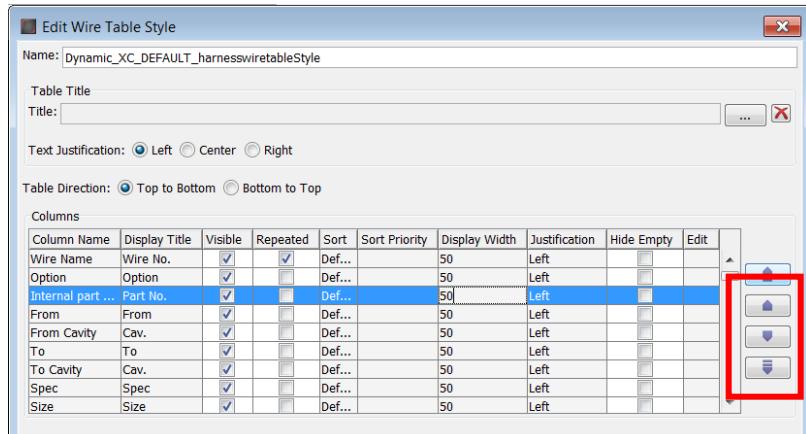
- ⇒ Select **Design / Decorations**
- ⇒ Select the default table you require or add a new one if needed
- ⇒ Click Edit

The table displays may be edited from within the form that pops up

Harness Table manipulation

Column sorting

Users may decide the sort order of columns. From within the edit tables form users may select the column headings and re-sort:



Multiple column headers:

The composite BOM table may also have several column headers, which are defined in the edit table style form for composite BOM tables. The additional column headings are: derivative description and derivative part number which can be made visible or not.

- Multiple Header

Sub Header2:	Derivative Part Number	<input checked="" type="checkbox"/> Visible
Sub Header1:	Derivative Description	<input checked="" type="checkbox"/> Visible
Main Header:	Derivative Name	

Text headings

Individual table column headings may be rotated by changing the settings in the table edit form:

Columns

Column Name	Display Title	Visible	Repeated	Sort	Sort Priority	↑	↓
ID	ID	<input type="checkbox"/>	<input type="checkbox"/>	Default		<input type="checkbox"/>	<input type="checkbox"/>
Description	Description	<input checked="" type="checkbox"/>	<input type="checkbox"/>	Default		<input type="checkbox"/>	<input type="checkbox"/>
Part Number	Part Number	<input checked="" type="checkbox"/>	<input type="checkbox"/>	Default		<input type="checkbox"/>	<input type="checkbox"/>

Multiple Header

Sub Header2:	Derivative Part Number	<input checked="" type="checkbox"/> Visible
Sub Header1:	Derivative Description	<input checked="" type="checkbox"/> Visible
Main Header:	Derivative Name	

Font Details

Cell Text	Main Header Text	Sub Header1 Text	Sub Header2 Text
-----------	------------------	------------------	------------------

Font Information

Font Name:	Arial	<input checked="" type="checkbox"/> Bold		
Font Height:	4	mm		
Color:	Override	<input type="button" value="Change..."/>	<input type="checkbox"/> Opaque	<input checked="" type="checkbox"/> Rotated

Tables available to place on the diagram:

Wire Table:

This displays all wires on the design along with wire details and location for each wire. If placed onto a composite all wires of the composite, along with their option expression will be present in the table.

Composite Table:

This is only available when a composite diagram is open and displays the derivatives along with their option relationships.

Composite BOM:

This is only available when a composite diagram is open and displays the following components and how many of each appears on each (and which) derivative.

- ⇒ Connectors
- ⇒ Clips
- ⇒ Grommets
- ⇒ Other main node components
- ⇒ Additional components
- ⇒ Multi-location Components

Against each component is a BOM ID that is represented in a customized position, typically close to the connector. By default the **BOM IDs** are alphanumeric and placed by the system, by choosing **Actions / Composite BOM / Regenerate all BOM IDs** the ID references can be regenerated.

The Composite BOM table does not update automatically therefore It is necessary to update it manually when objects change on the diagram using **Actions / Composite BOM / Update composite BOM Table**

Revision Table:

This displays the history of revisions of a harness design.

Plug –In Tables:

Users may define their own tables and import them as plug in tables in the styling tool. The plug-in table should already be loaded.

To set the plug-in table style:

- ⇒ Access the style set and select **Design / Decorations / tables / Add**
- ⇒ Select the plug in table
- ⇒ Define a name and associate a plug –in table using the browser button
- ⇒ Define the necessary headers and font details

Print Functions

The printing facility within the Harness XC tool provides various different options;

Standard Printing

- ⇒ Select **File / Print**
- ⇒ Choose a print option (project, diagram, design)
- ⇒ Configure the print choice if needed
- ⇒ The page setup and printer setup are standard settings and enable paper sizing and printer definition respectively
- ⇒ Set the scaling information (fit to paper or print to scale)

Clip to border will trim the print so that only the diagram and the border are printed.

The **Print to File** option enables users to print a project, diagram or design to a pdf, cgm or dxf file.

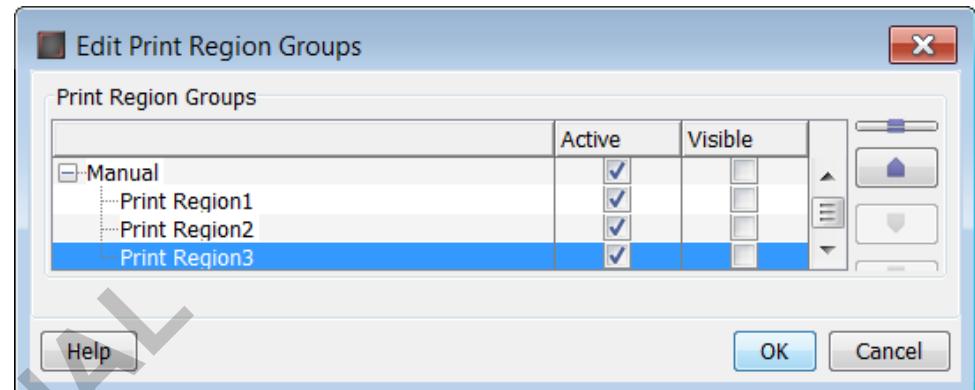
Print Regions

This facility enables users to specify 'zones' or 'regions' on a diagram for printing purposes. In cases where the diagram is large, users can split it up into regions and print out each region individually.

To set up a Print Region:

- ⇒ Select **File / Print Regions / Add**
- ⇒ Using the mouse, define the desired print regions by clicking and dragging the mouse from a start point to an end point
- ⇒ Define as many print regions as required, then click **Enter**
- ⇒ The print regions settings form will appear and users may check the visible box to make the region visible or not on the diagram.

Remember that each individual comment symbol with a print region defined will be visible in the print region settings table and can be printed on a separate sheet.



To print a Print Region:

- ⇒ Go to **File / Print**
- ⇒ In the print drop down field select **Print Regions**
- ⇒ Click the **Configure** button to specify which regions to print
- ⇒ Click **OK** to print the selected Print Regions

Print Selection

Users may select a number of designs within the project browser and opt to print that selection. Each individual design will be printed on a separate sheet.

Export a Single Diagram

It is possible to export the open diagram to a PDF or DXF format.

- ⇒ Select **File / print to file**
- ⇒ Select the **file type** to be exported
- ⇒ Enter the **file name**
- ⇒ **Save**

Language Translation

When a language dictionary is loaded in the Project tool, any user defined text (properties, free comment text, user defined names) can be translated on the diagram:

- ⇒ Select the language indicator in the bottom right corner of the screen
- ⇒ Choose the required language from the available list
- ⇒ All text with associated translations will change to the relevant selected language

Chapter 11

Synchronization

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Change Policies

Using synchronization, designs can be passed from Capital Logic or Capital Integrator into Capital HarnessXC.

Synchronizing with Capital Logic will import the following into Capital HarnessXC:

- ⇒ Wire/multicore wire details
- ⇒ Connector details
- ⇒ Splice information

Capital Logic does **not** contain any layout information for the design.

Synchronizing with Capital Integrator will import the following into Capital HarnessXC:

- ⇒ Wire/multicore wire details
- ⇒ Connector details
- ⇒ Splice information
- ⇒ Branch layout
- ⇒ Branch lengths
- ⇒ Component location

Harness Synchronization requires the following settings to be defined:

- 1: Change Policy
- 2: Harness Build list
3. Logic build list if synching from Logic
- 4: Harness Attribute definition (if synchronizing from Logic)
5. Connector poke home definition (if synchronizing from Integrator and using poke home information)

Change Policy Definition

As some objects can be authored in Capital HarnessXC it is necessary to determine which objects, during synchronization, should be allowed or prohibited from altering the harness diagram.

It is important to define the master – slave relationship with respect to the data exchanged during the sync process. There will be occasions where the data on the target design is the master data and vice versa.

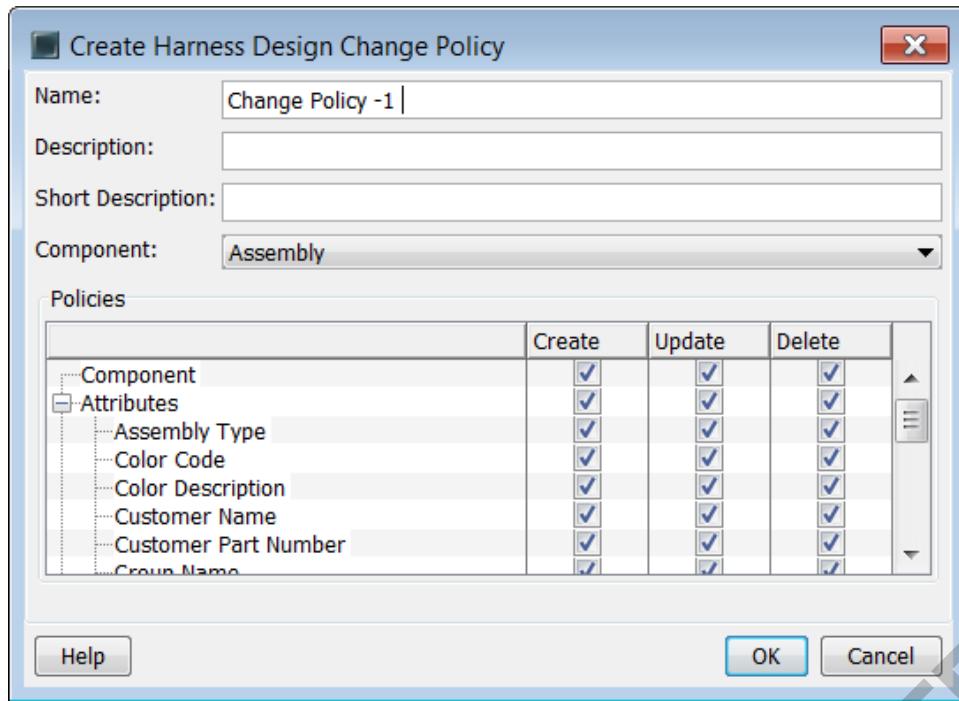
To control this exchange of data, a change policy is used to determine what object details should be passed to the target, Capital HarnessXC. There are three levels of management:

- ⇒ Main component **group** level
- ⇒ **Attributes** of component group; e.g. for a wire it's color, material etc
- ⇒ **Properties** of a component

The user can state in the change policy for each object type, what data is to be created, updated or deleted when it is passed to the target harness design from the Integrator or Logic design.

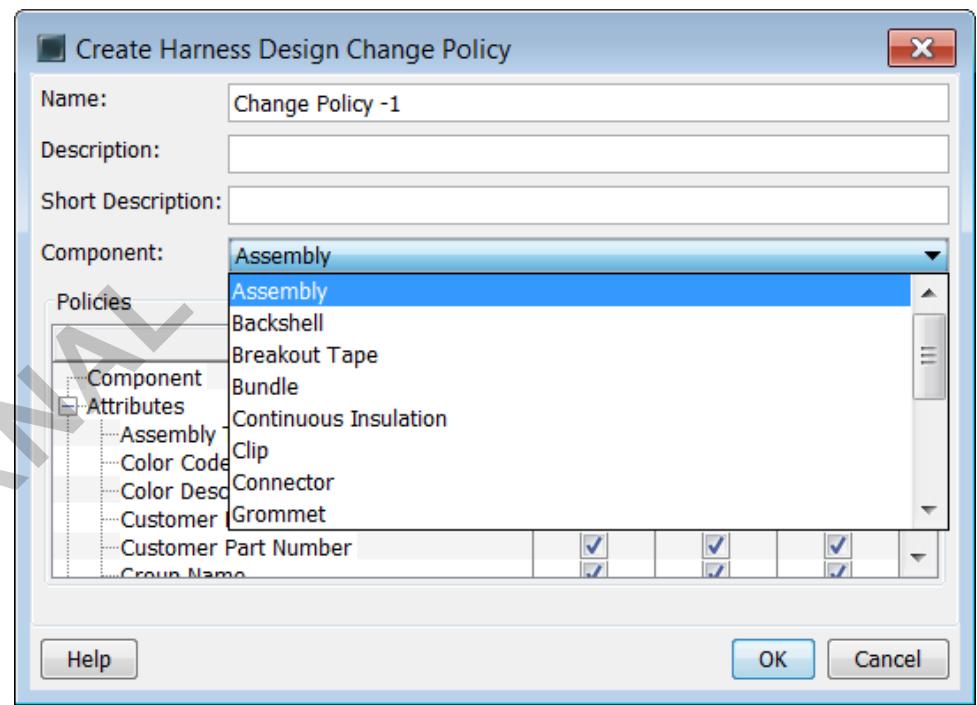
To create a change Policy:

- ⇒ Open Capital Project and your currently active project
- ⇒ Open the Change policy folder
- ⇒ Right Click Harness and select **New**
- ⇒ Define a name for the policy
- ⇒ Click **OK**



The default values allow “Creation”, “Update”, and “Delete” actions in capital Harness XC of all relevant “Components” and their inherent “Attributes” and user set “Properties”. This means that Capital Integrator/Logic is the master with respect to these objects and changes in Capital Integrator/Logic will always be passed along to Capital Harness XC.

Definition of an accurate change policy is not a simple task; it requires careful consideration of each component type and knowledge of the design process to be employed. Below we see the component types displayed:



The primary task in setting Up a Change Policy lies in determining the behavior of the Attributes and Properties. Some of the components have few attributes, as is the case for Bundles, while other components have more attributes such as wires.

Each component needs to be carefully considered and the following questions asked:

1. Is this component created in Integrator? If so, do I want it to be sent (“Created”) to Capital Harness XC?
2. If the component is changed in Capital Integrator, do I want the changed information sent (“Update”) to Capital Harness XC?

3. If the component is deleted in Capital Integrator, do I want it removed ("Delete") from Capital Harness XC?
4. Are attribute values for this component created in Integrator? If so, do I want the values to be sent ("Created") to Capital Harness XC?
5. If attribute values for this component are changed in Capital Integrator, do I want the changed information sent ("Update") to Capital Harness XC?
6. If attribute values for this component are deleted in Capital Integrator, do I want it removed ("Delete") from Capital Harness XC?
7. Is this attribute value created in Integrator? If so, do I want it to be sent ("Created") in Capital Harness XC?
8. If this attribute value is changed in Capital Integrator, do I want the changed information sent ("Update") to Capital Harness XC?
9. If this attribute value is deleted in Capital Integrator, do I want it removed ("Delete") from Capital Harness XC?

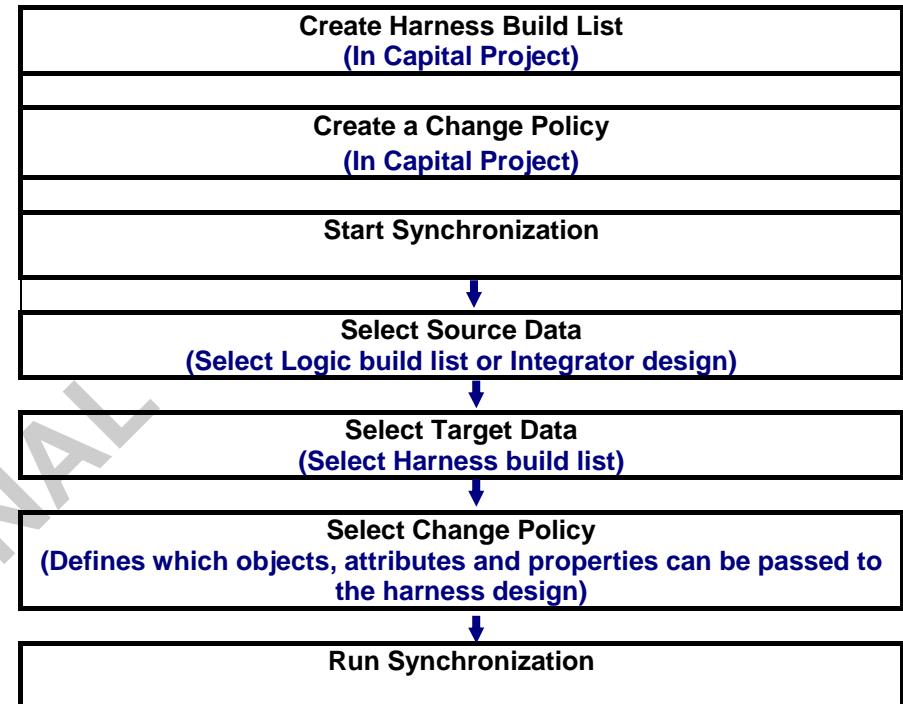
Some possible business examples:

- ⇒ Wire sizes and locations are authored in Capital Logic but the wire color is authored in Capital HarnessXC as this is for the harness manufacturer to define. After the first synchronization the wire colors are defined. On subsequent synchronizations it is important that the wire colors remain the same. To achieve this, the user would **prohibit** the wire color being imported onto the harness diagram but **authorize** all other wire details to pass.
- ⇒ Capital Integrator could be the master data for bundle information. The user may add a bundle or change a length in harness for a temporary basis. To ensure that Capital Integrator bundle information remains the master, the user would **authorize** bundle information to pass which would remove any changes made in Capital HarnessXC.

Note

This is a uni-directional flow to exchange harness data therefore any changes made in Capital HarnessXC will not be reflected in Capital Logic or Capital Integrator.

Access to the change policy can be restricted from within Capital User so that all users may see the available change policies but cannot change them.



Creating a Harness Build List

A harness build list is required for synchronization. The harness build list generally would contain all the relevant harnesses that are required to build a vehicle. It is a method of collating all required harnesses, at the most recent revision, in a list. A build list can be created by adding standard harness information to the list or alternatively it may be created from an effectivity definition.

To create a build list:

- ⇒ In Capital Project open the project and open the build lists folder
- ⇒ Right Click the harness reference and select **New**
- ⇒ Type in the name of the build list
- ⇒ Add designs to the build list using the **Add** button
- ⇒ Click **Validate**. The validate process ensures that where shared objects exist across designs, all instances of the shared objects are included in designs within the build list, thus ensuring a buildable model. The validate button will also indicate if the most recent design revision is assigned, if not the **Update** button will allocate the most recent revision.

Harness Build Lists and Effectivity

An effectivity range may be defined in Capital Project. The function for assigning effectivity must first be activated within Capital Project.

- ⇒ Open a project
- ⇒ Open Preferences / General
- ⇒ Check the Effectivity box

Effectivity ranges may be defined against a design:

- ⇒ Right Click the design name and select Effectivity
- ⇒ Define an effectivity range
- ⇒ Repeat the steps to create more than one range if required

To create a harness build list based on an effectivity range:

- ⇒ Right Click the build list reference
- ⇒ Select '**Create from Effectivity**'
- ⇒ Define a value using a predefined effectivity range
- ⇒ Click the dynamic box if you require the build list content to be automatically updated with designs that fall within the build list effectivity range
- ⇒ click **OK** and the Harness Build list form will open

Logic Build List Creation

Logic build lists must be created within the Logic or Capital Project tools:

- ⇒ Open the build list folder
- ⇒ Right click the Logic build list sub folder and select New
- ⇒ Define a name for the build list
- ⇒ Add the relevant logic designs to the build list
- ⇒ Validate and close the form

Logic Harness Attribute definition

If synchronizing designs from the Capital Logic tool into Capital Harness XC users must first assign harness attributes to the Logic design objects:

- ⇒ Select the objects to which a harness attribute is to be added
- ⇒ Right click the object and select 'Edit Harness Attribute'
- ⇒ Type in the harness attribute name
- ⇒ Save the Logic design

Different attributes may be added to objects in the same Logic design. If the same attribute is to be added to several Logic objects users may multi-select those objects and assign the attribute at the same time.

If users wish to establish poke home information in the harness diagram via a sync from Logic it is important that the 'treat wires terminating on a different harness as poke home' box is checked under Capital Project preferences.

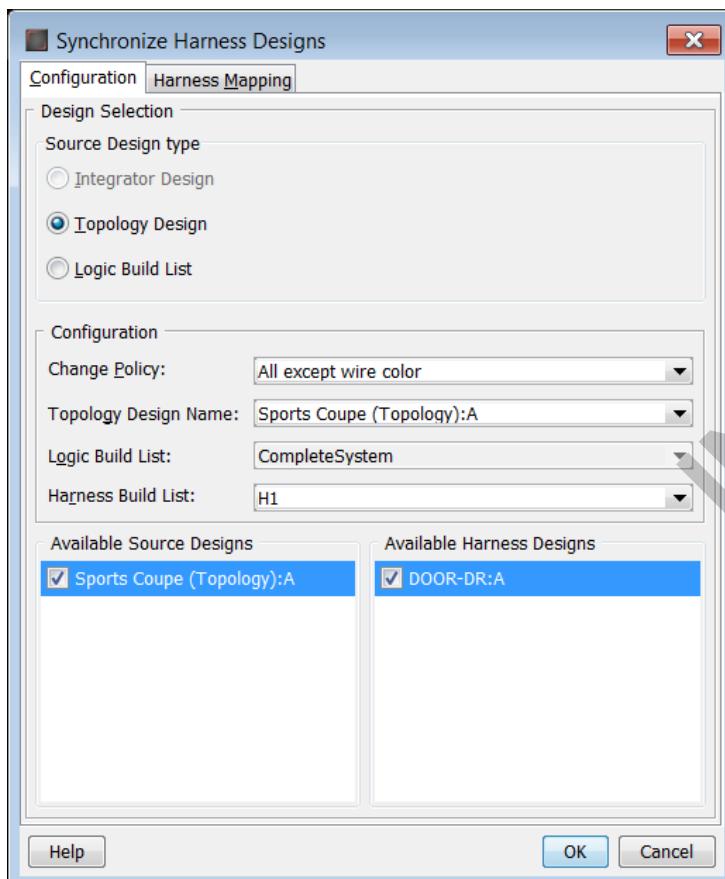
Synchronizing Designs

To synchronize designs in Harness XC:

- ⇒ Select **Actions / Synchronization/synchronize**
- ⇒ Select **Source Design Type**

Choosing the source design type presents further choices as follows.

Synchronizing Logic Designs



- ⇒ Select **Logic Build List** from the Source Design Type

- ⇒ Select the **change policy** to be used
- ⇒ Select the **build list** to be synchronized from the drop down list. This field will only be editable if the Logic Build List Design Type is selected
- ⇒ The system will present **available source designs** to be included or not in the synchronization
- ⇒ Select the **source designs** required for the synchronization
- ⇒ The system will select **available harness designs** to be synchronized with. Source harnesses are automatically mapped to target harness designs by the system on a one to one basis using the harness names. Users can override the default mappings and select which source designs to associate with which target designs by using the Harness Mapping Tab.

Harness Mapping

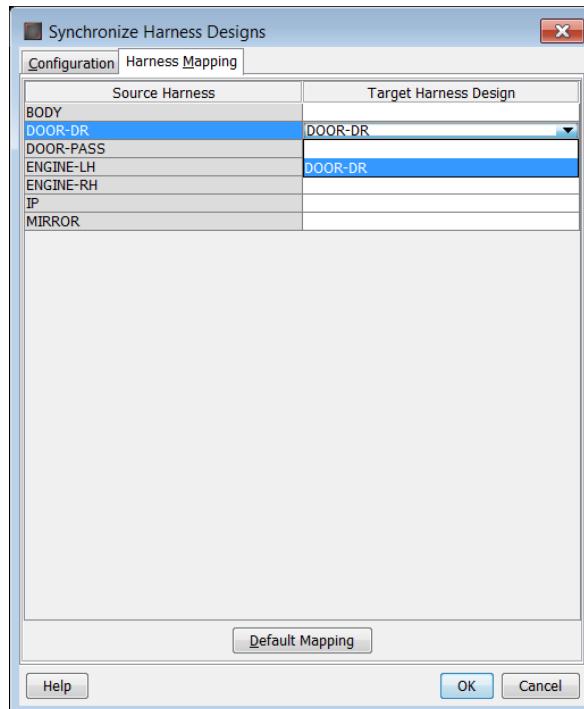
When the Logic build list or integrator design is chosen the system will present all designs in the available source designs pane.

When the harness build list is chosen the harness designs will be presented in the available harness designs pane.

The system will highlight all designs available for synchronization.

When a source design is selected, the system will select the matching harness design. The system assumes a match if the harness design name matches the harness name contained in the source designs area. Many source designs can map to the same harness design.

It may be necessary to override the harness mapping made by the system by selecting the Harness Mapping Tab.



When the Logic build list is chosen, the harness attributes will be displayed under the Harness Mapping tab and a manual mapping can be performed to match harness attributes to the relevant harness.

Note

When a connector with a different harness attribute is synchronized to a harness it will be assumed that the wires in that connector will be poke home. The system will ensure that the 'include on BOM' attribute for the connector is unchecked.

An Audit trail reference exists to allow users to identify what designs have been synchronized with a Harness design.

Note

Logic objects need to have one or more harness attributes assigned in order for the harness mapping to be performed.

Synchronizing Integrator Designs

Poke Home Wires

It is often the case that some wires on a harness must be terminated in situ, thus negating the need for a connector to be included on the Bill of Material. When an integrator or Logic design is synchronized with a harness design, any connector with a poke home attribute is automatically assumed to be not on the bill of materials and wires terminating at these connectors will be considered as poke home wires in the harness diagram.

To define a poke home attribute against an integrator connector:

- ⇒ Open the Integrator plane
- ⇒ Right click the connector you wish to assign a poke home reference to
- ⇒ Select properties
- ⇒ Select the Routes tab
- ⇒ Check the poke home box under the poke home column
- ⇒ If bundles from more than 1 harness exist at the connector ensure that you select the relevant bundle for the poke home assignation.

Harnesses bundles without a poke home assignation will be assumed to act as the home instance for the actual connector

- ⇒ Click OK

Note

When a connector is marked as poke home in Integrator, after synchronization the harness attribute of the poke home connector will be that of the harness containing the home instance of the connector (the harness where the connector is included on the bill of material). If cavity component management is run against a poke home connector, restrictions will be implemented to ensure cavity component selection is only implemented (plugs and seals etc) where no poke home wire exists (taking into account all harnesses containing the poke home connector).

Route codes

Where more than 1 bundle from the same or another harness exists at a connector a route code may be assigned to the connector to identify the route the wires will take to get into the connector. This route will also be transferred during synchronization. If poke home information is required, this may also be added.

Once the poke home and route code information has been defined users are ready to perform the synchronization:

- ⇒ Open the Synchronization dialog
- ⇒ Select **Integrator Design** from the Source Design Type
- ⇒ Select the **Change Policy**
- ⇒ Select the **design name** to be synchronized from the drop down list. This field will only be editable if the Integrator Design Type is selected
- ⇒ The system will present **available source designs** to be included or not in the synchronization
- ⇒ Select the **Harness Build List** to be synchronized to
- ⇒ The system will select **available harness designs** to be synchronized with. Source harnesses are automatically mapped to target harness designs by the system on a one to one basis using the harness names.
- ⇒ Override the default mappings if required and select which source designs to associate with which target designs by using the Harness Mapping Tab.

Note

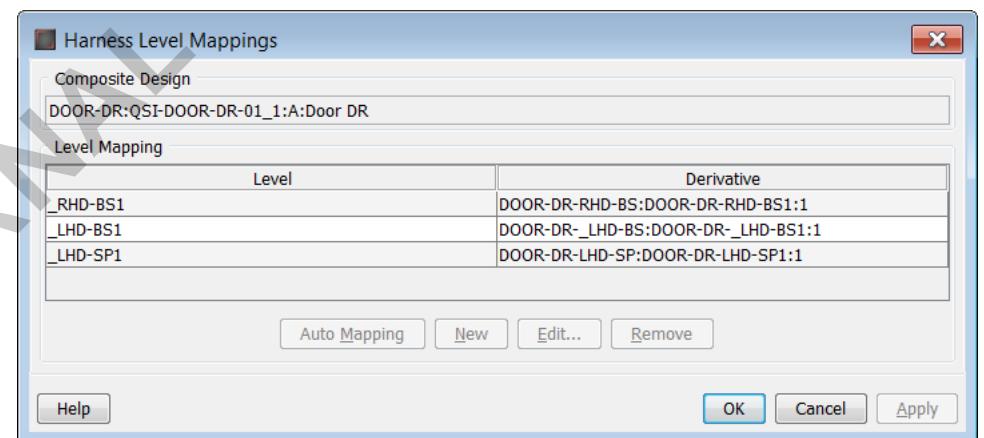
A harness design with released status will not be made available.
If the change policy does not have bundle synch turned on but connector and splice names in Integrator and XC match, wiring will be synched and correctly terminated. If no component name match is found the system searches for node name matches, if no match is found here, synched connectors and splices are placed in free space on the harness diagram.

Level Mapping

Harness level mapping gives the ability to map the levels from Capital Integrator into derivative harness designs in Capital HarnessXC.

A level can only be mapped if it has a part number assigned in Capital Integrator. This is done within Capital Integrator by defining the harness levels for the composite harness (**Define / Harness Levels**) and with the level highlighted, select **Properties** and enter the derivative part number in the Part Number Field.

In Capital HarnessXC to select harness level mapping, Once Synchronization has been performed select **Tools / Harness Level Mappings**.



Auto Mapping: will create a derivative for the selected level/s using the part number from Capital Integrator and the harness composite design name for the derivative design name.

New: allows you to individually select which levels are to be mapped and what derivative details it should have.

Edit: allows you to edit the derivative design details.

Remove: removes the mapping

Component Settings

General

Matching objects between source and target: All objects are matched between source and target primarily by searching for a placeholder component with the same name (connectors, splices, wires etc).

Additional Part Numbers: Any mandatory components associated with the part will be added.

Levels: The level expressions are passed to Capital HarnessXC and option expressions are derived. It is possible to modify the level expression after importing from Capital Integrator but it is not possible to create new levels.

The following rules apply to the synchronization of designs:

- ⇒ The system will automatically use Harness layout co-ordinates when synchronizing designs therefore any components/branches that have changed position will retain their new position as long as the component placeholder matches
- ⇒ Any designs which are not at a valid release status, as set up in Capital Project, cannot be synchronized
- ⇒ Designs which are currently locked for use by another user will not be selectable for synchronization
- ⇒ Any non-harness objects within the source designs e.g. nets, these components are ignored

Connector

Rules for connector synchronization are as follows.

When a direct match of component names is not possible the following options are available:

- ⇒ The name of the object will be matched to a node name (Integrator synch only where node names already exist)
- ⇒ The object cannot be placed on the branch configuration and will be placed on the diagram but not connected to the rest of the harness

Modular Connectors: The sub-modules created in Capital HarnessXC are preserved.

Device Translation: Wires terminating at Devices in the source design will terminate at connector nodes in the target but no part will be assigned.

Backshell Translation: Wires terminating at **Backshells** in the source design will terminate at the main node connector to additional pins added by the system.

Refined Connectors: In Capital Integrator specialized connectors are defined to represent the wire route to connector cavities. Where options are involved wires may be routed to different cavities or different connectors, thus a fanout can be defined within the signal map form in Integrator to help define the wire routing prior to synchronization taking place.

If specialized connectors have been defined in the Integrator plane, users must ensure that the name of the specialized connectors differ from that of the generalized connectors.

Note

If a connector footprint defined in Integrator or Logic has a part number associated but is marked as 'not on BOM' then a dummy connector will be placed against the connector in XC. An eyelet part can then be selected using the cavity component management tool.

Cavity Components

Cavity components are only authored in Capital HarnessXC therefore it is only necessary to state in the Change Policy that the target level object should not be deleted, where the main node components UID matches.

Terminal Plating

This is managed in Capital HarnessXC and any properties added in Capital Logic/Capital Integrator are ignored.

Splices

When a direct match of component names is not possible the following options are available:

- ⇒ The name of the object will be matched to a node name (Integrator synch only where node names already exist)
- ⇒ The object cannot be placed on the branch configuration and will be placed on the diagram but not connected to the rest of the harness

Pin translation: All pin names are overridden for wires not previously present in the target and placed in cavity X of the splice.

Multicores and Shield Management

Shields that are part of a multicore are translated as inner cores.

Exercise 1

Use the exercise worksheet to create a change policy and synchronize information from Capital Integrator or Logic

Synchronizing without a build list or change policy

In some circumstances users may prefer to synchronize wire diagrams without having to define a build list or change policy before hand. In order to do this:

- ⇒ Open your project in Capital Project
- ⇒ Open the project preferences
- ⇒ Select General and check the 'Allow synchronize without build lists' box
- ⇒ Open the project in Capital Harness XC
- ⇒ Right click the wire design name and select synchronize

Note

Remember when synchronizing without a change policy users may risk overwriting data that has already been defined in the harness diagram

Synchronization Simulation

In some cases it is preferable to run a dummy simulation to see what the outcome of the synchronization will be. The Synchronize simulation function offers users the ability to run a simulation without saving the changes to the database. The potential results of a synchronization will be listed in the information bar at the bottom of the drawing. If users wish to proceed with the actual synchronization, they may then return to the Actions menu and run the full synchronization.

To run a synchronization simulation:

- ⇒ Select **Actions / Synchronization/ Simulate**
- ⇒ Define the Logic/Integrator design source, change policy , build list etc
- ⇒ Define the Harness Mapping
- ⇒ Click **OK**
- ⇒ View the results in the information bar

At the base of the information window there are 2 tab buttons for synchronize details and a synchronize summary, each providing information regarding the simulation.

Note

Users should remember that running a synchronization simulation does not save any information to the database, it is more of a 'what-if' function.

Exercise

If users have time, make another change to the Integrator or Logic diagram used for synchronization and run the synchronization simulation process.

INTERNAL

Chapter 12

MCAD

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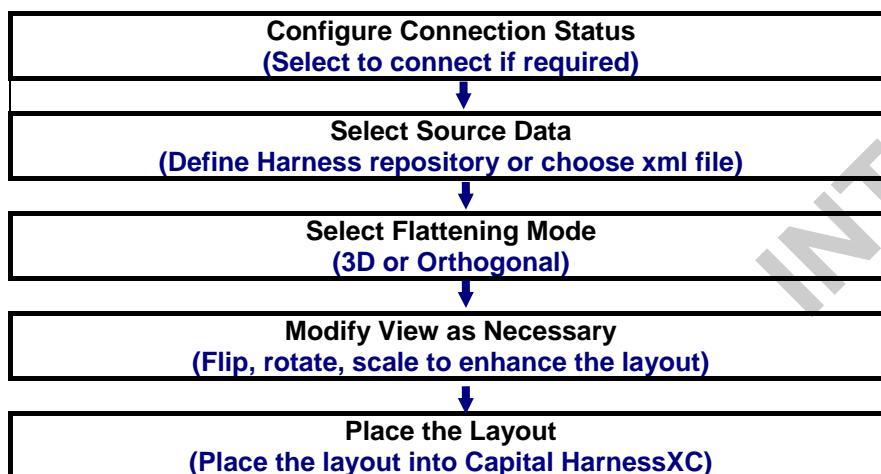
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MCAD

Using Capital Harness Bridges it is possible to import data from various 3rd party tools such as CATIA V5, Ideas, NX4, CATIA V4, Solidworks etc
The following section gives an example of importing and exporting data from Catia but the methods (with the exception of the live connection) are applicable for all available bridges.

The import file for CATIA V5 is in XML format (.xml extension) and has been exported from a CATIA V5 model.

It is possible to run Capital HarnessXC on the same machine or network as CATIA V5 i.e. with a connection between the two systems. This would allow the CATIA user to automatically transfer the harness files between systems. Where this is not the case the harness file will be sent via another medium (e.g. e-mail).

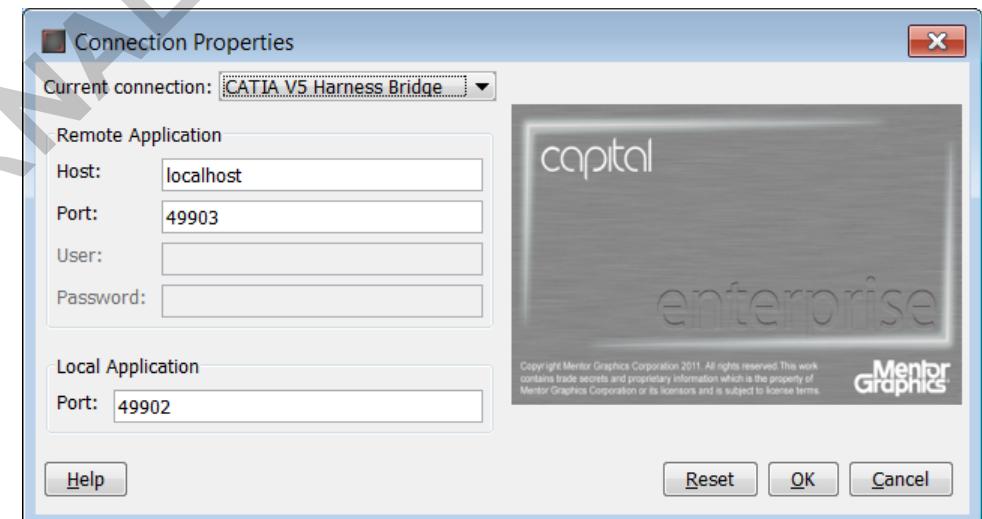


Connected vs Unconnected

Connected: where CATIA is running on the same network/machine as the Capital client install connected mode automatically transfers the CATIA model (in xml format) to the Capital bridge directory.

It is necessary to **Configure** the connection parameters on the Capital side:

The handshake section of the adapters file must be uncommented for the configuration to be available.



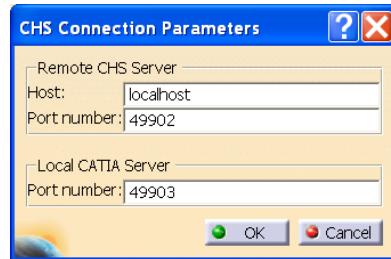
⇒ Remote Application (CATIA)

- Host - the host name of the computer running CATIA.
- Port number - the port number allocated to the CATIA process that listens to incoming messages. It should be the same as the Local CATIA Server port number as configured on the CATIA side

⇒ **Local Application**

- Port number - the port number allocated to the process that listens to incoming CATIA messages. It should be the same as the Remote Server port number as configured on the CATIA side.

To set up the connection parameters on the CATIA side:



Note

The corresponding lines must have been activated (uncommented) with the configuration file adaptors.xml

Unconnected: where the CATIA model xml file is placed manually into the Capital bridge directory.

To switch to connected mode:

⇒ Select **Bridge / Connected**

Bridge in features

When a new design is to be imported to a blank diagram users will activate the bridge in function.

Several options exist for the bridge in:

Import Type:

Replace all wires and multicores: Assuming an initial import of data has been performed, this option will replace only wires and multicores in the target harness

Replace all: This option will overwrite all pre-existing information in the target harness (recommended for the first import of data into a blank harness)

Update All: Selected when data pre-exists in the target harness but is to be updated with modified data in the incoming file (recommended for all subsequent imports)

Users will also have the option to preserve electrical content if required this will leave all wiring information unaffected by the update.

Flattening options: These are explained in detail later

Display report: Upon import completion a report is displayed to indicate the status of the import and any warnings or errors that may have occurred

Preview Harness: When the import button is pressed a preview pane will be displayed that shows the layout of the incoming data. Users can interact with the pane to change the flattening options and define rotation of the harness

Bridge Options: These are explained later

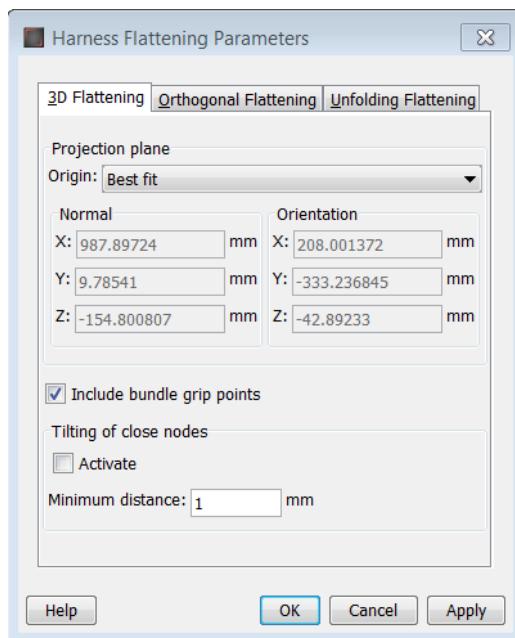
Harness Flattening Options

Often importing a 3D design into 2D can result in diagram which requires layout modification. This tool is designed to help create a readable 2D design.

There are 2 methods of creating the 2D diagram. Both methods find the main trunk by determining which trunk (group of branches) has the greatest bundle size, all other branches breakout from this main trunk.

- ⇒ Select **Display/Edit / Flattening Parameters** or choose the icon  in the preview pane to set up the parameters to be used to flatten the harness on opening the xml file

3D Flattening



3D flattening projects the harness on a plane. There are 3 main ways to find the best projection plane:

- ⇒ **Best Fit:** uses a system algorithm to determine the best layout
- ⇒ **Input Doc:** takes the plane as supplied by the input file

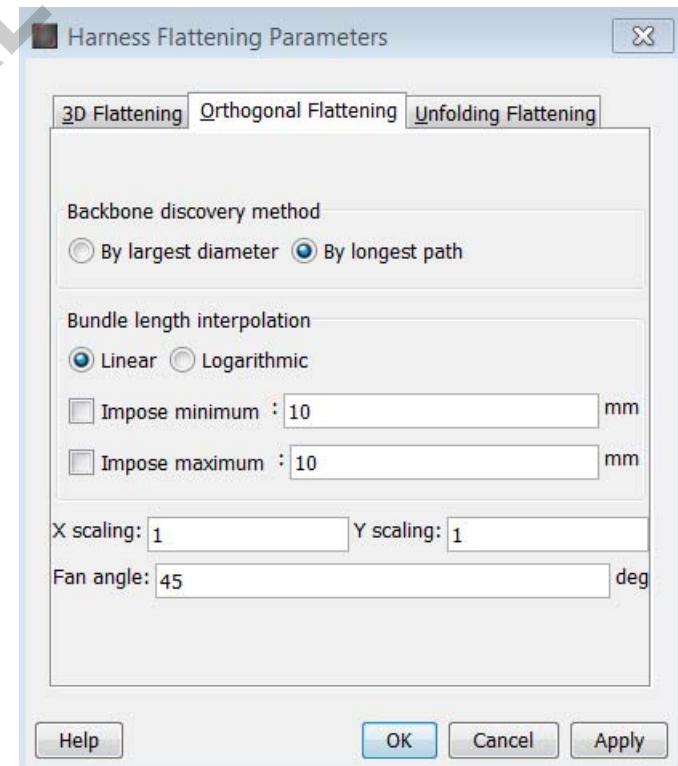
⇒ **User Defined:** allows the user to specify the projection plane

Tilting of Close Nodes

The minimum distance allowed between nodes. This is to help stop nodes overlapping.

Orthogonal Flattening

This lays out the branch configuration making sure all branches are vertical or horizontal where possible.



Backbone discovery method: Identifies how the backbone of the harness is identified i.e. **by largest diameter** where the bundle with the largest diameter is

used as the backbone and the next bundle with the second largest diameter is added on to it and so on. Alternatively it can be obtained **by longest path** where the longest branch is the backbone (the longest branch between 2 nodes)

Linear: displays the lengths as they appear in the imported model e.g. a branch 1500mm is graphically proportionally longer than 1000mm

Logarithmic: displays the lengths using logarithmic smoothing e.g. 1500mm is potentially graphically the same as 1000mm

Impose minimum/maximum: specifies the minimum and maximum graphical lengths of bundles to stop displaying bundles very small or very long. The distribution of bundle lengths will be (linearly or logarithmically) scaled between those 2 values.

X/Y Scaling: graphically scales the drawing as required

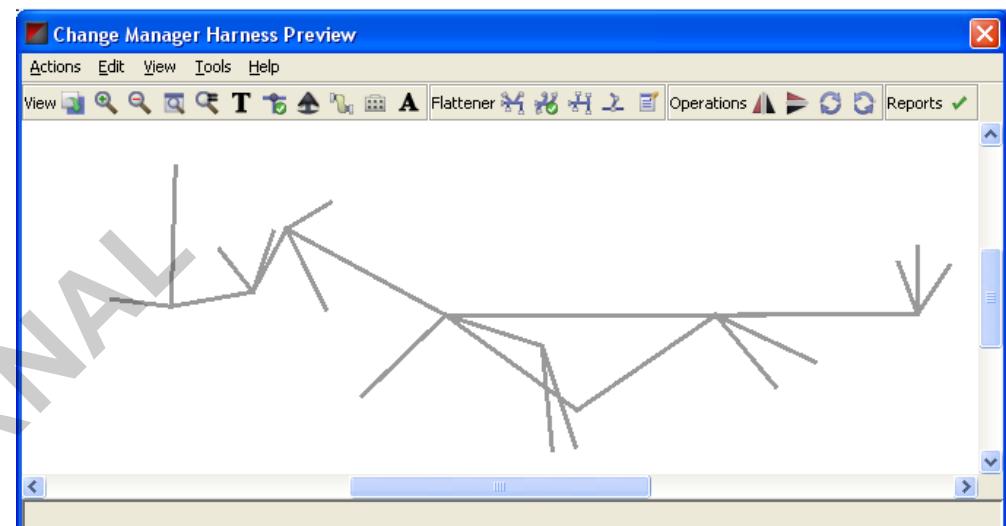
Fan Angle: where more than 3 branches exit the same breakout point the system will incrementally place the extra branches maintaining the angle specified

Unfolding Flattening

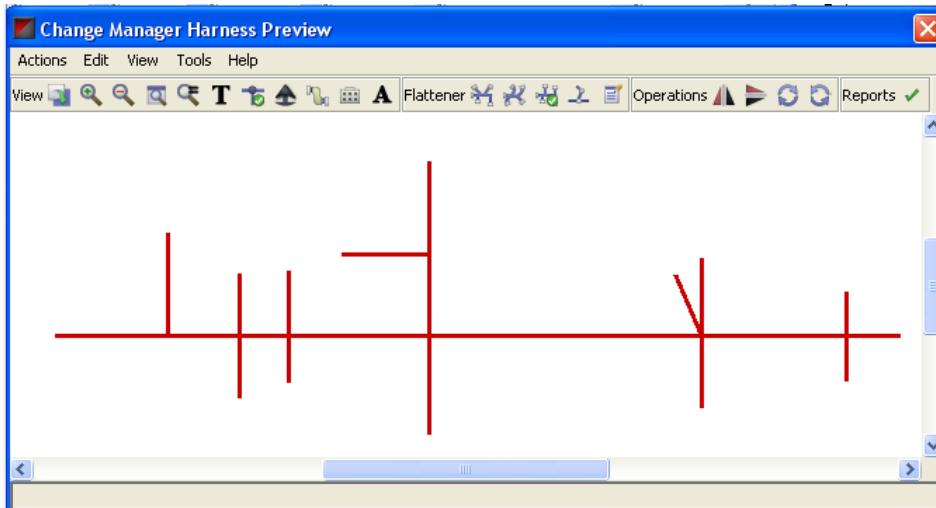
This section of the harness flattening parameters form is used if upon import the position /rotation of the components and bundles must be taken into consideration, for example where connectors with backshells and key codes are imported, it is vital that the correct positioning is retained to allow for accurate bundle length calculations.

Modifying the View

- ⇒ Select **Edit / 3D Flattening** or choose the icon  This uses the 3D flattening parameters to layout the branches

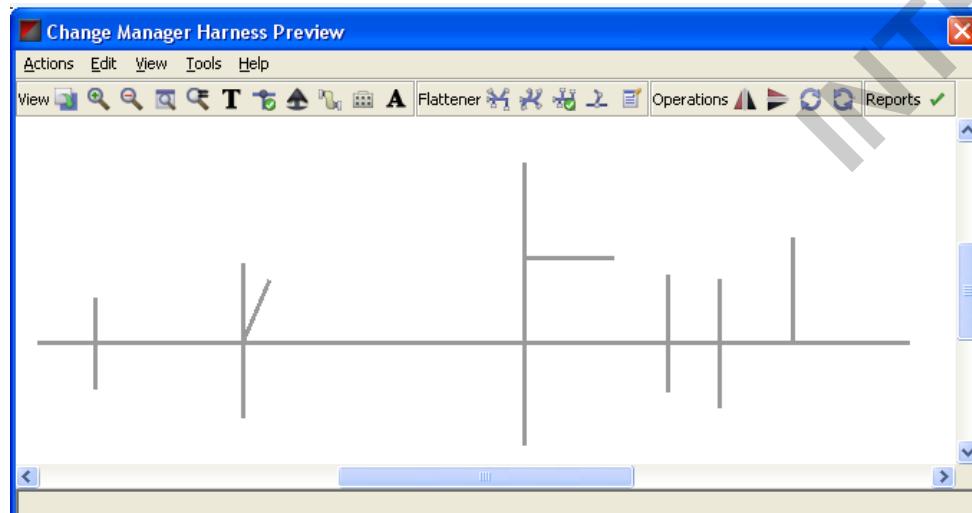
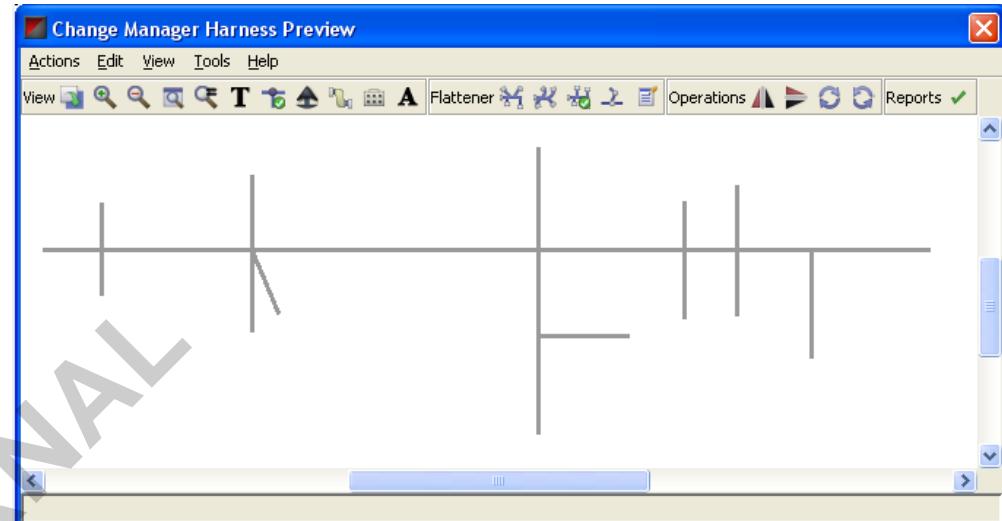


- ⇒ Select **Edit / Orthogonal Flattening** or choose the icon  This uses the Orthogonal Flattening parameters to layout all the branches vertically or horizontally

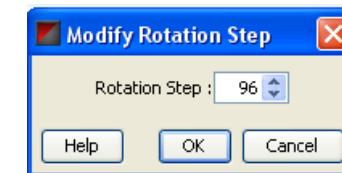


⇒ Select **Edit / Horizontal Flip** or choose the icon to flip the whole design about the y axis

⇒ Select **Edit / Vertical Flip** or choose the icon to flip the whole design about the x axis



⇒ Select **Edit / Rotation Step** to specify the increment that the whole design should be rotated



⇒ Select **Edit / Clockwise Rotate** or choose the icon to rotate the whole design in a clockwise direction

⇒ Select **Edit / Anti-Clockwise Rotate** or choose the icon to rotate the whole design in an anti-clockwise direction

Displaying Items

Select **View / Show Labels** or the icon  to show or hide the text labels.

Select **View / Show Splices** or the icon  to show or hide all splices.

Select **View / Show Fixings** or the icon  to show or hide any fixings (clips/grommets).

Select **View / Show Protections** or the icon  to show or hide any protections/insulations.

Select **View / Show Connectors** or the icon  to show or hide any connectors.

Bridge Options

Translation

Translate option expression: Option information will be translated based on mappings defined in the adapters.xml file (not applicable to dsi files)

Translate Codes: This will use any code translation mappings defined within Capital Project and apply them to incoming data

Import

Incoming part number: This indicates the source of the library parts (internal, customer, supplier) and is used to search for the matching part in the Capital Library.

Translate parts: This allows users to specify part numbers to be exported via bridge out or imported via change manager. Supplier or customer can be selected and the user can select the required customer or supplier information.

Processor: Users can select the bridge processor plug-in. Different customers may require a different plug-in to translate their data.

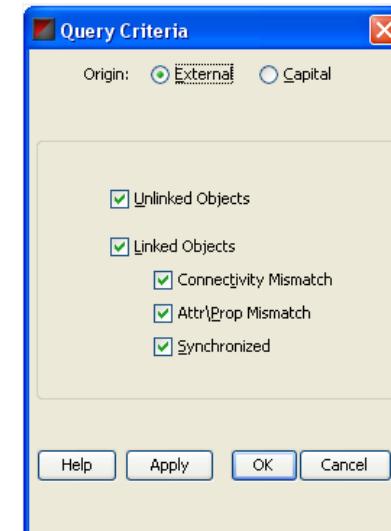
Autolink Criteria

Due to changes in the HarnessXC diagram or the incoming model, some objects may be unlinked i.e. have no link to the data within Capital HarnessXC. All unlinked branches are shown in green within the preview pane.

Query Editor

The **Filter** button allows the users to quickly determine the differences between the source and target data e.g.

- ⇒ what has been added, deleted or changed in either model
- ⇒ if the lengths have changed
- ⇒ bundles that have been affected by a topology change



To find new bundles, select MCAD **Unlinked** and the system will display only new bundles (not yet linked) in the Change Manager and in the Preview (unlinked objects are shown in green, whereas linked objects are colored in grey).

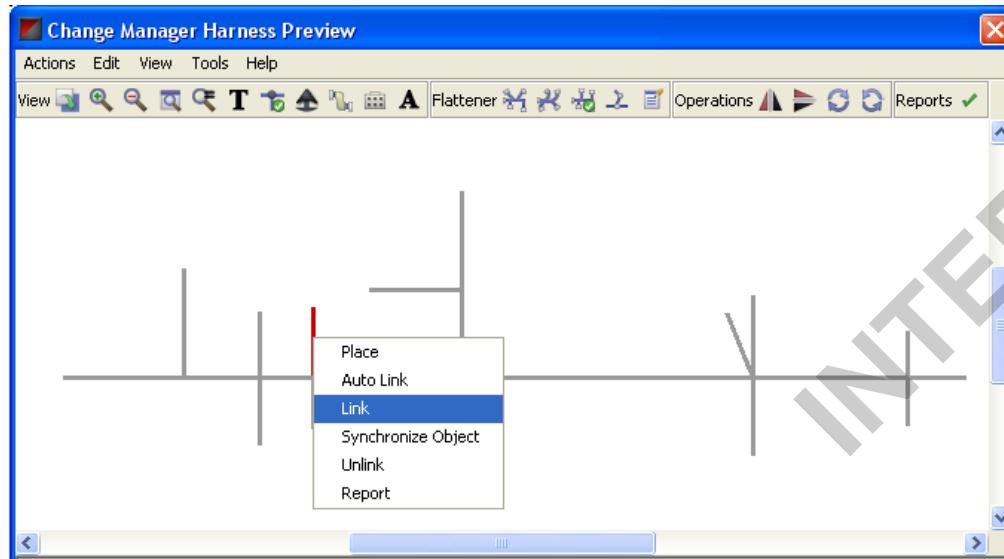
Note

Link and Unlink functions are used after the first import.

Link Objects

It is possible to manually link an object from the CATIA model to the Capital HarnessXC object.

- ⇒ Select the object in the **Change Manager pane** or the preview pane
- ⇒ Select **Actions / Link** or **Right Click** and select **Link**

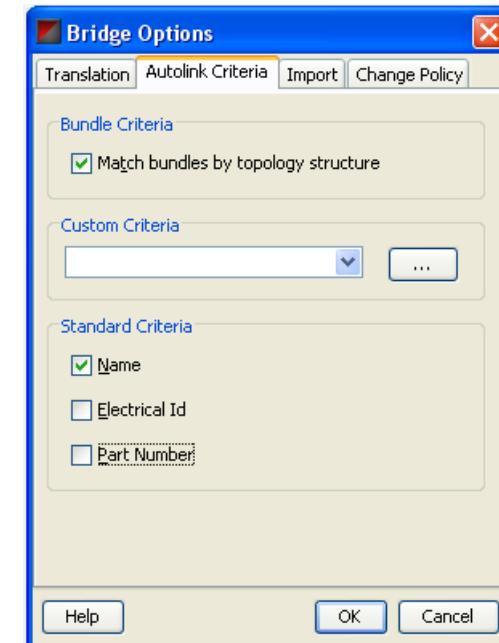


- ⇒ Select the object to link to in the **Capital HarnessXC design**

AutoLink Objects

It is possible to automatically link objects from the incoming model to the Capital HarnessXC object.

- ⇒ Select the **object** in the **Change Manager pane** or the preview pane
- ⇒ Select **Actions / Autolink** or select the autolink icon in the shortcut bar



The system can automatically link objects that have matching properties such as the Object Name, Electrical ID (UID) or Part Number.

Match bundles by topology structure

This will compare the existing bundle structure to the incoming bundle information. Any pre-existing bundles whose structure has been modified will be updated. An example would be where a new bundle is inserted in a pre-existing bundle, thus splitting the original bundle. The system will first autolink connectors at the end nodes then work out the bundle information connected to them. All nodes affected by the structure change are highlighted in the preview pane.

Custom Criteria:

Allows users to select a custom plug-in to apply to the data as it is bridged in.

Standard Criteria:

Name: If selected objects names will be matched

Electrical ID: If selected objects will be matched based on their electrical id (not recommended as the electrical ids may change frequently)

Part Number: If selected the library part associated to the harness objects will be used to update the source harness. This relies on the target and source library information being up to date.

Autolink harness objects

Once the autolink criteria have been defined the user can run the autolink function:

- ⇒ Right click the incoming harness reference
- ⇒ Select Autolink Harness objects or select the autolink harness objects icon in the shortcut bar: 

The incoming harness objects will be linked to pre-existing bundles. The bundles may have been moved to a new location during the import or added as new bundles during import and bundle end objects introduced or changed.

The tool uses the pre-defined autolink criteria to help place objects and update the bundle structure.

Import:

Merge all redundant nodes and bundles: bundles are applicable for merge if they have the same diameter, the same max diameter, the same bend radius and both have distinct user defined or common properties

Remove small bundle loops: the bundles in the loop must have a length smaller than the specified minimum bundle length

Merge all adjacent protections: Adjacent insulations on the same bundle are merged if the insulation type, type code, and part number are the same

Validate against schema: Used when a custom bridge adapter is used in a non-standard flow. Data created by the bridge is validated.

Process Mode: Incremental – this option will update the design with additional imported information, no objects are deleted even if the change policy allows it

Complete – this option will replace all existing data with the new incoming data. If restrictions on deletion are defined in the change policy they are adhered to. If a module design references are removed in the incoming file they will be deleted in the target design.

Harness Repository – This is the directory where x2ml external files are held

Change Policy

The change policy is defined in Capital Project. It is used to specify what objects will be updated, deleted or created upon import. Each harness object type will have its own definition within the change policy.

Synchronize harness design attributes and properties: This will sync the harness part number and other harness design attributes and properties such as manufacturing site, customer names, as long as they exist in the incoming library. If the Sync with customer part number is selected the harness part number will sync from the take the incoming customer part number.

Any user defined harness properties will be deleted prior to import if the allow design user property delete is selected.

Synchronize cavity components: This will ensure connector cavity components are brought in with the import.

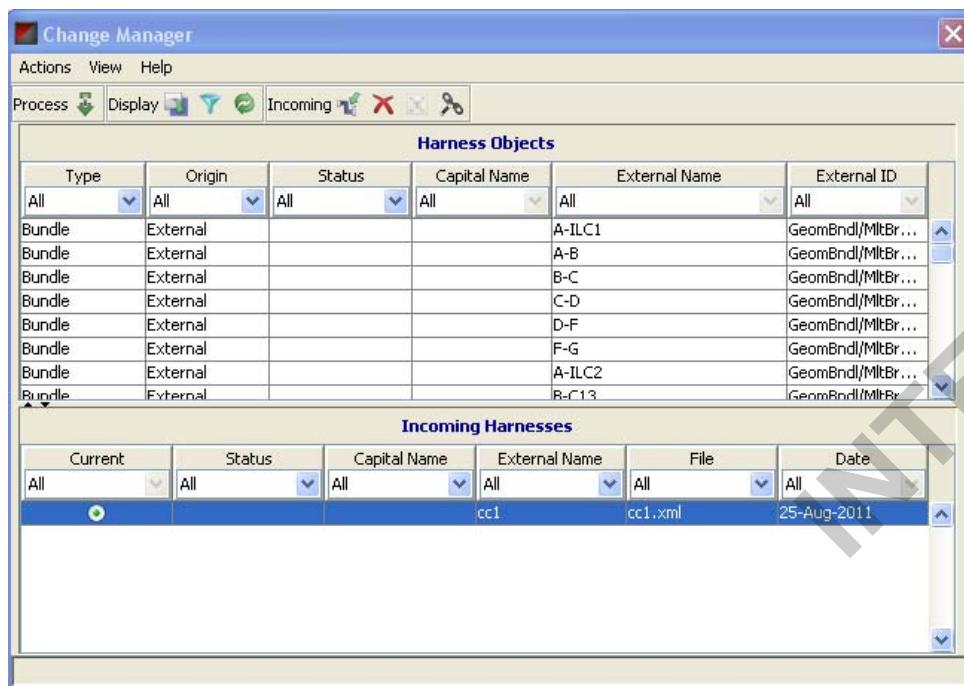
Allow addition of mandatory components: If left unchecked any mandatory additional components will not be added even if they are present in the incoming data. If checked all mandatory additional data will be added.

INTERNAL

Change Manager

If the Bridge in function is not delivering the expected import results (typically with any subsequent import of data), users can launch the Change Manager. This tool gives the ability to view incoming data and identify any changes that have occurred.

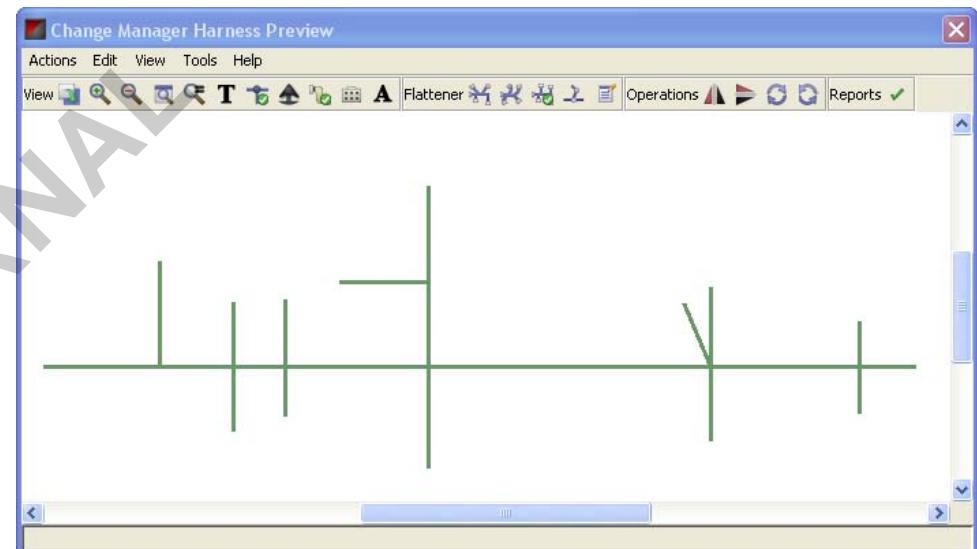
- ⇒ Select **Bridge / Change Manager**



- ⇒ **Incoming tab:** specifies where the harness xml files are stored. In connected mode, the CATIA model sends the xml files here
- ⇒ **Harness Objects:** shows the bundles and components and their link status
- ⇒ **Display:** Will display the preview pane for the incoming harness content
- ⇒ **Process:** will process the incoming data and place the information on the harness diagram

- ⇒ **Filter:** will allow users to specify filter criteria for incoming data (linked and unlinked object display)
- ⇒ **Delete File:** Will delete the current selected file
- ⇒ **Autolink Harness Objects:** Will link objects in the incoming file to the source harness using criteria specified in the bridge options

It is possible to view the data as shown above or in the preview pane by selecting **View / Change Manager or the Display tab:**



Select Harness for Import

Available harnesses will be automatically displayed if they are in the defined Harness repository. The harness repository can be defined under Incoming/Bridge Options/Import/ In connected mode incoming files will be automatically placed in this location.

Alternatively select the **Incoming** button to navigate to the xml file required for import.

The incoming harness can be linked to an existing harness design:

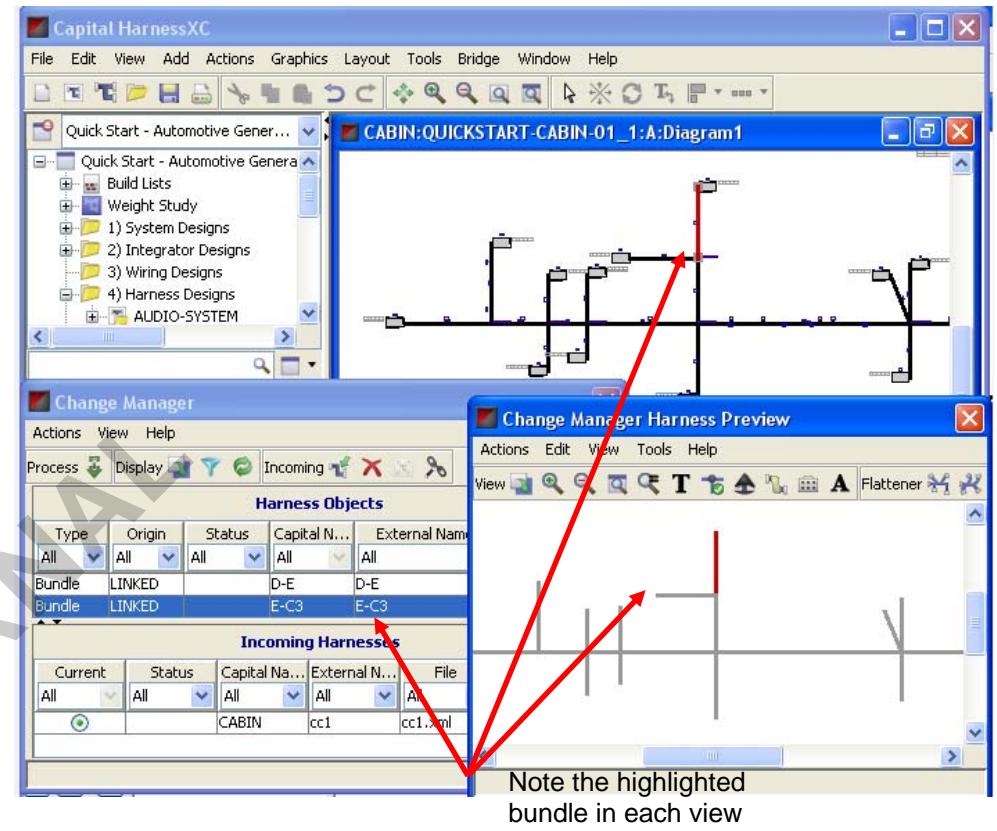
- ⇒ Right click the incoming file
- ⇒ Select Link harness
- ⇒ Select the harness design you wish to link to

The Capital Name field is now populated with the harness design name

Cross-Highlighting

It is possible to highlight an object (bundle, connector etc) in one view and this object will be highlighted in all other views. The different views are:

- ⇒ CATIA model – if on same machine
- ⇒ Change Manager View
- ⇒ Preview view
- ⇒ Capital HarnessXC diagram



Updating the harness design

All objects from the external file need to be placed in the harness diagram:

To place objects into the harness diagram:

- ⇒ Select Actions / Process

The process action will automatically assess the content of the incoming file and compare it with the existing diagram content. Any new or modified objects or bundles will be inserted, amended or deleted according to the incoming data. The update is made inline with the pre-defined bridge options.

The process action takes the following steps:

- ⇒ Run Link/Autolink on the incoming data– this action will use the autolink criteria defined in the bridge options
- ⇒ Synchronize – this updates existing objects
- ⇒ Delete obsolete objects from the target harness design
- ⇒ Place new objects in the target harness design

A linked object stores a reference to the object it is associated with. The action of linking sets this reference on the object and additionally synchronizes it.

The action of synchronizing copies the object properties to the object it is linked with.

Note

Once the harness has been processed, any changes to the view (flip, rotate etc) will not alter the harness diagram.

After placement, if the view is changed (flipped etc) and a new branch is added, this new branch will be positioned as if the harness diagram has been flipped etc.

Manual placement of objects:

In a minority of cases users may need to manually place or link an object on the harness diagram

- ⇒ Highlight the object to be placed
- ⇒ Right click and select Place or link/autolink

Perform the place on harness objects in the following order:

- ⇒ Connectors
- ⇒ Bundles
- ⇒ Insulations
- ⇒ Clips
- ⇒ Grommets
- ⇒ Other objects

Property Mismatch

There may be occasions where the objects are linked but a property is different e.g. a bundle exists in both models and has been linked but the bundle length is different (for instance, it has been changed in CATIA).

This will be shown in the Change Manager as a **Property Mismatch**.

Harness Objects						
Type	Origin	Status	CHS Name	MCAD Name	MCAD ID	
Bundle	LINKED		BUN202	C-D	GeomBndl/MltBndl...	▲
Bundle	LINKED		BUN201	C-C11	GeomBndl/MltBndl...	
Bundle	LINKED		BUN206	D-H	GeomBndl/MltBndl...	
Bundle	LINKED		BUN207	D-E	GeomBndl/MltBndl...	
Bundle	LINKED		BUN215	F-C9	GeomBndl/MltBndl...	
Bundle	LINKED	Property Mismatch	BUN217	A-ILC1	GeomBndl/MltBndl...	
Bundle	LINKED		BUN205	F-C8	GeomBndl/MltBndl...	
Bundle	LINKED		BUN204	G-C6	GeomBndl/MltBndl...	▼

To resolve this issue, synchronize the data by

- ⇒ Select the mismatched object/s
- ⇒ Select **Actions / Synchronize** or **Right Click and Synchronize**

Note

When objects are linked, the system automatically performs the synchronization

Connectivity Mismatch

There may be occasions when the layout changes e.g. a branch exists in CATIA but not in Capital HarnessXC, length changes etc. This will be shown as follows:

Missing Branch: No Capital Name

Harness Objects						
Type	Origin	Status	CHS Name	External Name	External ID	
All	All	All	All	All	All	
Bundle	LINKED	Connectivity Mismatch	D-E	D-E	GeomBndl/...	
Bundle	External		E-C3	E-C3	GeomBndl/...	
Bundle	LINKED	Connectivity Mismatch	E-C2	E-C2	GeomBndl/...	
Bundle	LINKED	Connectivity Mismatch	F-C4	F-C4	GeomBndl/...	
Bundle	LINKED	Connectivity Mismatch	F-C9	F-C9	GeomBndl/...	

Connected Branches: Topology Mismatch

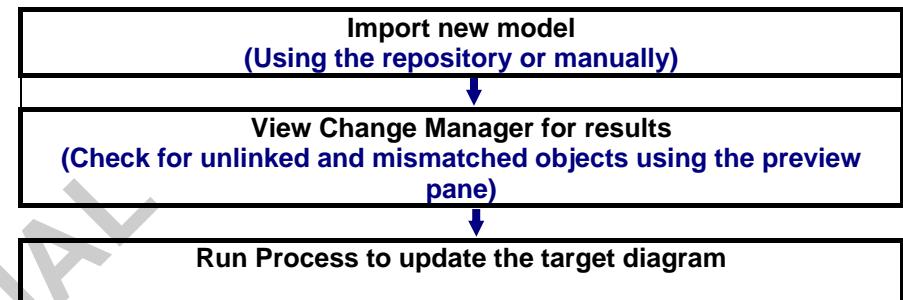
Harness Objects						
Type	Origin	Status	CHS Name	External Name	External ID	
All	All	All	All	All	All	
Bundle	LINKED	Connectivity Mismatch	A-ILC1	A-ILC1	GeomBndl/...	
Bundle	LINKED	Connectivity Mismatch	A-B	A-B	GeomBndl/...	
Bundle	LINKED	Connectivity Mismatch	B-C	B-C	GeomBndl/...	
Bundle	LINKED	Connectivity Mismatch	C-D	C-D	GeomBndl/...	
Bundle	LINKED	Synchronize Bundle Topology Manually	D-F	D-F	GeomBndl/...	

To resolve the issue **Place** the additional branch.

Possible Flow for Changes

Below shows a possible flow when changes have occurred from the imported model.

Using the preview pane or the change manager, view for unlinked objects and objects that are mismatched with the Capital HarnessXC design.



Exercise 1:

Import a CATIA Model.
See Exercise Sheet for importing CATIA V5 data

Chapter 13

Styling Editor

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Introduction

Introduction

A major objective for Capital HarnessXC is to separate graphical style from data content. The styling editor enables the user to configure how harness data is rendered on a drawing and to save these preferences for re-use on other data sets.

Each object can have specific styling performed (decorations, graphics and shape).

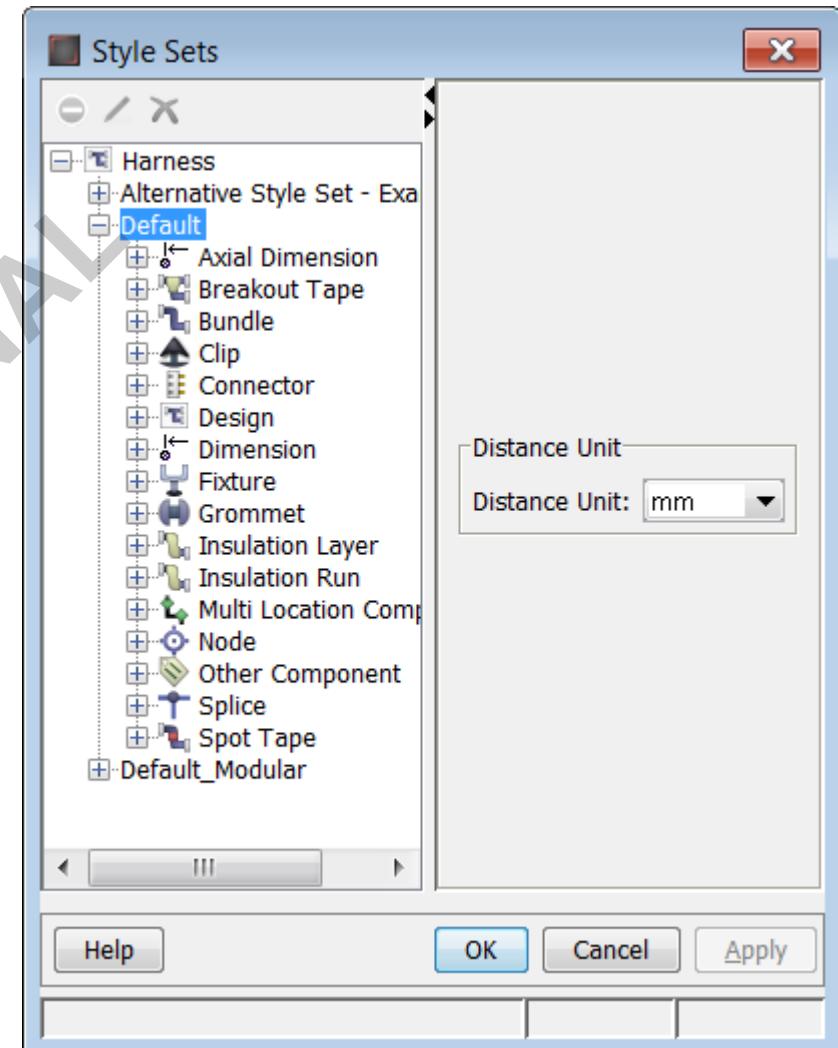
Each installation of the Capital tool will automatically receive a free style set with some basic styling functions. If users wish to define a more advanced style that would include queries and conditions, this will require an advanced styling license.

Style sets can be edited from within a harness diagram by right clicking an object or in free space and selecting **Style/Edit Style**

Your instructor will now deliver the Styling Introduction powerpoint to introduce the concept of styling

Default Preferences

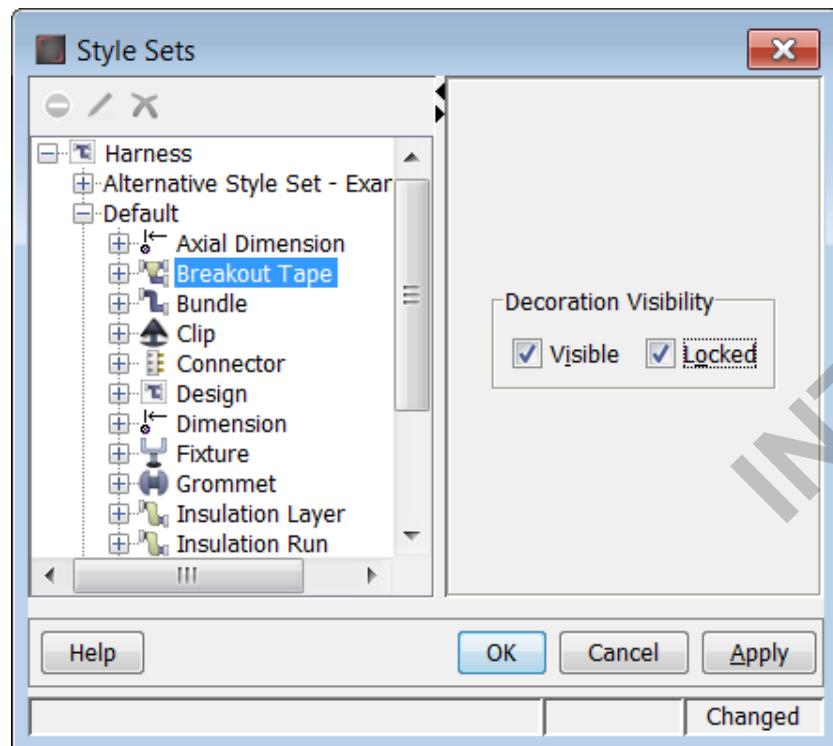
The styling parameters are found under **Edit / Style / Style sets**



Each system will be loaded with a default style set. Additional style sets can be loaded if an additional styling license has been purchased.

All objects available for styling are listed in the tree browser.

Style settings for all objects may be locked to limit user ability to override styling within individual diagrams.



When expanded each of the objects has a certain set of styling parameters that can be defined.

Decorations

This is optional text, symbols & tables that the user selects, formats and positions relative to a nominal origin.

Attributes: is any attribute applicable to the selected object type and is in the form of text e.g. name, part number, length, border zone information etc

Composite Texts: Where composite name text is to be used the composition can be defined here

Properties: is any property name as specified by the user and is in the form of text.

Query Results: is the result of a query, selected from the system queries and is in the form of text e.g. display the number of wires at a connector, or does this connector have a backshell?

Symbols: is any comment symbol, selected by the user from the available symbol libraries assigned to the style set

Tables: is any table applicable to the selected object type i.e. connector cavity tables, wire tables

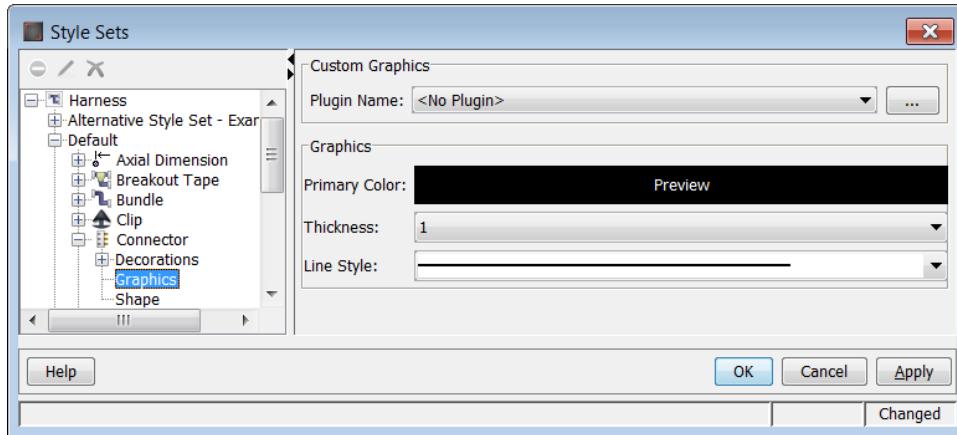
The user can choose to add a new decoration, as explained above, by choosing the **Add** button and selecting from the available list.

- ⇒ When adding Attributes or Symbols, the system will present all attributes as shown on the General tab when adding each object or all symbols in the associated libraries. This list cannot be changed by the user
- ⇒ When adding Properties or Query Results, the system will present all properties/query results that have been pre-defined. The user can also create a new one if required

The user can remove or rename any of these items once added. Items that are renamed do not affect the pre-defined lists.

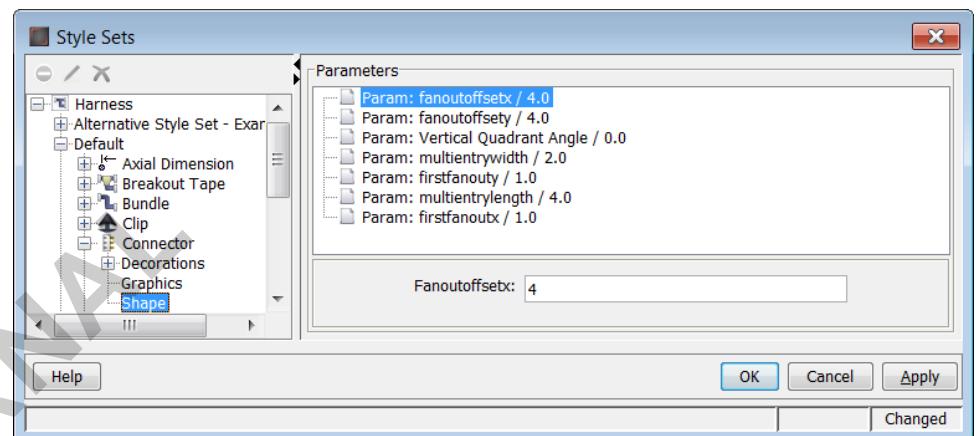
Graphics

This is the default linestyle settings for bundles and dimensions. The user selects from a list of pre-defined linestyles.



Shape

This section contains editable parameters relative to the object to be styled e.g. fan-out offsets for modular connectors, dimension line offset dimensions etc.



The following explains what can be styled for each object shape.

Bundles:

- ⇒ Show inter node dimensions – true/false
- ⇒ Define the dimension precision
- ⇒ Show zero length bundles –true/false
- ⇒ Show width under insulation
- ⇒ Remove Trailing Zeros

Connector:

- ⇒ Specify the fanouts for modular connectors and the 'child' branches

Dimension:

- ⇒ First dimension line offset distance from bundle
- ⇒ Subsequent dimension lines offset from previous dimension line
- ⇒ Reference style dimensions – length of horizontal line
- ⇒ Reference style dimensions – vertical line length offset extension from the horizontal line

Insulation Layer:

- ⇒ For each individual insulation type a linestyle shape may be defined

Insulation Run:

- ⇒ State which linestyle is to be used for each insulation based on a query type

Node:

- ⇒ The user shall be able to specify named glyphs to be displayed at a node based on the design behavior at that node. As the behavior at a node changes in the design the glyph displayed shall dynamically change. There will be an order of precedence to define which glyph is displayed when there is more than one behavior at a node.

- Normal node
- Reference node
- Marker node
- Connector
- Splice
- Junction
- Sealed splice
- Other component
- Junction tape
- Fixing
- Covering

Spot Tape:

- ⇒ This glyph will be vertically scaled to represent its layer position relative to any other insulation layers passing through/terminating at the node.
- ⇒ Width of the glyph

Applying the style set

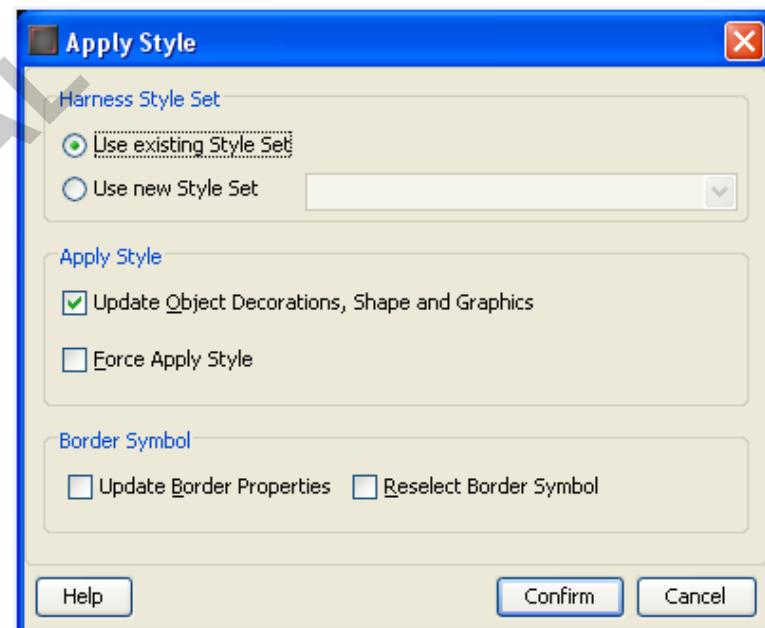
The style set can be applied to individual diagram objects:

- ⇒ Right click the object

- ⇒ Select Edit style/apply style

A style set may be assigned to selected diagrams:

- ⇒ Open the project browser
- ⇒ Select the diagrams to be assigned the style set
- ⇒ Right click/Apply style/continue
- ⇒ Define the settings for the style application (style set choice, update or force style, update border etc)
- ⇒ Click Confirm

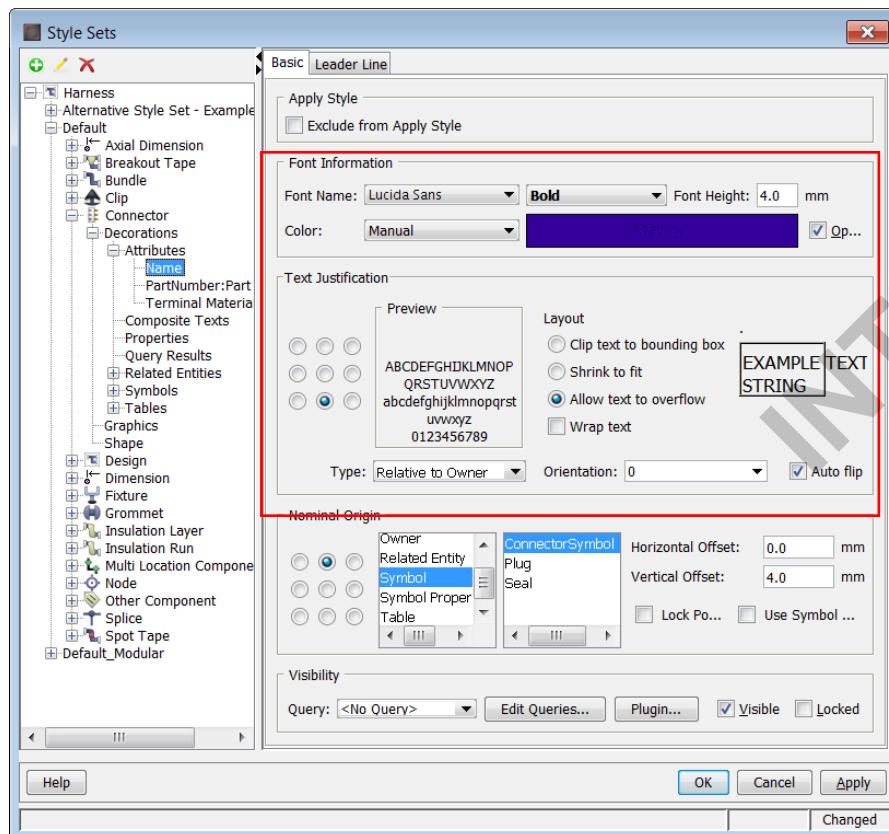


Styling the Decorations

This section will describe a number of basic styling functions available for use within the edit style set dialog

Text formatting

Any text placed on the diagram can be styled as follows (this may be an attribute, property or query result). Here is an example of formatting a connector name attribute:



- ⇒ Open the relevant style set and navigate to the connector name attribute
- ⇒ In the font name field select the required Font Name - Arial etc.
- ⇒ Define the style: Normal, Bold, Italic, Bold/Italic
- ⇒ Define the height: – using preferred units for the style set
- ⇒ Specify the color
- ⇒ Specify opacity – Yes/No
- ⇒ Define orientation

Exclusion of applicable style

Often users will make minor changes to diagrams, such as moving objects to accommodate spacing issues or adding additional notes to text. When the Apply Style is run these changes will be overwritten. To avoid this an exclude from Apply Style option is available.

If a particular object type (for example all connector names) is to be ignored during the Apply Style action, the 'Exclude from Apply Style' box should be checked in the styling editor. If individual objects on a diagram are to be modified and users do not wish for them to be changed during the Apply Style process, but do want other objects of the same type to change, these objects can be excluded from the apply style process on an individual basis:

- ⇒ Right click the object in the diagram and select Style/Fix

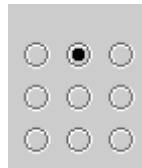
All other instances of that object type will have the styling applied when users click Apply style but that object will not. To remove the fix simply select style/unfix.

If the styling is to be temporarily forced on an object with a fix:

- ⇒ Right click the object/Style/Force Apply Style

This allows users to make modifications to individual object types on a diagram without risking an overwrite when apply style is run.

Justification



Each object to be placed requires an insertion datum point. This is based on a nine point justification grid.

Fit and Wrapping

When a text value is inserted on a drawing (attribute name, property name etc) it may be modified to fit into an available space. A text box is temporarily visible on the diagram when and can be used to help ensure that text fits into a specific area.

The styling tool offers a number of options to define how text display should respond when the text box is modified:

Clip: If the text box is made smaller on the diagram this option will render any text that spills out of the given text box size invisible. If the box is made larger the text will be visible again.

Shrink to Fit: If the text box is made smaller on the diagram this option ensures that all text is still visible by shrinking the size of the text to fit the box. This applies to the horizontal flow of text only.

Overflow: – If the text box is made smaller this option allows all text to remain visible by letting it overflow outside of the box perimeter. This is the default setting

Wrap Text – Ensures that the text is broken down to multiple lines.

Nominal Origin

It is necessary to define the relative position for the placement of the decoration to its owner or any other decoration defined. This is known as the nominal origin.

A datum point on the target decoration must be defined (the nominal origin), also using the nine point grid.

Offsets for the datum

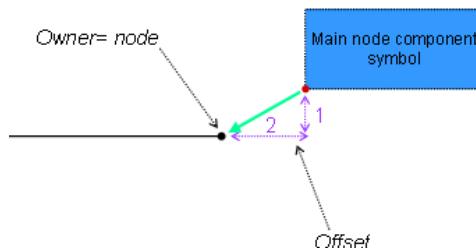
Using the vertical and horizontal offset values, the user defines the relative positions between the object and its nominal origin.

Note

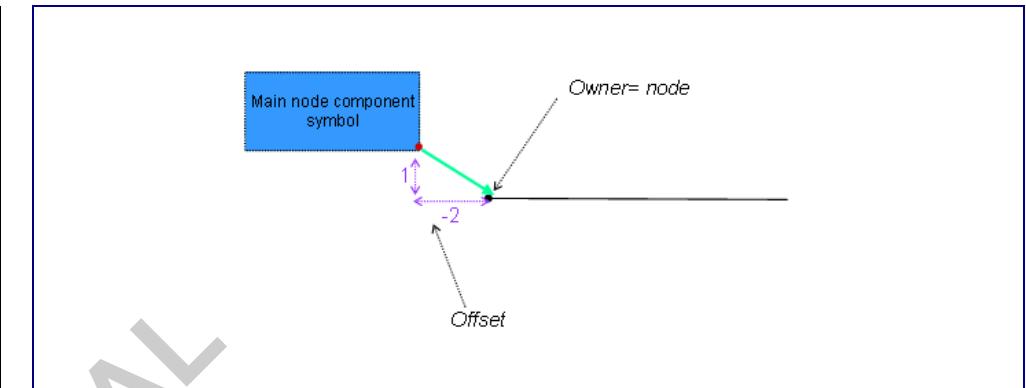
By assuming the bundle is drawn from left to right and any node objects being styled are at the right hand side of the bundle, the x and y offsets will be as expected.

When the objects are actually placed at the other side of the bundle in the diagram the x and y offsets will mirror correctly.

e.g. assume layout is left to right when defining the offsets:



If drawn the other way in the diagram the symbol will be placed as follows.



Owner as the nominal origin

The owner is dependent on the type of object that is selected.

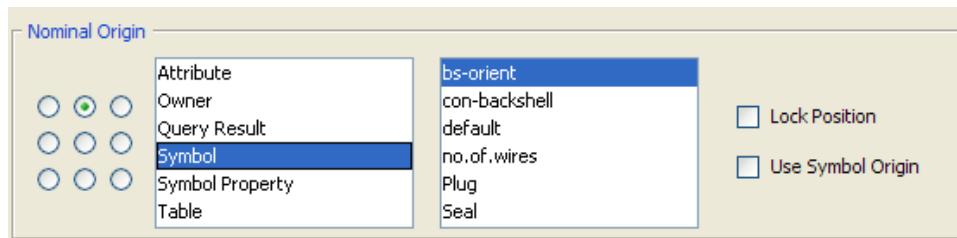
- ⇒ For a **node inserted object** the owner is the node itself. In this instance the nine point grid is not used and the centre of the node is taken as the nominal origin
- ⇒ For a **linear/run based object** (bundle, insulation, dimension) the owner is the entire run. The user can also specify which segment of the run shall be used to represent the justification grid.

In the style manager section it is possible to specify the segment that the decoration to be placed (under line decorations). Either the centre most or longest segment of the bundle can be chosen

The nine point grid is used to represent the symbol either side of the bundle.

Attribute/Symbol as the nominal origin

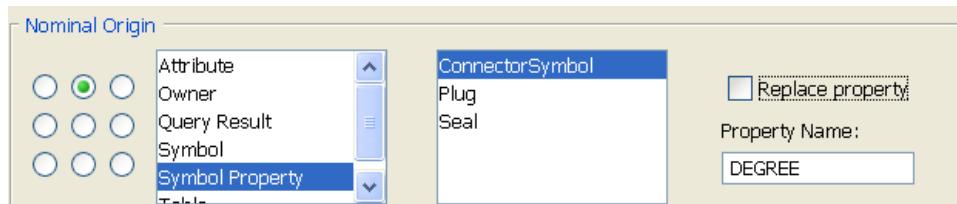
If a previously styled decoration (either an attribute or symbol) is chosen as the target decoration (e.g. the name of the connector is to be located relative to the part number **attribute** or the main connector **symbol**) the system will present all relevant decorations that have been added to the selected object (e.g. connector). The user must then select which actual attribute or symbol they wish to use as the target decoration.



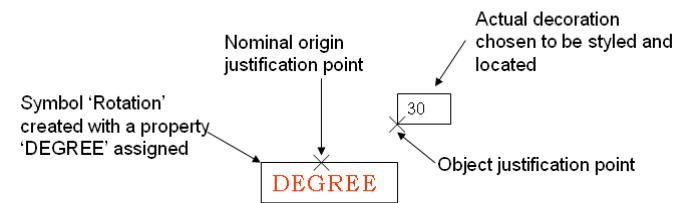
The Lock Position groups all aspects of the object together so that if it is moved then all associated objects are moved with it.

Symbol Property as the nominal origin

If a symbol property is chosen as the target decoration the user must enter the property name that has been assigned to the selected symbol for use as the nominal origin.



In the above example, the user is placing the decoration, relative to a symbol property 'DEGREE' that has been assigned to the symbol 'connector symbol'. The user can choose to place the decoration using its own justification point and the nominal origin of the property connector function (see below). The property will be visible or not depending on how it has been defined in Capital Symbol.



Alternatively the user can choose to use the position and formatting of the actual property within the symbol rather than using it as a datum for placement which means in the above example, the text 30, will be shown inside the rectangle and will be red with the correct font type. By selecting **Replace Property** the text formatting and position controls will be disabled as they are not applicable. This is particularly useful for placing various entities on the border.

Visibility of the Decoration

The visibility of the decoration (attribute, property, query result, symbol or table) can be controlled by switching visibility on or off.



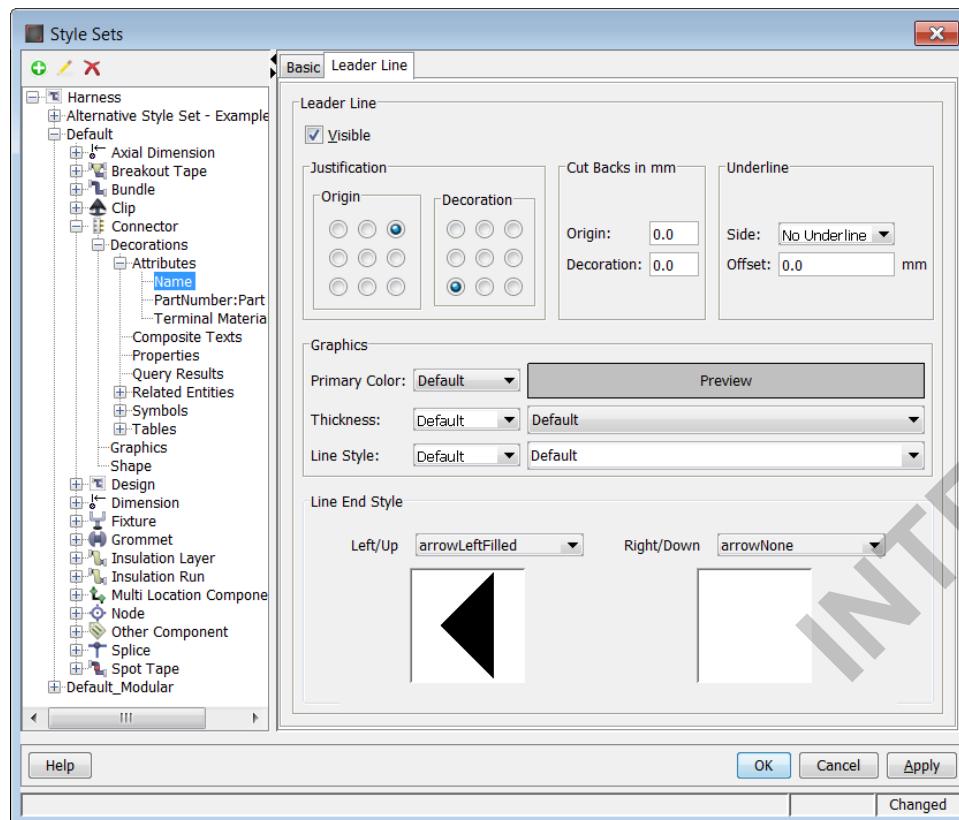
Additionally the visibility can be controlled by a query to display the selected decoration when a particular query is met.

E.g. show a symbol to identify a connector is sealed
show a symbol to display the orientation of a clip

The visibility of the decoration can be **Locked** so that whilst editing the diagram it would not be possible to change the visibility of the object.

Leader Lines

A leader line can be specified and drawn between the object decoration and its nominal origin.



The justification points to place the leader line against the object decoration and nominal origin are specified using the standard nine point grid.

Formatting the Leader Line

The visibility of the leader line is controlled by a check box – on/off.

The leader line can have the standard linestyle formatted

- ⇒ Color
- ⇒ Thickness
- ⇒ Linestyle – selected from a choice of pre-defined linestyles
- ⇒ Cut backs – defined for either end of the leader line so as to leave a gap between the line and attached objects
- ⇒ Underline - an additional line be rendered on the left, right, top or bottom of the styled decoration at an offset dimension specified by the user

Your instructor will now deliver the Styling examples powerpoint

Configure Component Symbols

There are numerous ways to configure which symbols should be displayed on the diagram.

Symbol Library

If users do not want to use the default libraries, user defined symbol libraries can be associated to a project for use in diagrams:

- ⇒ Open style sets
- ⇒ Select Design
- ⇒ Select **General settings / library sets**
- ⇒ Click Add at the bottom of the browser and add the required symbol libraries to the design style set

More than one library can be selected and a hierarchy defined such that when searching for a symbol the system looks in each of the specified libraries in sequence. Via the application, the user can select a different symbol from a different symbol library that has been assigned.

The system will first look to place a symbol with the same name as the library part number inserted on the diagram – note that this part does not need to have a symbol associated to it in library. The system will search the symbol libraries assigned to the Design, in sequence, to find an appropriate symbol. If a symbol matching the library part number is not found, a default symbol may be placed.

A symbol can be set to appear or not based on the results of a query. Queries are created to investigate the drawing content and respond with an appropriate action depending on the information gained. An example of a query may be to investigate if a component is a ring terminal, if so a ring terminal symbol may be placed on the diagram, if not a single cavity connector symbol will be inserted.

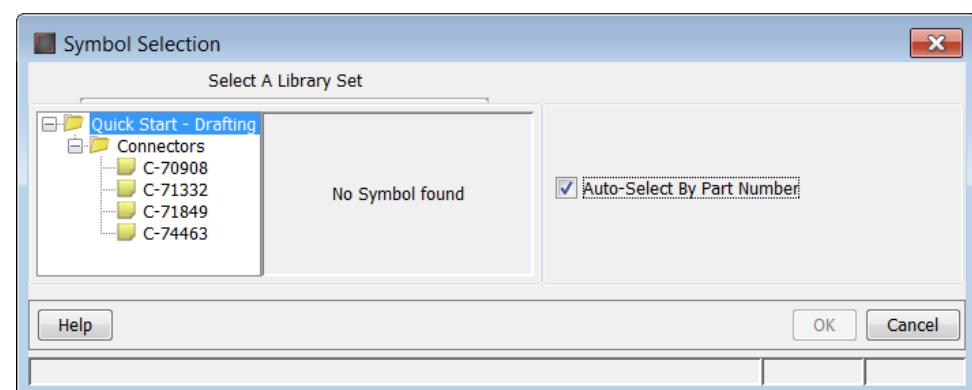
Multi-Location Components (MLC)

When defining styling for multi-location node components the user will have the option to style two distinct sets of decorations; one set that will use the first node as the owner node and one set that will be rendered at all subsequent nodes that the MLC is present at.

Default Symbol

Where no symbol exists for the component in Capital Library or Capital Symbol, it is possible to assign a default symbol (for standard symbols) – this is contained in the default library.

In the style settings under each decoration menu there is a symbol sub menu. Default symbols may be set under this menu. When checking the ‘main symbol decoration’ box, this infers that the selected symbol should be used when no matching part number symbol can be found in the system.



Symbols and related Entities

The notion of connector cavity and wire datums in symbols was introduced earlier in the symbols chapter. We will now take a look at how to implement the datum points in a drawing:

In a connector symbol for example we can define that the symbol displays cavity details and information relating to the wires inserted into the connector cavities. The connector cavity and wire datum tell the system where on the symbol this information should be placed, however, it is up to the user to define in the style set if the wire cavity and its contents should be displayed and if so, what wire information should be shown and what it should look like – this information is referred to as the connector **related entities**

Example:

We may want the cavity names to be displayed on the connector symbol so to set this information we need to do the following:

- ⇒ Insert a connector onto the harness drawing
- ⇒ Associate a library part number to the connector ensuring that the part number matches with the name of a relevant symbol in the symbol library

The connector wire table and symbol should now be placed on the diagram

- ⇒ Now open the **style set / Connector / Decorations / Related Entities**
- ⇒ Click **Add**
- ⇒ Select the related object : **Connector pin**
- ⇒ Layout: **Datum symbol**
- ⇒ Symbol Name : **Connector symbol**
- ⇒ Define Layout order

Hide Unconsumed Decorations: if checked this will ensure that any information held in the connector that exceeds the number of assigned datums will not be shown on the harness diagram, i.e. if there are only 3 cavity datums defined but the connector has 5 cavities then only 3 will be displayed, if the box is unchecked the 2 extra cavities are placed by default at the node.

Name Match: this option will match up any cavity datum that has the same name as the connector cavity

Sort Criteria - defines the order in which the related entity will be placed on the grid.

- Default takes the sequence they were added to the diagram
- By Attribute places objects by the attribute value named in the 'sort by' field
- By Property – will place the objects on the grid using the property value defined in the 'sort by' field

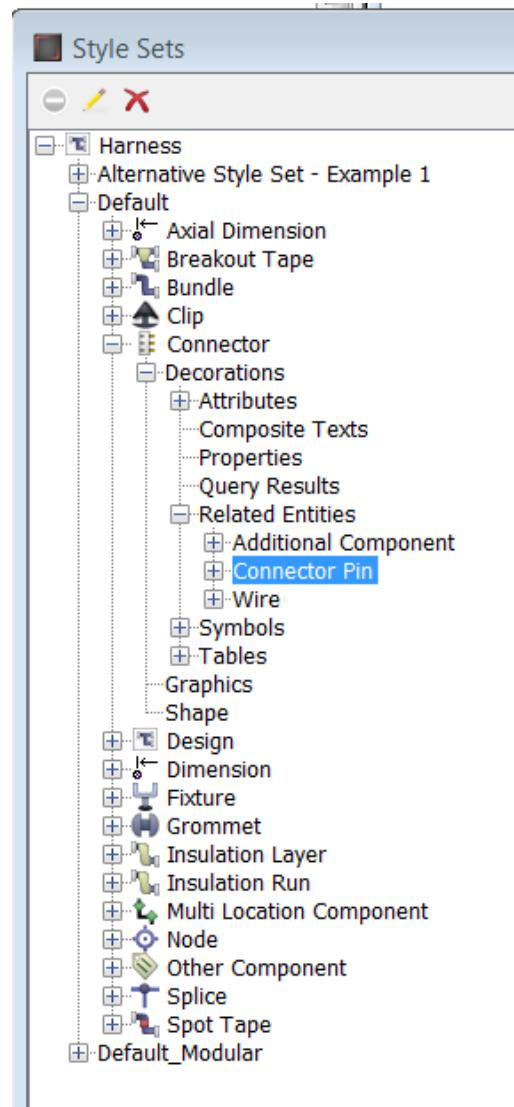
Sort by - defines the attribute or property to used for sorting objects in the grid

Sort Type – Defines whether the objects should be displayed in ascending or descending order – the ordering is alphanumeric

- ⇒ Click **ok** and close the form
- ⇒ Apply the style to your diagram

If you wish to add wire information to the cavities:

- ⇒ Open the style set again
- ⇒ Select the connector pin added above and select **Related Entities**



- ⇒ Click **Add**
- ⇒ Select related object: **Wire**
- ⇒ Click **OK**
- ⇒ Select **Attributes ./ Add**

We now need to define what wire attributes we want to display in the cavity:

- ⇒ Select the required attributes i.e. wire csa, color, material etc

You will now need to define how the various attributes are positioned in relation to one another:

- ⇒ Select an attribute and define it's nominal origin – the first one will be owner and the owner is the connector pin wire datum.
- ⇒ Select the second attribute
- ⇒ Define the nominal origin – this may be the owner or the previously set attribute
- ⇒ Define the offset etc for the attribute
- ⇒ Repeat the steps until all attributes have been positioned

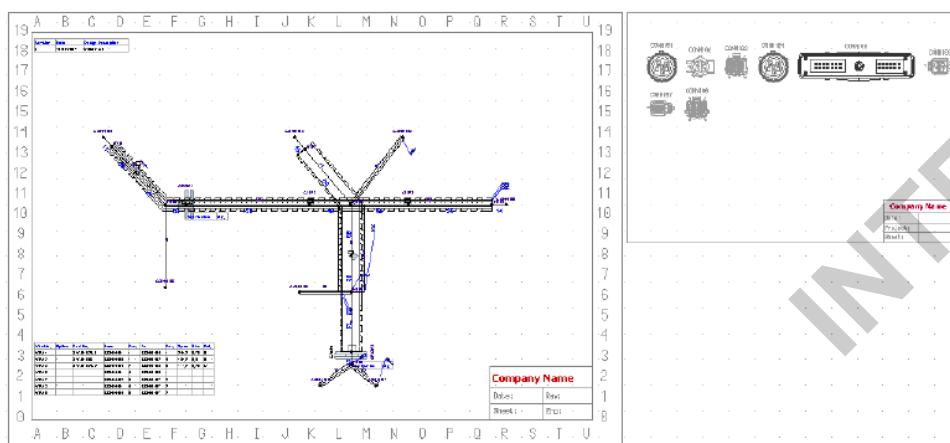
Close the style set and apply the style set to the drawing

Constraint boxes

A constraint box may be inserted on a drawing to contain connector symbols and/or additional component symbol information. Constraint boxes can improve the clarity of information held on a drawing by containing all node component information in a constraint box which may be positioned off to the side of the border.

The symbols chapter detailed how to create a constraint box symbol in Capital Symbol. To implement this on a harness drawing, it is necessary to specify this requirement in the relevant style set.

In the example below a constraint box is placed to the right hand side of the border, similar to the diagram shown:



This constraint box style setting will be assigned to the design itself:

The constraint box needs to be associated to something that will exist in the drawing, associating it to the border is a valid option:

Add a border to the diagram:

- ⇒ Open the relevant style set
- ⇒ **Design / Decorations / symbols / Add**
- ⇒ Select the border symbol you wish to associate / **ok**

The constraint box will now be placed to the right of the border:

- ⇒ Open the desired style set
- ⇒ Navigate to **Design / Decorations / Related Entities / Add**
- ⇒ Select **Connector / OK**
- ⇒ Check the **symbol constrained box**
- ⇒ Define the symbol name to be used for the constraint
- ⇒ Define the subsequent grid placement
- ⇒ Define any advanced grid options
- ⇒ Define the symbol justification
- ⇒ Define the nominal origin – i.e. will the symbol be placed in relation to a particular property or symbol such as the border

Associate a name attribute to the constraint box – this will associate a name to each component symbol placed in the constraint box (using the component name assigned at the placement node)

- ⇒ **Decorations / Related Entities / Connector / Attributes**
- ⇒ **Add / select Name**
- ⇒ Specify the text justification of the name attribute
- ⇒ Specify the nominal origin for the Name attribute

The constraint box is now ready to be placed on the diagram

Bundle Insulation Configuration

The linestyle graphic can be configured as follows.

Creating Linestyles

Under Capital_HOME/Config there is an xml file (harnessLineStyles.xml) that allows the user to create the different linestyles as required.

Each linestyle, with it's unique name, is made up of 3 sections:

- ⇒ one or more end cap glyphs at the start of each insulation run
- ⇒ one or more end cap glyphs at the end of each insulation run
- ⇒ a repeat pattern that creates the linestyle graphic along the bundle run.

Each glyph is based upon a 20x20 point grid. For further information on how to create new linestyles contact technical marketing.

Linestyles associated to Insulation Types

A default linestyle is mapped to each of the hard coded insulation types

- ⇒ Untaped
- ⇒ Overlap Tape
- ⇒ Spiral Tape
- ⇒ Space Insulation
- ⇒ Component
- ⇒ Fixed Tube
- ⇒ Fixed Cut-Back Tube
- ⇒ Fixed Pulled-Back Tube
- ⇒ Selected Tube
- ⇒ Selected Cut-Back Tube
- ⇒ Selected Pulled-Back Tube
- ⇒ Water Hose
- ⇒ Unknown Insulation

In the xml file, each insulation type has a linestyle assigned to it.

Note

The linestyle is not selected from a list and must be manually typed.
The same linestyle can be assigned to multiple insulation types.

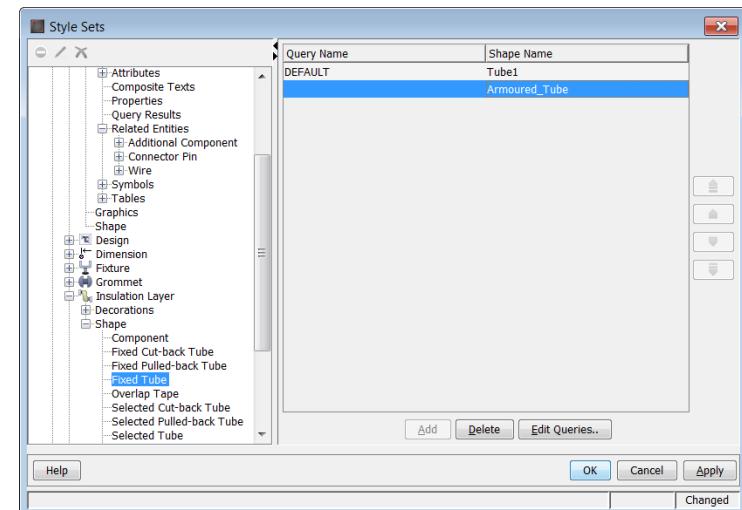
When rendered on the drawing the linestyle configured for each insulation is displayed at the same time allowing the user to see the layering of the insulations.

Assigning new linestyles

It is possible to assign a new linestyle to an insulation by associating a query:

- ⇒ Click the **Add** button and select a pre-defined query or
- ⇒ Select the **Edit Queries** button and create a new query
- ⇒ Select the **Shape Name** to be used for the linestyle

An example of this would be to set a query so that a linestyle would change if an insulation material was pvc, or a certain specified color.



Note

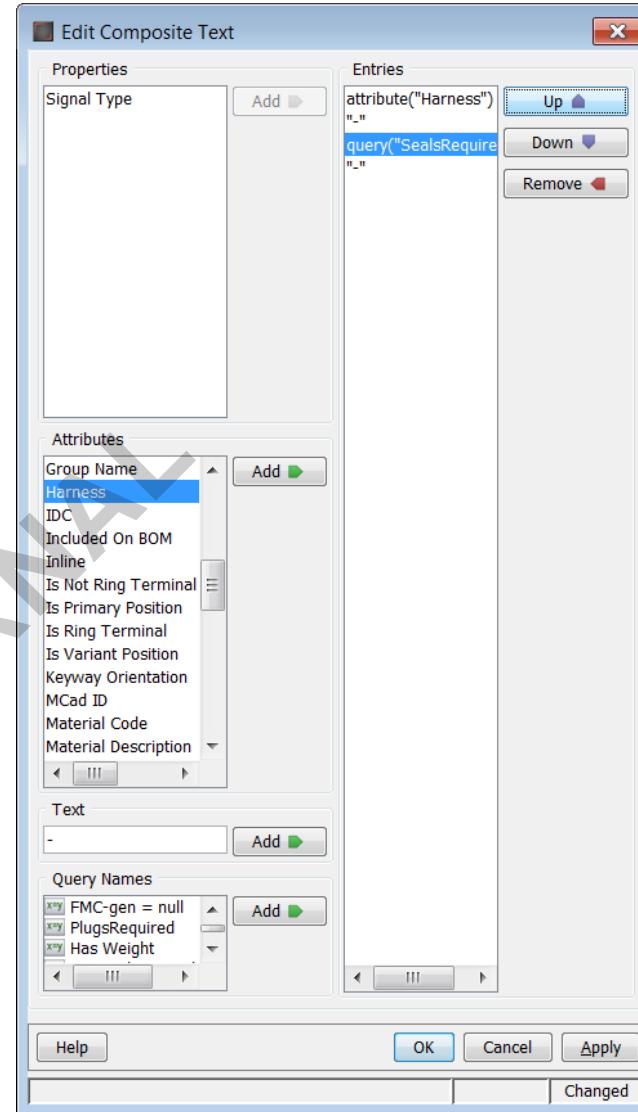
Default linestyles cannot be deleted

Composite Text

The composite text settings held under the style sets (Decorations/Attributes) enables users to insert additional references to diagram objects:

The composite text feature enable users to select a combination of attributes, properties, simple text and query Names in order to create a naming composition:

An example of composite text has been created below:



This composite text has been created for the insulation layer inserted in **Exercise 5**. When the style sets are applied again, the insulation layer will now have the following text associated to it:

-H-1-true-test

H1 is the Harness attribute

True indicates that the query setting for 'overlap tape' has been met

Test is the simple string text that has been added.

INTERNAL

Multi Decorations

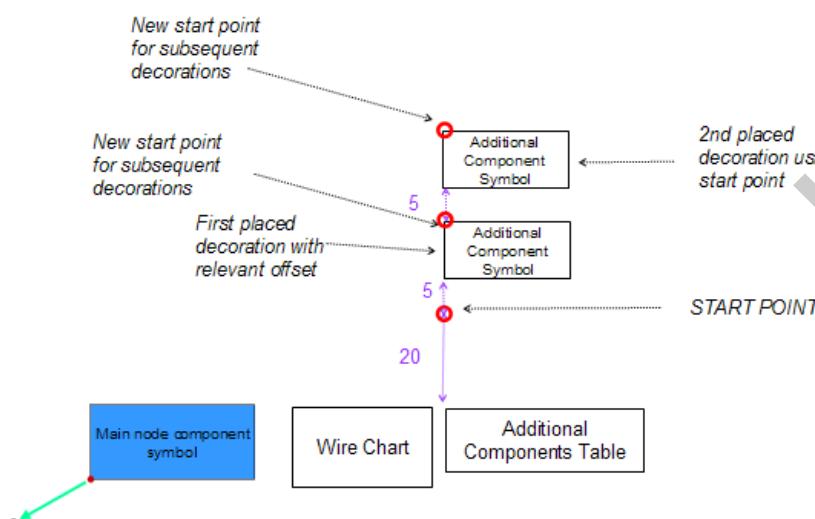
Sometimes components such as connectors will have additional sub components assigned to them in the library. Users may wish to make these additional components visible in some way on the diagram.

Locating and styling additional decorations is done in 2 stages:

- 1: Locate the start point for all additional decorations
- 2: Define the style and location for subsequent decorations

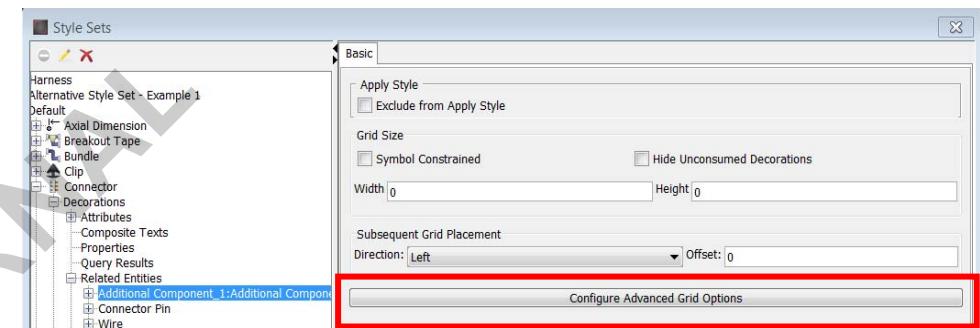
Start Point

This is the point that all additional decorations will be located from. The start point will change once each additional component is added as the nominal origin for the very first additional component that is added will be the owner but for subsequent additional components the start point will move to being the last placed additional component.



To set a multi decoration setting:

- ⇒ Open the Style set / select the object in question/Decorations/Related Entities/additional component
- ⇒ Highlight additional components
- ⇒ Specify the styling requirements
- ⇒ Configure the advanced grid options where this is required to define how any subsequent components will be displayed



Note

It is also possible to style multiple main node components so that a number of symbols may represent the main node component, but each symbol may be represent a different view of the symbol i.e end view, side view. This is described in more detail in the form board chapter.

Connectors, Backshells and keyway styling

This section only needs to be completed by students who will be using backshells and will require the ability to display keyway and backshell orientation information on the diagram

When a connector with a backshell covering is instantiated on a diagram, often in the aerospace environment it is required that orientation information for the backshell and keyway is assigned to the component to identify how they are rotated and locked into position.

The styling tool enables users to define the representation of this information using symbols and conditional styling that indicate the orientation of the backshell and keyway.

Through a combination of specific style settings and queries users can ensure that the resulting diagram displays the correct representation of the backshells and keyway codes.

Before users can begin to define the style sets for backshells, the base, keyway orientation and Backshell orientation symbols must be pre-defined in the Symbol library.

Definition of Queries

The second stage for creating a style set for backshells is to define several queries that question the existence of the objects you wish to style. For example we first want to ascertain whether a connector has a backshell assigned to it. If the Backshell does exist then this would determine that a backshell base symbol should be placed on the diagram.

Once the base symbol is placed, the keyway orientation symbol and backshell orientation symbol will be placed relative to the base symbol, again we will need queries to help identify that these orientation attributes do exist and then we can define in the style set how their symbols should be placed.

The Queries can be set up from within the style set or Capital Project system area under the Query Expressions folder.

Query Expressions use a combination of pre-defined expressions to create a Query that tells the system to investigate a certain aspect of the design and upon finding the results to perform a certain action i.e. look to see if a bundle run contains an insulation with blue foam tubing, if this is true then change the bundle linestyle to show a blue foam linestyle.

Styling the clocking symbols

Once the queries have been set for the clocking symbols we can define the style sets to ensure that the placement of the clocking symbols is in accordance with the backshell and keyway orientation attributes assigned to the connector. By including the queries in the style set we can ensure that the orientation attributes are adhered to and the symbols will rotate in accordance with the various attributes.

To define the style sets for clocking purposes there are 3 steps we need to ensure are covered;

- 1) Define the base symbol styling such as its nominal origin, its orientation in relation to the main node symbol and the query that identifies if the connector has a backshell associated to it
- 2) Define the Backshell orientation symbol, its symbol justification, orientation angle using the BackshellOrientation query and its nominal origin in relation to the base symbol
- 3) Define the Keyway orientation symbol, its symbol justification orientation angle using the KeywayOrientation query and its nominal origin in relation to the base symbol

The Helpfile has a very detailed step by step guide on how to perform this style set definition – this will also be covered in the exercise worksheet.

Your instructor will now deliver the styling avi which gives a brief overview of some more advanced styling features available in the tool..

Table Styling

Along with the standard styling and formatting available for all decorations, tables have some additional styling capabilities.

Table Splitting

Depending on the object selected for styling (connector or bundle), it is possible to define columns to be displayed, column titles, title justification, column sort priority as well as the maximum number of rows and columns that a specific table should use therefore splitting the table into multiple tables.

Tables may be split interactively on the diagram:

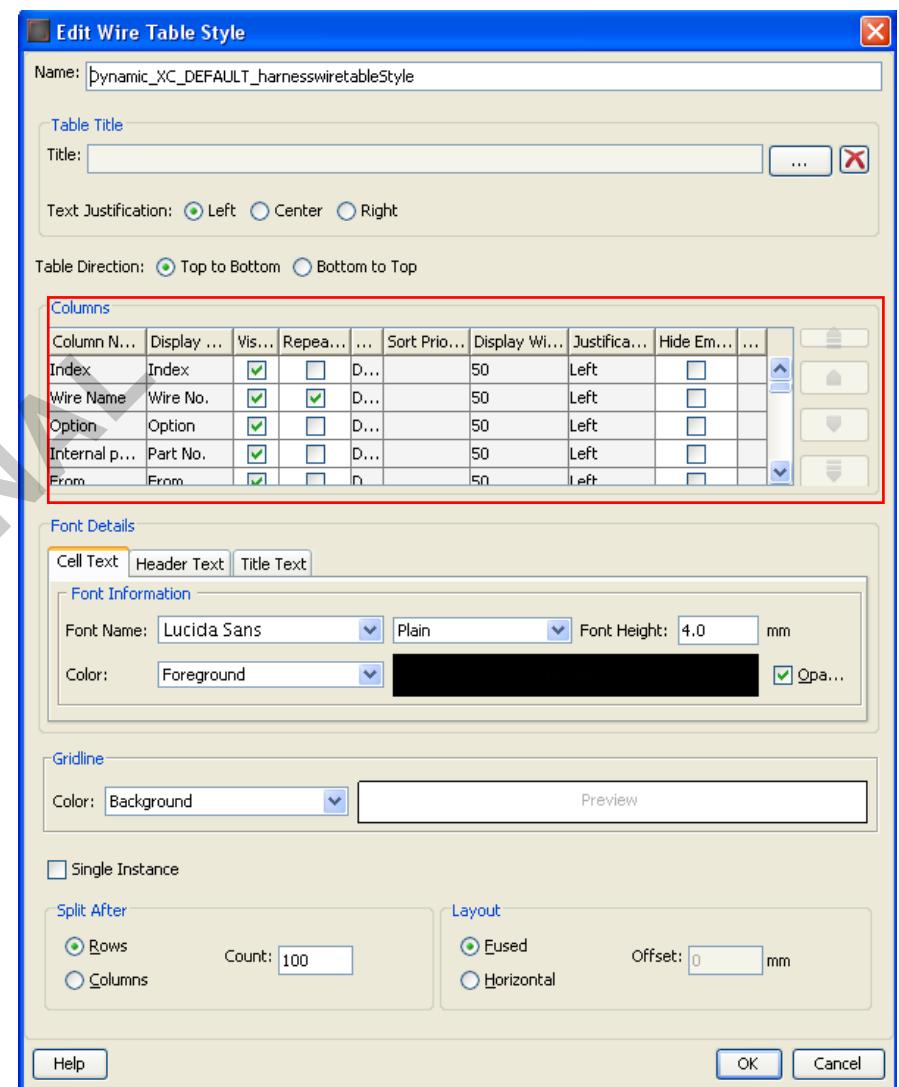
- ⇒ Within the harness diagram Right click on the table
- ⇒ Select Override styling
- ⇒ Define the number of required rows, columns and the layout information
- ⇒ Click OK and the selected table will be split as required

Setting the **horizontal offset** and **vertical offset** the user can determine where the subsequent tables are located with respect to the first table.

Note

When the split feature is implemented on a table this information is retained when applying the style set.

Gridlines can be colored separately to the rest of the table content, thus allowing the lines to blend in with the diagram background or making them stand out to the rest of the diagram.



Each column style can be defined independently from the others.

Wire Table Style Settings

The placement of wire tables on a design enables users to identify wire part numbers and locations within that design. The various wire tables that are available for use in Harness XC are:

- Assembly Table
- Harness Composite BOM table
- Harness Composite Derivative Information Table
- Functional Module information
- Multicore Table
- Plug-In Table
- Overbraid Table
- Harness Revision Table
- Harness Wire Table

These tables may be styled under the style sets menu:

- ⇒ **Edit / Style / Style Sets / Design / Tables**
- ⇒ Highlight the table you wish to set a style for
- ⇒ Click **New** or **Edit** to edit a pre-existing table
- ⇒ If you click **New** then you must specify a new name for the table and define the required columns and style for the table
- ⇒ If you click **Edit** for a table you may edit the style of the table

A number of style settings may be defined:

- ⇒ You may define the font style and color for the table column headings and cells
- ⇒ You may wish to define table splitting and layout for the wire tables to suit your own diagram standards
- ⇒

Note

To instantiate a wire table on a drawing users must first place a border from the border symbol library if the nominal origin for the table is set to symbol / border

Once a border has been placed on the XC drawing the wire table may be instantiated on that drawing by selecting:

- ⇒ **Actions / Update border**

Table Plug-Ins

Users are able to upload their own tables as plug-ins.

Plug-ins are loaded by the system and may be re-loaded via File / plugins

Table plug-ins specifically may be styled in the style set under the standard style settings for tables. Multicore tables will be loaded and styled in this format.

- ⇒ Navigate to the table style set for the design
- ⇒ Select **Add / Select Plug-in table** from the drop down list
- ⇒ Click **Yes** to the styling message that appears
- ⇒ Name the plug-in table
- ⇒ Browse for the particular plug-in table you wish to style
- ⇒ Define the style settings for the plug-in table
- ⇒ Click **OK**

There is an exercise worksheet available for users to try example styling in their own time.

Chapter 14

Form Board Basics

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INTERNAL

Introduction to Form Board

Introduction

A major requirement for many harness manufacturers is the ability to produce a scaled form board drawing directly from the Harness drawing. The Harness XC tool provides this ability with the introduction of scaled drawing functionality and the ability to introduce bends, curves and harness rotation within the harness drawing.

The option to create full scale diagrams is a licensed operation, this option will only be available if you have the correct license.

Multiple Drawings within the same design

In addition to creating a standard 2D harness diagram users who wish to create full scale diagrams may do so by creating a new diagram based on the information held in the original harness diagram.

- ⇒ Right Click the design name in the project browser
- ⇒ Select **New Diagram**
- ⇒ Define the new diagram name, style set and reference diagram
- ⇒ Define the diagram scaling
- ⇒ Define the diagram physical layout and grid spacing

The following options are available for scaling definition:

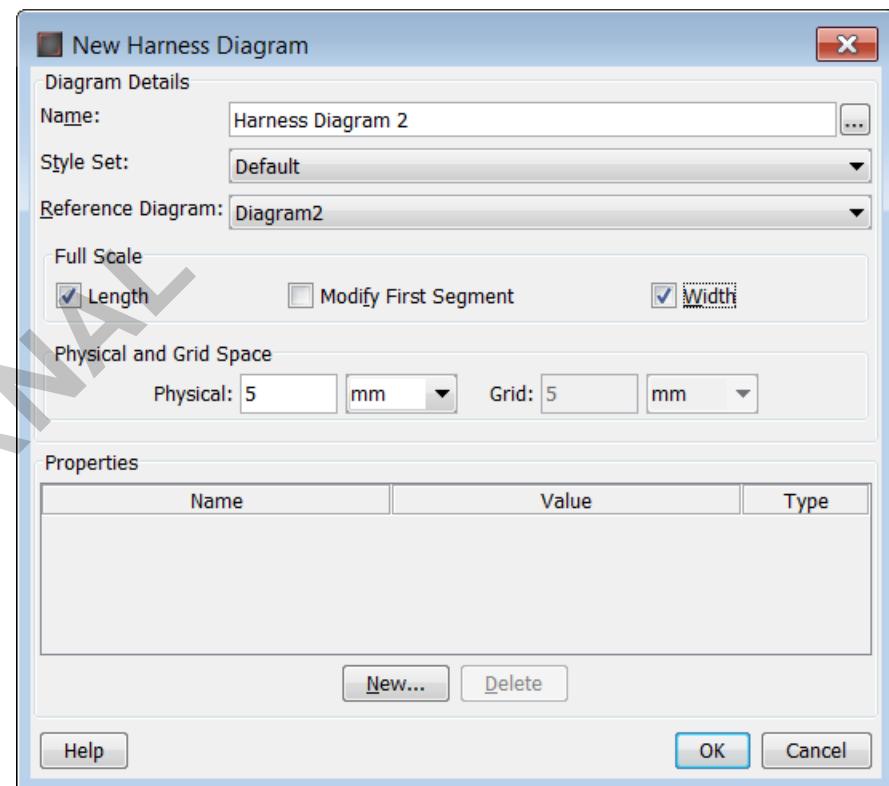
Length: The physical scale for the diagram is locked to 100%, thus bundles are displayed at the same length as they will be printed out.

Width: Available only if the Length option is selected and displays bundle widths at full scale. Note that this is dependent on running the engineering calculations.

Modify First Segment: Again this is only available if the length option is selected. This option ensures that if length changes are made they are applied to bundles closest to the datum node. By default length changes are applied to bundles furthers away from the datum node.

Physical Spacing: Defines the actual distance between 2 points on the drawing grid

Grid Spacing: Defines the distance between 2 points on the drawing grid when the drawing is printed



Once defined, the new diagram will be placed within the design folder. The new diagram has a reference link to the original diagram, thus any changes made to original diagram will be replicated in the new diagram. However, any changes made in the new diagram **will not** be replicated in the original diagram.

Bundle Manipulation

Inserting an arc into a bundle

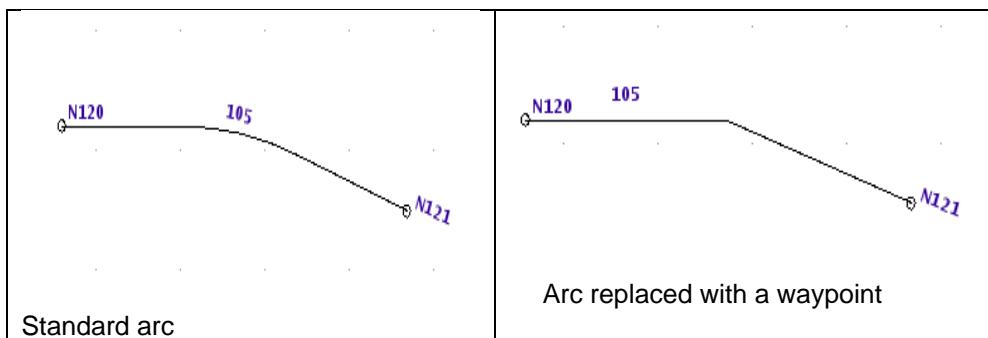
In the formboard domain it is very useful to be able to insert a curve into a bundle, taking into account the flexibility available on the bundle due to the bundle content or thickness.

To insert an arc:

- ⇒ Right Click the bundle on which you wish to insert the arc
- ⇒ Select **Arc / Add arc**
- ⇒ Specify the point on the bundle from which you wish to start the arc
- ⇒ Specify the exit point for the arc
- ⇒ Specify the radius for the arc (the higher the value for the radius the smoother the bend will be)
- ⇒ Specify the Travel Angle for the arc – this is the angle between the entry point for the arc and the exit point. By default the travel angle will be placed anti-clockwise to the bundle, in order for the arc to be placed in a clockwise direction the travel angle must be defined with a negative value
- ⇒ Check the box to rotate attached bundles if this is required
- ⇒ Click **OK**

The arc may be edited at any time by right clicking the bundle and selecting **Arc / Edit Arc**.

The arc may be converted to a standard waypoint (**Right Click bundle / arc/convert to waypoint**), or alternatively it may be converted to a straight line.



Bend Radius

Users may manually edit the bundle to define the minimum bend radius. This tells the system that the particular bundle in question should not have an arc introduced that has a bend radius less than the specified bend radius. When users try to insert an arc with a radius less than the specified value that value will be shown in red to indicate there is a problem.

To define a minimum bend radius:

- ⇒ Right Click the bundle / Edit bundle
- ⇒ Define the Min Bend Radius in the form that pops up

If users bridge information in from another data source i.e. MCAD this information can be brought through in the MCAD file.

Bundle Width representation

Users may display the full scale bundle width on a diagram. This feature helps users decide where best on the harness to place a bend or curve. The width displayed on the diagram will be the total outer diameter of the bundle including any applied insulations. If an insulation run or insulation layer is set against a bundle the outer layer or run linestyle will be rendered in full scale width.

This information may also be bridged in via an MCAD file.

Note

Accurate bundle diameter and bundle length calculations are only available once the harness processing has completed.

Rotating a harness section

Although this function is not restricted to the creation of a formboard drawing, it is more likely to be required within a formboard, scaled drawing.

The rotation of a harness section enables users to change the positioning of a whole section of a harness so that it can better fit a form board. Users also have the choice of whether other bundles associated to the main section that is to be rotated, will also be rotated or not.

To rotate a harness section:

- ⇒ Select **Layout / Rotate harness section** or right click a bundle and select **rotate section of harness**
- ⇒ Select a point on the bundle around which you wish to insert a rotation (this would typically be a node or waypoint)
- ⇒ Select a second point on the bundle close to the waypoint you wish to rotate around – this will denote which section of the harness to be rotated i.e. to the left of point 1 or to the right of point1
- ⇒ Begin the rotation

Points to consider:

Users may do a full rotation of the selected section or a rigid rotation whereby only the main bundle is rotated and any attached component or bundles will retain their original position.

To switch between rigid or full rotation, use the **T** key

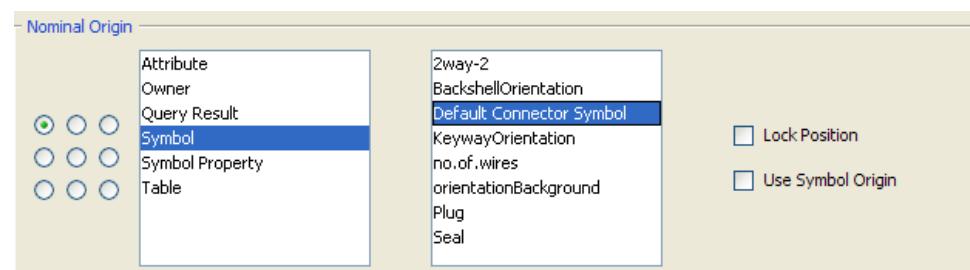
When the rotation function is initiated it is assumed that the angle of rotation will be measured from the current angle of the section selected. i.e. if there is a selected section at 45° and you wish to rotate it 80° you will need to rotate until the tool indicates 35°. If users were to press the CTRL key the angles are displayed with reference to the global horizontal axis, in which case rotating the angle to 80° is done by rotating until users see the angle increase to 80°.

Multiple Main node component Symbols

When creating a full scale diagram it may be necessary to display connector end and side views on the diagram to help identify where the wire ends are located and in which connectors. To help with the positioning of the connectors on the diagram users can implement the styling tool (restricted by a styling license) to help place the connector end views in one location and the side views and cavity tables in another location, for example in a separate constraint box placed off to the side of the diagram.

For a basic definition of multiple components display against the main node users can manipulate the styling as follows:

- ⇒ Open the desired style set (edit/styles/style sets)
- ⇒ Navigate to connectors /decorations/symbols
- ⇒ Click **Add** and select a library set and symbol from the libraries in the chosen library set
- ⇒ Check the Main symbol Decoration box
- ⇒ Click **OK**
- ⇒ Highlight the symbol defined above and define the justification and orientation points
- ⇒ Repeat the above steps to add a second symbol, remembering to check the main node symbol box
- ⇒ Locate the second symbol in relation to the 1st symbol : in the **Nominal origin** section select 'symbol' and in the table to the right hand side select the original symbol name



- ⇒ Click **Apply** and **OK**

For a more advanced explanation of styling users should refer to the styling chapter for further details on styling the diagram.

Exercise

Instructors will now give a short demonstration of the basic capabilities of the harness XC full scale drawing tool.

INTERNAL

Appendix 1

Introduction

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Getting Started

Diagram User Interface

The diagram user interface is comprised of the following:

Project Browser – used to display the current contents of the project and provide access to open/close designs and diagrams.

Menu Bar – used to access all commands. For example: Menu bar item ‘View’ contains commands related to viewing the contents in the Graphics window.

Tool Bar – used as quick means to execute commonly used commands.

Diagram Browser – used to view contents of the current diagram.

Graphics Window – used to create and edit the electrical diagrams. Provides graphical entry of connectivity data into the database.

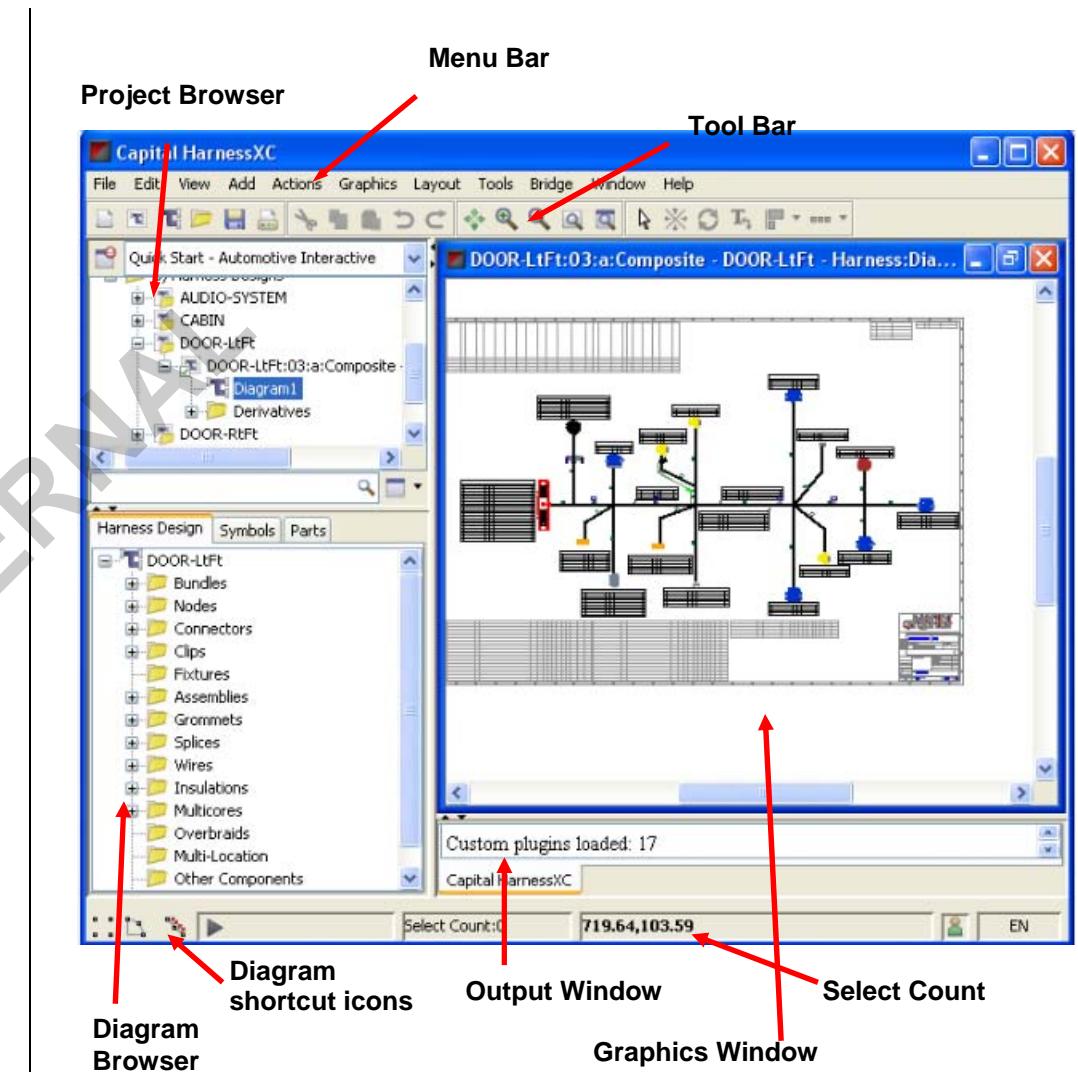
Message Area – used to display messages about the current executed command.

Output Window – used to provide a transcript of the most recent action/command.

Select Count – used to display the number of design objects currently selected.

Cursor Location – used to display the x-y location of the cursor in the graphics window.

Shortcut icons – used to quickly enable grid snapping, polar snapping, proportional stretching



Popup Menu

In addition to the menu bar and tool bar, the tool provides popup menus containing commonly used commands for the window in which the popup menu was called. A right mouse click in the desired window accesses the popup menu for that window.

Hot Keys

It is possible to access commands by using Hot Keys (like in MS Office products).

For example, **Alt + A** will bring the **Add** command up, pressing **N** will allow you to use the **Add Net** command. These are listed in the help files and are shown at the right hand side of the menus.

General Window Control

The Capital tool provides capabilities to control which windows you want visible and to some extent how those windows are arranged in your Capital session.

Three of the windows may be toggled to be visible or hidden. These are the **Browse** Window, the **Output** Window, and the **Status Bar**.

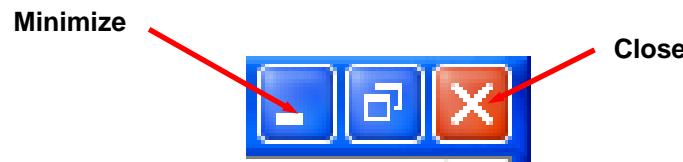
The Browse Window, Output Window, and the Graphics Window have a resize capability by placing the cursor directly over an edge of the window and when the cursor changes shape, left click the mouse and hold until the desired size is reached.

The Output Window and Browse Window also have a resize control built into their window. These controls, by left clicking the mouse on the arrow of the desired direction will automatically resize that window.

Window Minimize/Maximize/Close Controls

The Graphics Window, the window in which you create the diagrams, has controls that affect the size and layout within the session.

The graphics window can be resized, minimized, maximized/restored, and closed.



Window Layout Commands

Cascade: Cascades all open diagrams across the diagram browser

Tile Horizontal: Places each open diagram horizontally on the diagram browser

Tile Vertical: Places each open diagram vertically across the diagram browser

Tabbed Mode: Displays a tab button for each open diagram along the bottom of the diagram browser and users click on each one to view the contents

Modify Grid

A grid facility is provided to assist with graphical construction.

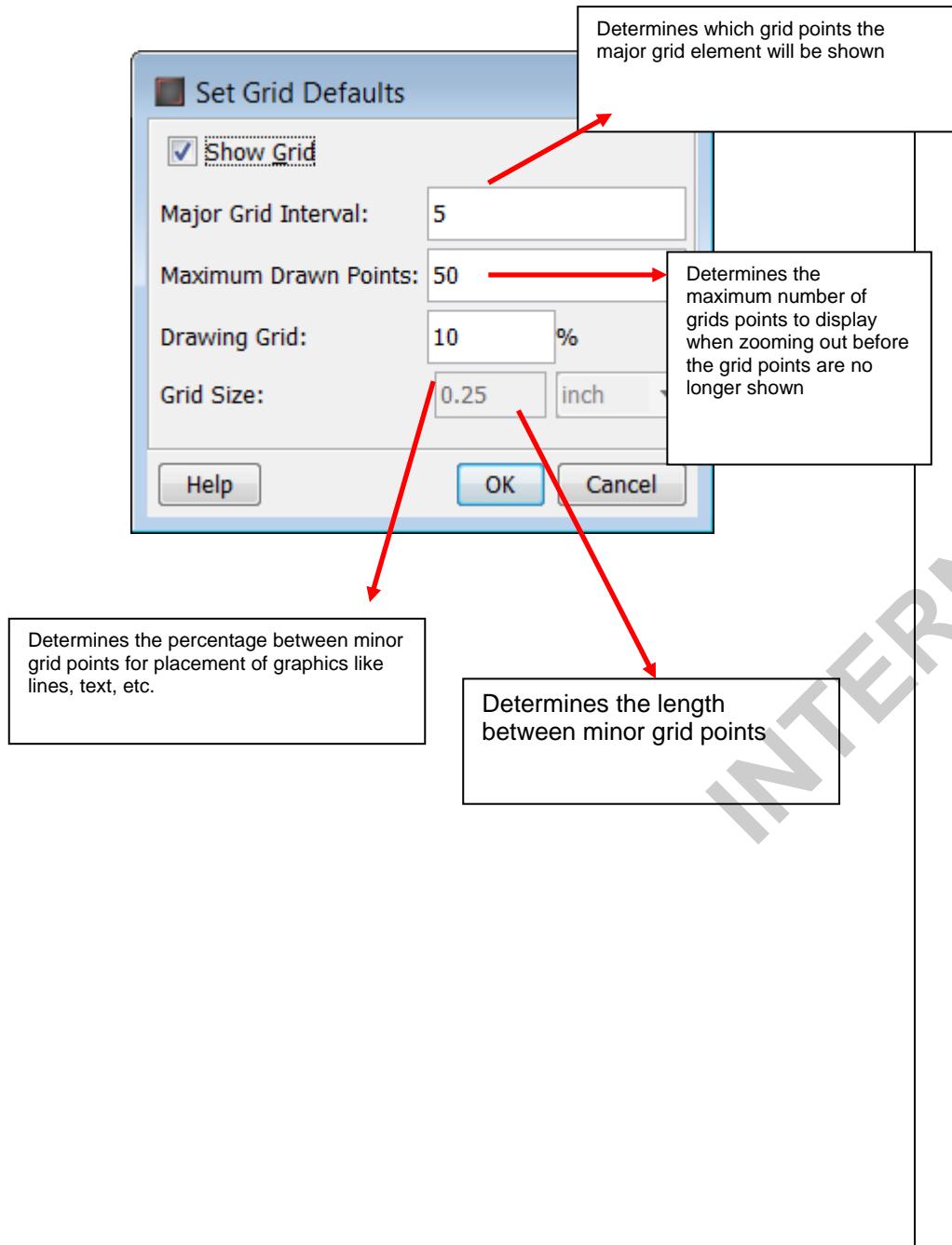
The grid consists of a matrix of points arranged at specified intervals on the diagram. This allows for easy alignment of objects on the diagram by causing the objects to 'snap' to the nearest grid point when placed or moved.



The Grid facility is accessed by the menu bar Layout / Grid / Set Grid defaults. Executing this command displays the Modify Grid dialog box. This dialog box is used to specify the grid display within the symbol diagram.

Note

Pins can only be placed on visible grid points.



Mouse Interactions

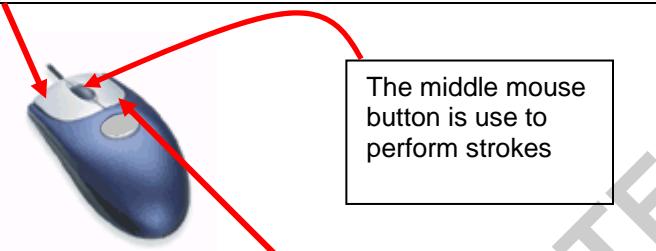
Each of the following standard mouse interactions unselects any selection and performs:

Dragging (Click and Hold) is used to select a range of objects in a graphical application.

Clicking (Click and Release) selects or activates the object beneath the pointer.

Double-clicking is used to select larger contiguous text strings (example: double-click to select a word in a text field vs. Dragging to select a single character) or is used to select and open an object.

Triple-clicking (Text only) is used to select entire text field.



Right clicking is used to display popup menus

Note

A **Control-left click** within toggles its selection without affecting the selection of any other objects. This is used for additive selection in diagrams. For text lists, using the Control key with the left mouse provides additive selection of non-contiguous objects.

The **mouse wheel** is used in graphic applications to zoom in and out.

Strokes

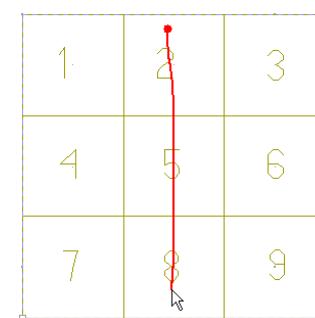
Strokes are predefined cursor movements used to execute a command. They are performed when pressing and holding the middle mouse button.

Each stroke maps to a pattern on an imaginary 3x3 matrix. The path of the cursor through this matrix identifies them.

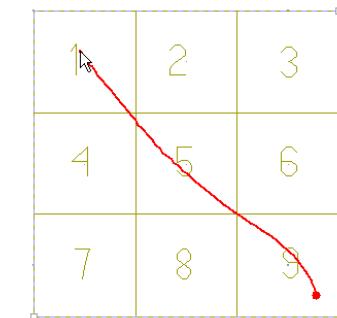
For example, the following stroke (a counter-clockwise "C" pattern) is named the 3214789 because of the path through the matrix.



Examples: (The matrix was drawn using non-electrical objects and is shown here for reference only.)

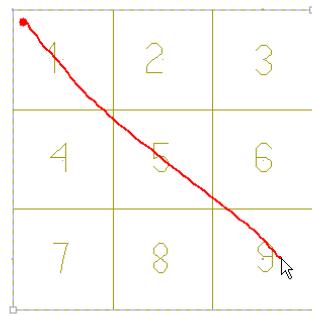


Add Net – stroke 258

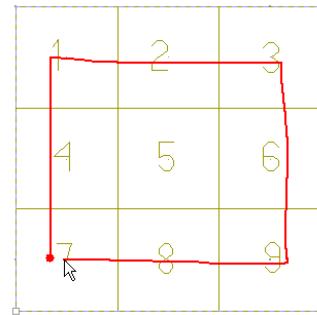


View All – stroke 951

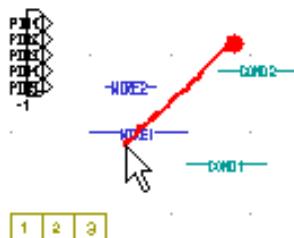
There are several common functions with predefined stroke paths.



View Area – stroke 159



Delete – stroke 741236987



View In –stroke 357 View Out –stroke 753

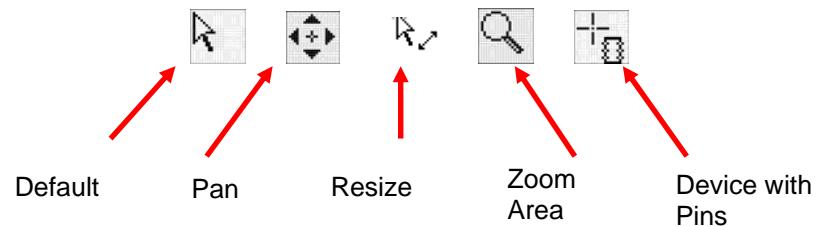
Other stroke patterns shown in table below.

Command	Stroke pattern
Undo	1478963
Redo	9632147
Autoroute Selected	96321

Cursor

The cursor is dynamically updated to indicate the current action:

Examples:



Each cursor is shown as used later in the training material.

Appendix 2

Object Naming

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Object Type Information

Objects within the Capital application can have properties associated to them and naming conventions for each specific project.

The following are objects and can have properties and naming conventions specified:

Assembly: is several components grouped together to create one single entity (typically one part number or a complete harness)

Axial Dimension: Typically used for formboard fixtures, depict the dimension between specified points

Backshell: is a seal that covers the back of a connector

Backshell Termination: indicates the termination for a backshell

Breakout Tape – taping at a junction

Bundle: a path for electrical conductors

Clip: used to fix a group of wires or bundles of wires to a static point

Connector: is the component that transfers the signal to the Device e.g. ring terminal, connector

Continuous Insulation: Is the layered insulation layer on a harness bundle

Device: is the electrical component that receives and acts on the signal e.g. headlamp, fuse box, battery

Dimension: typically an external dimension to indicate the distance between specified nodes within a restricted area

Fixtures: Used with formboard functionality to assign fixture points to the harness diagram

General: is used for all objects unless specified

Grommet: a protective covering to prevent chaffing – fixed to a static point

Ground: is used for all ground objects

Harness: a combination of wires and components

Harness Diagram: Graphical representation of the harness

Insulation Run: Is a run of a particular type of insulation on a harness bundle

Interconnect: a representative path for interconnect wires, multicores and overbraids

Integrator Design, Logic Design and Harness Design: enable users to create names and properties for Logic, Integrator and Harness designs – these can be assigned to designs in Capital Project when the designs are created.

Integrator Plane, Logic Diagram and Harness Diagram: enable users to create names and properties for logic diagrams and integrator planes – these can only be assigned in Capital Logic, Capital Integrator and Capital Harness XC respectively when the initial diagram/plane is being created

Multi Location Component – any component that may span several nodes, cavities or wires i.e. busbar

Multicore: is a group of wires usually covered with a sheath (or jacket)

Net: is the electrical signal route from point to point (could connect several pins)

Node – point at which a component may be placed

Pin: (or cavity) is the connection point on a connector that the signal is taken to

Shield Indicator: is the representation of the type of multicore

Shield termination: is the representation of the logical extension of the shield

Splice: is a method that electrically connects several wires together

Spot Tape: refers to the application of a one off section of tape (not on a run)

Wire: is the individual route from pin to pin

Project: enables users to create properties that can be assigned to a project in the Capital Project Edit form.

Note

Object type properties can be manually transferred from one project to another using the drag and drop facility

INTERNAL

Object Properties

An object property consists of a name, a value type and a value constraint type, it is a way of further identifying an object e.g. color, temperature resistance, wire size etc

Name: this is the reference by which the property is selected from a drop down list

Value Type: - this can be one of 3 options:

- ⇒ **Integer** – a whole number only
- ⇒ **Float** – a decimal or whole number
- ⇒ **String** – Alphanumeric characters

Value Constraint Type: This can be one of three options:

- ⇒ **Unconstrained** – A value is applied once the property is assigned to an object
- ⇒ **List** – a list of specific values are applied to a property. When that property is assigned to an object that specific list will become available for use
- ⇒ **Range** – A numerical range can be created with an upper and lower limit, the range is applied to an object property and if a user enters a value outside of that range a warning is given

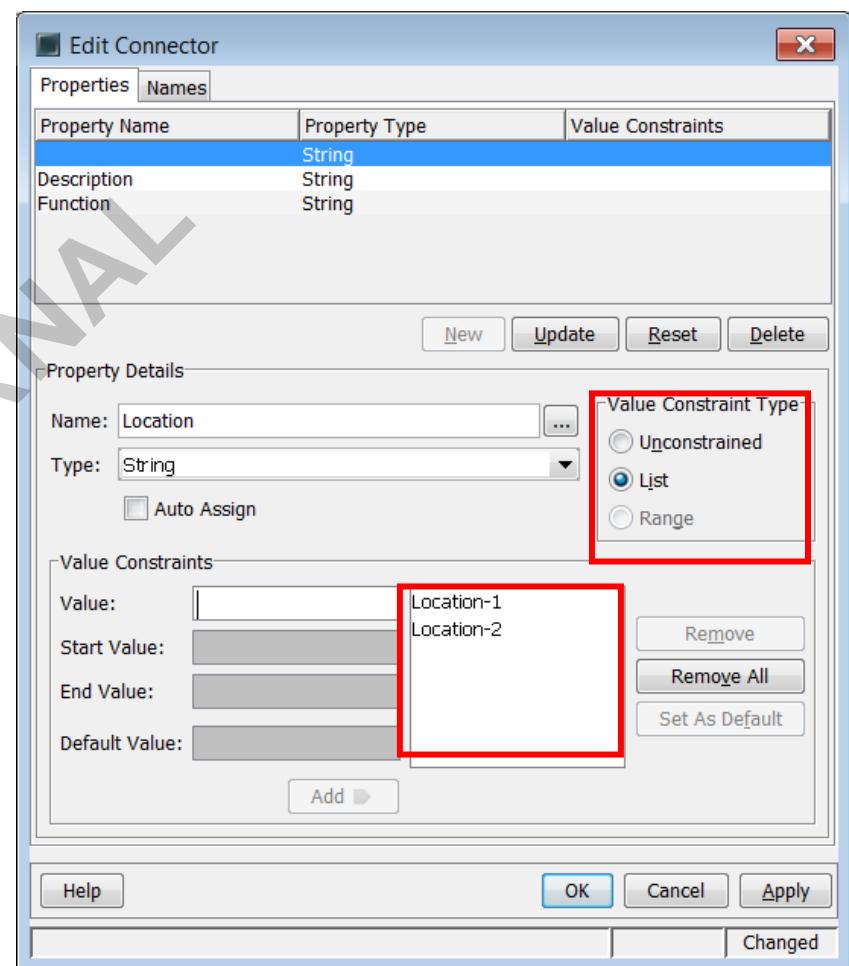
Note

The system will prevent you from entering a value that is not in the list or range.

Creating a List Property

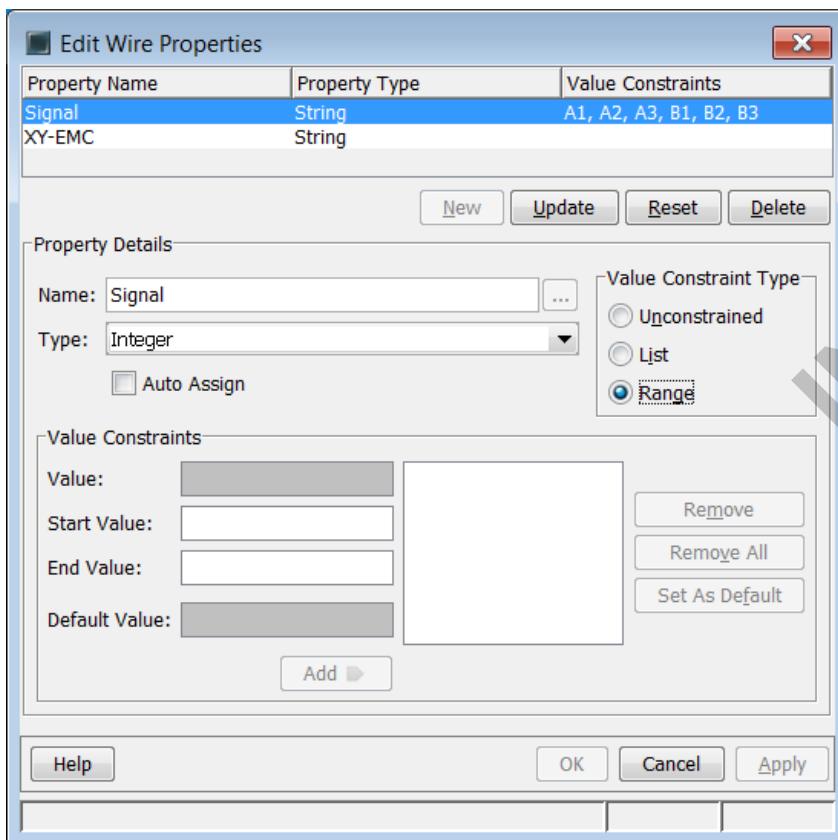
- ⇒ Double click the object icon you wish to add a property to
- ⇒ Click the **New** button in the Edit properties form
- ⇒ Enter a Name for the property
- ⇒ Select the property type – remember that if you choose integer or float only numerical values can be entered for the list
- ⇒ Choose the **List** radio button

- ⇒ Enter the value you wish to have available in the property values list and click **Add**- repeat this process to add more values
- ⇒ Click **Update** and **OK** to confirm the details and close the form



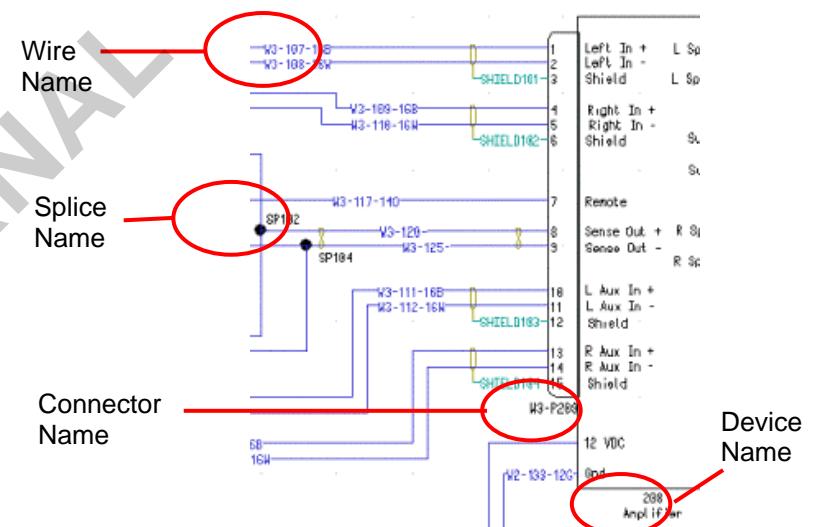
Creating a Range Property

- ⇒ Double click the object icon you wish to add a property to
- ⇒ Click the **New** button in the Edit properties form
- ⇒ Enter a Name for the property
- ⇒ Select the property type you wish to use – remember only numerical values may be entered in a range so float or integer property type must be selected
- ⇒ Select the **Range** radio button
- ⇒ Enter a start and end value for the range and click **Add** to add the range to the available list
- ⇒ Click **Update** and **OK** to confirm the details and close the form



Assigning Object Properties

Default property names and values can be associated to the various objects used in the project designs. These properties and values are available whenever the properties for that type of object are edited in a design.



In the diagrams the user can assign the property values created in Capital Project to the required object types in a design.

- ⇒ Place the design objects on the design
- ⇒ Double click the object type or right mouse click and select properties
- ⇒ In the properties section of the form click on the **New** button

A drop down list will appear containing all properties created for that object type.

- ⇒ Select the property you wish to assign to the object

Click OK and the property name will appear in the properties section of the form. The applied property value will appear on the design if the user has checked the visible box in the properties line

If the property **value constraint type** is unconstrained then the user may type in any value for the property as long as that value matches the value type (float, integer or string) assigned to the property when it was created in Capital Project.

If the constraint type is **List** the user will chose the value from a drop down list.

Note

The user must choose a value from the available list and cannot enter any other value.

If the value constraint type of the property is **Range** then the user may only enter a value within that specified range.

Note

If the value exceeds the specified range for that property the value field will be colored Red and the value may not be applied:

Assigning properties to projects

Properties created for projects may be assigned to a Project from within the Edit project names form

- ⇒ Right click the project name and Edit
- ⇒ Click the properties button to the right of the Name field
- ⇒ Select New and select the property to be assigned from the drop down box
- ⇒ Update and Ok the form

Assigning properties and names to Designs

Assigning Names and properties to Logic, Integrator and Harness designs can be done from within Capital Project:

- ⇒ Right click the design name
- ⇒ Edit, select the properties button to the right of the Name field
- ⇒ Select New and then select property to be assigned from the drop down box

Note

To assign Logic Diagram names and properties the user must be in Capital Logic and so on.

Default Naming

The Capital designs consist of electrical objects. All electrical objects within the Capital applications are required to have a name. In general, these names serve as the unique identifier within the project for that object. The naming of these objects is accomplished automatically based on the naming composition specified in Capital Project. The naming compositions for each object type are defined on a project-by-project basis.

This naming composition specification is user definable through one of two methods:

- ⇒ Simple Prefix (Default)
- ⇒ Composite Name

Note

For each object type listed above, you can choose either the default naming or composite naming convention. The tool will not allow a combination of the 2 for a single object type.

Users can overwrite the automatically generated object name at any time while editing the design.

Simple Prefix

The **Simple Prefix (Default)** specification generates names using a prefix and a counter. The prefix is specified in Capital Project and applied in the diagram.

For example:

Capital Project specifies

Pin prefix = **PIN**

Diagram places	First pin:	NAME = PIN1
	Second pin:	NAME = PIN2
	...	

Composite Name

In addition to the default object naming (Simple Prefix), it is possible to create name compositions containing any or all of the following:

- ⇒ Property Names – will display the corresponding value
- ⇒ Literal Text Strings
- ⇒ An incremental Counter, tracking the number of such Objects in the Project

These are ordered and combined (concatenated) to provide the name applied in Capital Logic.

For example Capital Project specifies:

Plug Name = **harness** + “-“ + **class letter** + <counter>

Where,

Harness and class letter are properties, which are applied to the plug connector within Capital Logic

And

“-“ is a literal string,

And

<counter> is an internally managed and incremented sequential numeric counter.

Capital Logic places First plug **NAME = -1**
 Second plug **NAME = -2**

User adds the property

class letter = P to -1 and -2 plug connectors

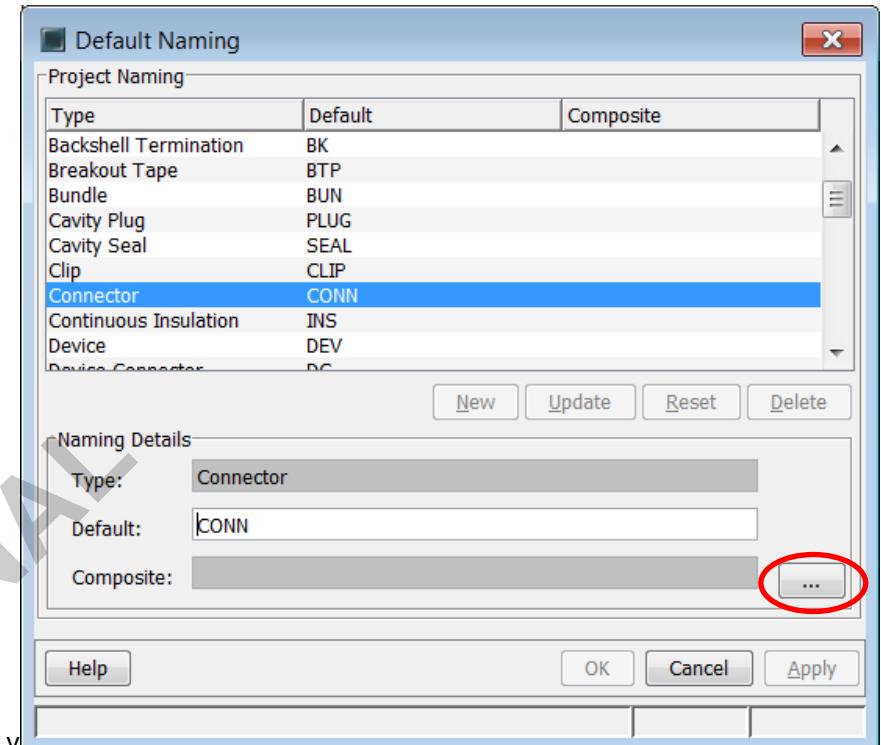
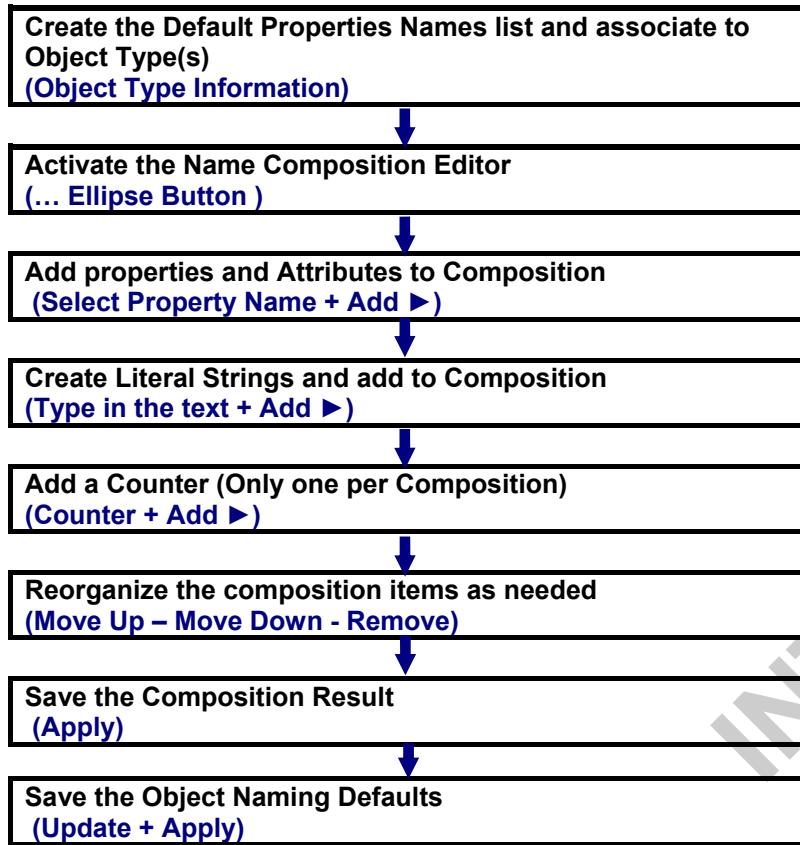
The NAMES automatically change **NAME = -P1**
 NAME = -P2

User adds the property

Harness = W1 to -P1 and -P2 plug connectors

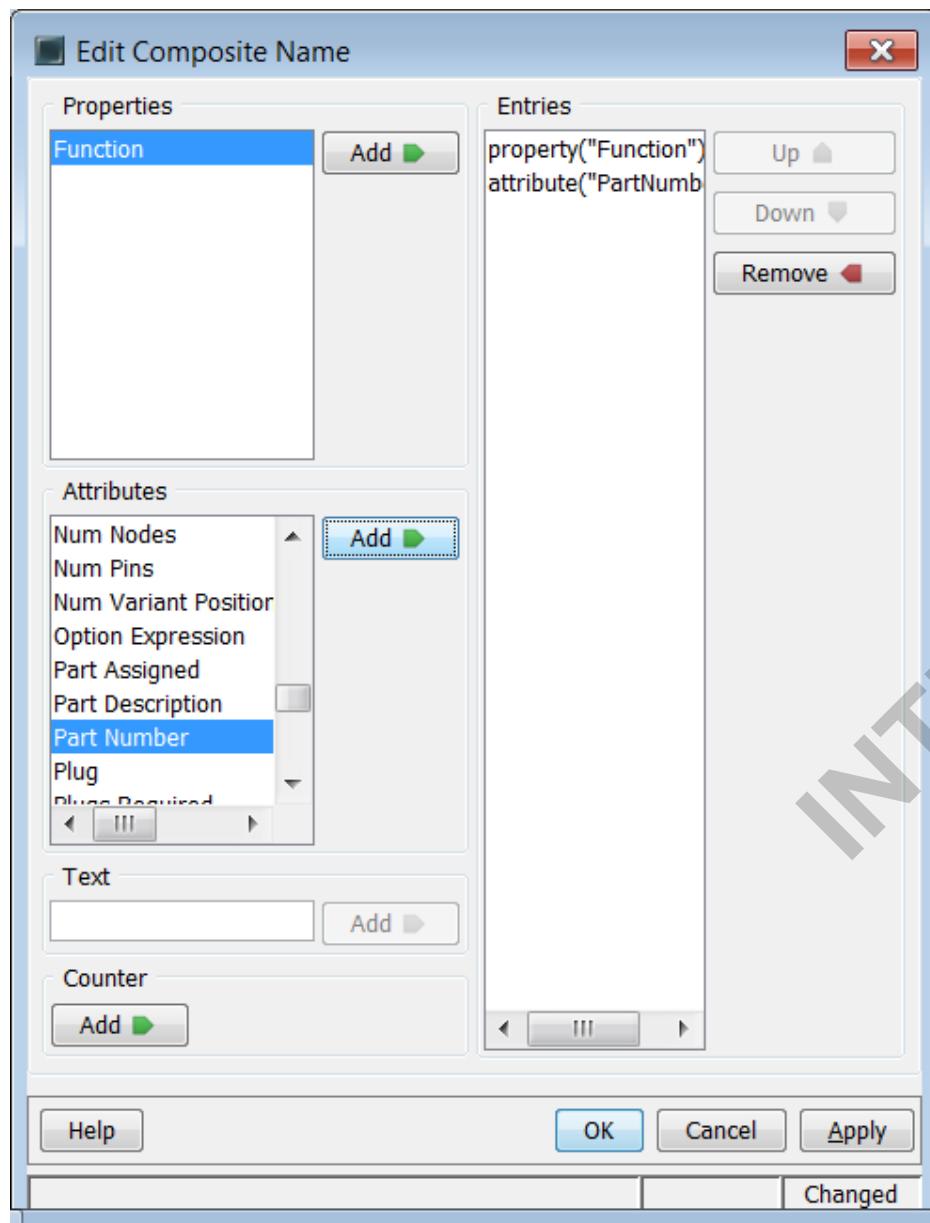
The NAMES automatically change **NAME = W1-P1**
 NAME = W1-P2

Create a Composite Name



The Edit Composite Form enables users to create a composite name for a specified component type using pre-defined properties, attributes, simple text and counters.

If the established Naming Composition requires a specific property and the property does **not** exist on the object in the diagram, the name of the object will be created without that property. However, as soon as that specific property is added to the object, the name of that object is updated to reflect that property according to the established Naming Composition.



Object Names

Another alternate method of naming objects within a design is by defining a set of predefined names and short descriptions. The list of names is available from the Object Type Information Editor, Name tab or by expanding Object Type information, expanding the object and double clicking or Right click and Edit on Names.

Object names can also be dragged from the System definition into Object naming in the project definition. By default, System defined object names are available in the schematics to be assigned to objects. The list of names and short descriptions for a particular object appears when clicking the ellipses next to the name field in the Edit → Properties window.

Note

Object Naming automation assumes the following rules.

1. When an object is placed on a diagram, the default name plus incremental value becomes the objects name.
2. If an object name composition is defined, the default will never be applied for that object type. Instead, the object will inherit the name composition.
3. If the user edits the Name value in the properties dialog, the value will not follow either default naming or name composition rules.
4. If the user selects a name from the list, this will become the object's name.
5. If the user wishes to view the short description on the diagram the short description field box must also be checked.

Appendix 3

Harness Checks

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INTERNAL

Example Warning of Harness Checks

Check for unlocated wire ends

Customer Harness: Steering Harness Issue: a
Internal Harness: Steering Harness Issue: A
Customer Name: European Motors Model:
Drawing Title: Stat.: Draft

Wire No.	Cust. Wire	Unlocated At End
9	Both	

Warning: Wire ends which have not been assigned a cavity are listed here.

Example: Wire no. 9 has both ends unlocated.

Solution: Use the Insert / Edit Wire form and place unlocated wire end into a cavity.

Check for unterminated wire ends.

Customer Harness: Steering Harness Issue: a
Internal Harness: Steering Harness Issue: A
Customer Name: European Motors Model:
Drawing Title: Stat.: Draft

Wire No.	Cust. Wire	Wire End	Node	Route	Cavity
10	2	C30	6		
11	1	C30	4		
11	2	C10	5		

Warning: Wires which have not had a terminal specified or automatically selected by the system.

Example: Terminals have not been selected for wire 10, node C30 cavity 6.

Solution: Use the Insert / Edit Wire form and check that a terminal is required, and if applicable set the termination method to 'System Calculated'. Make sure the **Housing Definition** of the connector contains a terminal fitting the wire, and the terminal fits the wire specification in the **single wire termination** form.).

Check for components with graphics status = 'NO'.

Customer Harness: Steering Harness Issue: a
Internal Harness: Steering Harness Issue: A
Customer Name: European Motors Model:
Drawing Title: Stat.: Draft

Internal Part Number

AB-45938

GRE-100

Warning: The part numbers of those components which have the graphical status check box 'unchecked' in the component base details form are listed.

Example: Part No.s AB-45938 and GRE-100 have not got the graphical symbols box checked.

Solution: Check the form board classic symbol? box in the Component Maintenance form, if the component should have a Form Board graphical symbol. Those part no.s that should not have a Form Board graphical symbol can be ignored.

Check for branch ends with no components / wires.

Customer Harness: Steering Harness Issue: a
Internal Harness: Steering Harness Issue: A
Customer Name: European Motors Model:
Drawing Title: Stat.: Draft

Node Route Missing

80 NO COMPONENTS OR WIRES

Warning: Branch ends which do not have a component, or branch ends containing a component without wires are listed here.

Example: Above node 80 has no components or wires present.

Solution: Check in **Report on Branch Configuration**, if the branch definition is correct. If this is the case, check in the **Components and Terminations** window if the component definition is correct, and check in the **Cavity Details** that the wires have been routed properly.

If the harness is a Derivative Harness, check to see if the options set against the branch configuration, insulation or components have caused the branch to be used.

Check for new components.

Customer Harness: Steering Harness Issue: a
Internal Harness: Steering Harness Issue: A
Customer Name: European Motors Model:
Drawing Title: Stat.: Draft

Internal Part No.

C-64442
GRE-100

Warning: Components which have been specified as **New** in the **Component Base Details** are listed here.

Example: Part no.s C-64442 and GRE-100 are specified as new in the component base details.

Solution: Alter the component base details as required.

Check for obsolete components.

Customer Harness: Steering Harness Issue: a
Internal Harness: Steering Harness Issue: A
Customer Name: European Motors Model:
Drawing Title: Stat.: Draft

Internal Part No.

AB-45938

Warning: Components which have been specified as **Obsolete** in the **Component Base Details** are listed here.

Example: Part No. AB-45938 is specified as obsolete in the component base details.

Solution: Check the component definition in **Update Component Base Details**, or change the component for another one, as no obsolete component should be used in a current harness.

Check for components / wires at nodes which do not exist on the branch config.

Customer Harness: Steering Harness Issue: a
 Internal Harness: Steering Harness Issue: A
 Customer Name: European Motors Model:
 Drawing Title: Stat.: Draft

Node	Route	Cavity	Wire	End	Internal Part Number	Customer Part Number
c90					C-64442	CONN-5000
sp20	11				C/S	

Warning: Components or wires which are located at a node that does not belong to the harness are listed here. This node will not be drawn, and this case may be acceptable if the node is to receive some items for tying up or packaging the harness, or if it is a splice which needs to be placed on the harness.

Example: Node C90 has Part No. C-64442 present but it is not on the branch configuration.

Solution: Check in **Report Harness Branch Configuration** if the node is missing, or in **Report Components** if the component has been assigned to the wrong node.

Check for invalid / unselected part numbers.

Customer Harness: Steering Harness Issue: a
 Internal Harness: Steering Harness Issue: A
 Customer Name: European Motors Model:
 Drawing Title: Stat.: Draft

Wire Number/ Item Type	Comp. Int. Part No.	Node 1 Route 1 Cavity 1 Node 2 Route 2 Ins. Layer	Cavity 1/ Mult.Number
Multicore Wire	Mult-B No Mult Part		
Tube	SP10 Unselected	J10	10

Warning: The components or wires for which an invalid part number has been specified or for which the system has failed to select a valid part number are listed here.

Example: No multicore part no. has been found and there is an unselected tube between SP10 and J10.

Solution: Check the component definition in the **Component Base Details**, as well as the sub components listed in the **Housing Definition**.

Check for cavity components not fitting wires.

Customer Harness: Steering Harness Issue: a
Internal Harness: Steering Harness Issue: A
Customer Name: European Motors Model:
Drawing Title: Stat.: Draft

N.B. Includes terminals at nodes not fitting wires - cavity column is blank in report

Node	Cavity	Internal Part Number	Wire Mat.	Wire Spec.
C10	1	3202511J9	TWC	16/.2
C10	2	CP-6322	TWC	16/.2

Warning: The cavity components (i.e. terminals or cavity seals) which do not fit the wires they have been assigned to are listed here.

Example: At node C10 cavity 1 has a terminal which doesn't fit the wire and cavity 2 has a cavity plug which doesn't fit the wire.

Solution: Check the **Cavity Component Management** form and the **Component Base Details (component maintenance)** check the **Attributes** of the terminals/seals for the wires specified (i.e. Single/Multiple Wire Termination).

Check for wires/cavity plugs at same cavity.

Customer Harness: Steering Harness Issue: A
Internal Harness: Steering Harness Issue: A
Customer Name: European Motors Model:
Drawing Title: Stat.: Draft

Node	Cavity	Internal Part Number	Wire Number
C10	2	CP-6322	8

Warning: Cavities which contain both a plug and a wire end are listed here.

Example: Node C10 cavity 2 has a cavity plug aswell as wire no. 8.

Solution: Rerun the **Processing Options** and **Engineering Calculations**.

Check for center strip nodes not on the wire route.

Customer Harness: Steering Harness Issue: A
Internal Harness: Steering Harness Issue: A
Customer Name: European Motors Model:
Drawing Title: Stat.: Draft

Node	Route	Wire
SP10		3
SP10		4

Warning: Wires for which a center strip has been specified in a splice, ultrasonic weld or solder sleeve which is not on the route of the wire.

Example: Node SP10 has wire no.s 3 and 4 as center strips, which do not appear on the wire route.

Solution: Check the splice **Center strip** definition and the location of the wire ends.

Check for unsourced components.

Customer Harness: Steering Harness Issue: a
Internal Harness: Steering Harness Issue: A
Customer Name: European Motors Model:
Drawing Title: Stat.: Draft

Internal Part No.

127128031
3202511J9
329999932
6860440292
AB-45938
C-64442
C-75369
C3-6320
CL1234
CP-58365
SW-1234
SW-1235
T-54045
tube

Warning: Components which have no supplier number are displayed here.

Example: All the above components do not have supplier information issued against them.

Solution: Input supplier information against those components listed here.

Check for wires at cavity 'X' at splices and welds.

Customer Harness: Steering Harness Issue: a
Internal Harness: Steering Harness Issue: A
Customer Name: European Motors Model:
Drawing Title: Stat.: Draft

Node	Internal Part Number	Componen Group	Wire Number
SP10	329999932	Splice	7

Warning: Wires are located in cavity X of a splice or ultrasonic weld.

Example: Wire No. 7 is still present in cavity X.

Solution: Run the option **Distinguish Left/Right Splice Wire Entry** from the **Engineering Calculations**.

Check for terminals / seals with no wires.

Customer Harness: Steering Harness Issue: a
Internal Harness: Steering Harness Issue: A
Customer Name: European Motors Model:
Drawing Title: Stat.: Draft

Node	Cavity	Terminal Part Number	Seal Part Number
C10	3	t-54084	CS-6320

Warning: Any terminal or seal located in a cavity where there is no wire present is listed. This usually happens when a wire is removed from a cavity but the cavity components are left on the harness.

Example: Cavity 3 has a terminal and seal selected but no wire present.

Solution: Check the wires in **Cavity Component Management** and from the **Component Base Details (component Maintenance)** check the **Attributes** of the terminals/seals for the wires specified (i.e. **Single/Multiple Wire Termination**).

Check for wire/insulation node/route inconsistencies.

Customer Harness: Steering Harness Issue: a
 Internal Harness: Steering Harness Issue: A
 Customer Name: European Motors Model:
 Drawing Title: Stat.: Draft
 Key to inconsistencies :-

1N = End 1 of item has node that does not exist on the branch config.
 1I = End 1 of item has invalid node/route position, but node does exist on the branch config.
 1C = End 1 of item has invalid option, but node does exist on the branch config.
 2N = End 2 of item has node that does not exist on the branch config.
 2I = End 2 of item has invalid node/route position, but node does exist on the branch config.
 2C = End 2 of item has invalid option, but node does exist on the branch config.

Ins. Lay./	Wire No./		
Start Node Start Route Start Cav. End Node	End Route End Cavity Option Ins. Part		
SP20 C40	X 1 C10 10 SP10	2 ALL 11 UNSELECTED TUBE N	Y N N N N Y N N N

Warning: This section will be specified if any routing of a wire or insulation specified is invalid. The Harness Checks report provides keys to inconsistencies.

Example: End1 or wire 11 (Node SP20) does not exist on the branch configuration. End 1 of a selected tube (Node C40) has an invalid route but appears on the branch configuration.

Solution: Check the wire routing by clicking on the wire. Check Branch insulations by showing and modifying Insulation Tables.

N

Check for component housing definition validity.

Customer Harness: Steering Harness Issue: A
 Internal Harness: Steering Harness Issue: A
 Customer Name: European Motors Model:
 Drawing Title: Stat.: Draft

Node	Route	Cavity	Item	Type	Internal Part Number	Main Node Comp.
C10		E001	CL	CL1234	c-61483	
C30		E001	G	GRE-100	c-61483	
C10	1	Term	TERF	3202511J9	c-61483	
C10	1	Seal	Seal	Unselected	c-61483	
C10	2	E001	plug	CP-6322	c-61483	
C20	1	E001	PLUG	CP-58365	C-64442	
C20	2	Seal	Seal	Unselected	C-64442	
C40	2	E001	PLUG	CP-58365	C-64442	

Warning: Any inconsistencies in the housing definition of the components will be specified here.

Example: Item E001 (Node C10) is present as an extra node component but is not in the housing definition of the main component.

Solution: Check the **Housing Definition** of the component (i.e. **Attributes** in the **Component Maintenance form**).

Check for mandatory selection validity.

Customer Harness: Steering Harness Issue: A
Internal Harness: Steering Harness Issue: A
Customer Name: European Motors Model:
Drawing Title: Stat.: Draft

Note: Qty.1 = Number of items currently on the harness.

Qty.2 = Number of items that would be present after mandatory component re-selection.

Node	Route	Ins. Type/ Layer/ Cavity	Wire Num.	End	Part Number	Main Node	Part No.
Qty.1	Qty.2						
C30		AB-45220			C-75369	1	0

Warning: This section will specify any inconsistencies in the selection of mandatory items, i.e. the number of items currently on the harness not matching the mandatory selection criteria.

Example: Part No. AB-45220 is currently on the harness but would not be re-selected as a mandatory component.

Solution: Check the mandatory items specified under the **Housing Definition** in the **Component Base Details**, and rerun the option **Select Mandatory Items** from the **Engineering Calculations**.

Check for branches with no wires or multicores.

Customer Harness: Steering Harness Issue: a
Internal Harness: Steering Harness Issue: A
Customer Name: European Motors Model:
Drawing Title: Stat.: Draft

Start Node	Route	End Node	Route	Option
80		J20		
C10		J10		
C20		J10		
C30		J20		
c40		g20		
C40		J20		
g20		sp10		
j10		g10		
j10		s10		
J20		SP10		
s10		c20		

Warning: Branches on the **Branch Configuration** where no wire or multicore is routed are listed here.

Example: Branch 80-J20 , Branch C10 – J10 etc..

Solution: Check the **Cavity component Management Details** or edit the wire.

Check for components with cavity seals/plugs marked as 'none'.

Customer Harness: BOB Issue: A
Internal Harness: Bob Issue: a
Customer Name: European Motors Model:
Drawing Title: Stat.: Draft

Node	Cavity	Component Type	Internal Part Number
C10	Cav. Plugs Cavity Plug	CP-58365	

Warning: Component has cavities for which the use of plugs or seals has not been specified but plugs or seals are present (manually added)

Example: Node C10 has Cavity Plugs present but the check box is not checked in Update Components and Terminations.

Solution: Check whether the connector needs seals/plugs, in the **Cavity Details** in the **Edit Connector** form.

Check for components using more cavities than registered.

Customer Harness: Steering Harness Issue: a
Internal Harness: Steering Harness Issue: A
Customer Name: European Motors Model:
Drawing Title: Stat.: Draft

Node	Cavities Used	Cavities Registered	Internal Part Number
C10	3	2	c-61483
C30	6	2	c-61483

Warning: Components for which more cavities have been populated than the number of cavities existing are specified here. In Designer it is possible that when placing the Connector Table, too many cavities have been added. This situation frequently occurs with ring terminals where the number of cavities has not been specified.

Example: Node C10 has 2 cavities registered but uses 3.

Solution: Check the **Cavity Details** field of the connector/terminal in the **Component Maintenance** form.

Check for wires with the same colour at connectors.

Customer Harness: Steering Harness
 Internal Harness: Steering Harness
 Customer Name: European Motors
 Drawing Title:
 Node Colour

Issue: a
 Issue: A
 Model:
 Stat.: Draft

Node	Route	Cavity	Wire Number	Wire Colour	Marker
C10	1	7		B	
C10	2	8		B	
C20	2	1		B	
C20	2	4		B	
C20	3	4		B	
C30	2	2		B	
C30	3	10		b	
C30	4	11		b	
C30	5	11		b	
C30	6	10		b	
C40	1	8		B	
C40	3	5		B	
g10	1	1		B	
g10	2	2		B	

Warning: Wires of the same colour are present at a connector.

Example: Node C10 has wires 7 and 8 present which are both B (Black).

Solution: Change the wire specification if required in Capital Library

Check for wires with different colours at splices/weld/solder sleeves.

Customer Harness: Steering Harness
 Internal Harness: Steering Harness
 Customer Name: European Motors
 Drawing Title:
 Node

Issue: a
 Issue: A
 Model:
 Stat.: Draft

Node	Part Number	Wire Number	Wire Colour
SP10	329999932	10	B
SP10	329999932	3	g
SP10	329999932	5	B
SP10	329999932	7	B

Warning: Splices, welds and solder sleeves which have wires of different colors, are specified here.

Example: Node SP10 contains 3 wires B (Black) and 1 wire g (Green).

Solution: Check the color of the wires using **Insert / Edit Wire** or Report on **Wire**.

Check for wires with zero strip length.

Customer Harness: Steering Harness Issue: A
Internal Harness: Steering Harness Issue: A
Customer Name: European Motors Model:
Drawing Title: Stat.: Draft

Wire	End	Node	Route	Cavity	Part Number
10	2	C30	6	c-61483	
11	1	C30	4	c-61483	
11	2	C30	5	c-61483	
2	1	C30	2	c-61483	
5	2	C40	3	C-64442	
8	1	C40	1	C-64442	
8	2	C10	2	c-61483	
9	1				
9	2				

Warning: Wires belonging to the harness for which no strip length has been specified, is listed here.

Example: Wire 10 end 2 Node C30 Cavity 6 has zero strip length.

Solution: Check the **strip / multi-strip** field of the terminal in the **Component Maintenance** form.

Check for tape over single wire / multicore.

Customer Harness: Steering Harness Issue: A
Internal Harness: Steering Harness Issue: A
Customer Name: European Motors Model:
Drawing Title: Stat.: Draft

Start Node	Route	End Node	Route	Type	Tubed Notes
C10		SP10		Multicore NO	
SP10		C20		Multicore NO	

Warning: Single wires or multicores belonging to the harness for which a tape insulation has been specified, are listed here.

Example: There is tape over a multicore on branches C10 – SP10 and SP10 – C20.

Solution: Check whether the tape is needed and if not, modify the **Branch Insulations**.

Check for incorrect number of cavity plugs.

Customer Harness: Steering Harness Issue: a
Internal Harness: Steering Harness Issue: A
Customer Name: European Motors Model:
Drawing Title: Stat.: Draft

Node Int.Part Number No. Cavities No. Plugs Cavs. With Wires Cavity Plugs?

Node	Int.Part Number	No. Cavities	No. Plugs	Cavs. With Wires	Cavity Plugs?
C10	C-61483	2	1	2	YES

Warning: This section will list those connectors, which contain more Cavity Plugs than required. (i.e. more cavity plus than available cavities without wires)

Example: Node C10 has 2 cavities both containing wires and also 1 plug.

Solution: Run the option **Select Blanking Plugs** the **Engineering Calculations**.

Check for presence of sealed and unsealed connectors.

Customer Harness: Steering Harness Issue: a
Internal Harness: Steering Harness Issue: A
Customer Name: European Motors Model:
Drawing Title: Stat.: Draft

Node Internal Part Number Sealed?

C10	c-61483	YES
C20	C-64442	YES
C30	c-61483	YES
C40	C-64442	YES
c90	C-64442	NO

Warning: Checks each connector on the harness and specifies whether they are sealed or not.

Example: Node C10 has Connector C-61483 which is sealed.

Solution: Check the seals checkbox within the **Edit Connector** form if seals are required, or uncheck it if they are not.

Check for junctions under unslit tubing.

Customer Harness: Steering Harness Issue: A
Internal Harness: Steering Harness Issue: a
Customer Name: European Motors Model:
Drawing Title: Stat.: Draft

Start Node	Route	End Node	Route	Layer	Part Number	Junction
t10		t20		10	TU-54123	c20

Warning: Junctions on the harness with branches which have insulation specified as unslit tubing.

Example: Branch t10 – t20 contains a junction at node C20

Solution: Check **Bundle Configuration**, **Modify Insulations** Add new **Insulation** if required.

Wire and node option names not on harness.

Customer Harness: Steering Harness Issue: A
Internal Harness: Steering Harness Issue: a
Customer Name: European Motors Model:
Drawing Title: Stat.: Draft

Node	Item Splice?	Main Component Wire	Customer Number
		2	
	Option: ALL	4	

Warning: Inconsistencies in the option names assigned to wires and nodes belonging to a composite or derivative harness are specified here.

Example: Wires 2 and 4 have option all against them but the option is not specified as being in any derivatives.

Solution: Check the options against branches on the Harness Design, and against wires in the **Edit Wire** form. Also check the Composite table, and **Insert Options** if required.

Unnamed cavities on the harness.

Customer Harness: Steering Harness Issue: a
Internal Harness: Steering Harness Issue: A
Customer Name: European Motors Model:
Drawing Title: Stat.: Draft

Node	Cavity	Wire	Error
C30	3	10	Valid cavities for connector: 1-2
C30	4	11	Valid cavities for connector: 1-2
C40	A	11	Valid cavities for connector: 1-2
C40	B	10	Valid cavities for connector: 1-2

Warning: Cavities that are used on the harness but have not been registered against the connector in the component database.

Example: Node C30 has cavities 3 & 4 populated on the harness but in the component database only cavities 1 and 2 have been registered.

Solution: Check the name and number of cavities using **Edit Wire**, and **Modify Cavity Parameters** as required.

In-Line Connectors on the harness.

Customer Harness: Steering Harness Issue: a
Internal Harness: Steering Harness Issue: A
Customer Name: European Motors Model:
Drawing Title: Stat.: Draft

Node 1	Node 2	Different	Error
10	20	No	Number of cavities mismatch

10	20	No	In-line cavity has no wire located
10	20	No	In-line wire color does not match
10	20	No	In-line wire material does not match
10	20	No	In-line wire CSA does not match
10	20	No	In-line wire specification does not match

This node Cavity Mat. Other node Cavity Mat. Customer Harn. Number Issue
Internal Harness Number Issue

C10 1 GP C20 1 TP

Warning: the number of cavities at the connector at one side are different from the number of cavities at the connector at the other

Example: connector at node 10 has 5 cavities, connector at node 20 and 8 cavities

Designer Solution: if this will cause problems then change the connector using **Add Connector**

Warning: there is a wire in a cavity on one side and the pin mapped to this cavity on the other has no wire

Example: connector at node 10, cavity 1 has a wire inserted. This cavity is mapped to connector 20, cavity 3 and this cavity is empty.

Solution: using **Insert / Edit Wire** modify the locations of the wire/s accordingly

Warning: one or more details of the wire on one side differ from that of the wire on the other side

Example: wire in cavity 1 has a different color, CSA, material or specification than that of the wire in the mapped cavity

Solution: using **Insert / Edit Wire** modify the details of the wire/s accordingly

Warning: the terminal material on side is different from the terminal material on the other side

Example: the terminal selected for wire in cavity 1 has a different material than that of the terminal selected for the wire in the mapped cavity

Solution: using **Insert / Edit Wire** specify the details of the terminal material to be selected

Wire marker colors on the harness.

Customer Harness: Steering Harness

Issue: a

Internal Harness: Steering Harness

Issue: A

Customer Name: European Motors

Model:

Drawing Title:

Stat.: Draft

Item Type

Option

Item Name

Location Error

Wire

1 ALL

End 1 Marker color R conflicts with wire color R.

Warning: the color chosen for the wire marker is the same as the wire color

Example: for the above example, the marker color for wire 1 is red and the wire color has been changed since adding the wire marker

Solution: using **Insert / Edit Wire** modify the color of the wire accordingly

Wire through node inconsistencies.

Customer Harness: Steering Harness Issue: a
Internal Harness: Steering Harness Issue: A
Customer Name: European Motors Model:
Drawing Title: Stat.: Draft

Wire No.	Option	Node/Multicore	Route/Option	Description
1	ALL			wire does not go through the shortest route
1	ALL	10		wire goes through the same node twice
2	ALL			wire does not go through the shortest route

Warning: the route that has been specified for the wire is not the shortest OR has been specified twice.

Example: there are two possible routes that a wire could take and the one specified means that the wire length is longer.

Solution: use Set Path if necessary to modify the route of the wire

Branch insulation through node inconsistencies.

Customer Harness: Steering Harness Issue: a
Internal Harness: Steering Harness Issue: A
Customer Name: European Motors Model:
Drawing Title

Start Node	Route	End Node	Route Option	Layer	Through Node Route Description
10	shortest route	20	10	insulation does not go through the	
10		20	10 10	insulation goes through the same	
	node twice				

Warning: the route that has been specified for the insulation is not the shortest OR has been specified twice.

Example: there are two possible routes that the insulation could take and the one specified means that the insulation has taken the longest route.

Solution: use set Path if necessary to modify the route of the insulation

Splices with 3 or fewer wires.

Customer Harness: XY-P014-J01-X34 Issue: C
Internal Harness: XY-P014-J01-X34 Issue: C
Customer Name: European Motors Model:
Drawing Title: Stat.: Draft

Node	No.	Splice Type	Int. Part No.	Cust. Part No.	Wires
S10	2	Splice	SP-45220		4, 1
T20	1	Splice			5
T40	0	Splice			
W10	3	Splice			6, 3, 2

Warning: there are splices/welds/solder sleeves on the harness with 3 or less wires

Example: after composite breakdown, 3 or less wires have been left in the splice. Depending on several aspects, the splice may be removed.

Solution: using **Insert / Edit Wire** modify the wire details as necessary

Check for Wire-fits-cavity.

Customer Harness: training Issue: a
Internal Harness: Training Issue: a
Customer Name: European Motors Model:
Drawing Title: Stat.: Draft

Node	Cavity	Error

10	1	Wire(s) failed wire-fits-cavity check.
10	2	Wire(s) failed wire-fits-cavity check.
10	3	Wire(s) failed wire-fits-cavity check.
20	2	Wire(s) failed wire-fits-cavity check.

Warning: the wire which has been specified for the cavity does not fit.

Example:

Solution: using **Insert / Edit Wire** modify the wire details as necessary

Wire Length Calculations

There are many factors involved in calculating wire cutting lengths:

The Wire Dead-Length

From the data read in the **Branch Configuration** and the wire ends assigned in the **Cavity Details**, the lengths of every branch, which the wire passes through, are totaled.

This gives the dead length of the wire.

Component Add-On and Knock-Off

For each end of the wire, the Base Details of the component is examined. The conventions for dimensioning are checked in the Customer Details

If the harness is dimensioned to the FRONT of connectors,
The main component KNOCK-OFF value is subtracted from the wire length.

If the harness is dimensioned to the BACK of connectors,
The main component ADD-ON value is added to the wire length.

If the harness is dimensioned to the TIP of terminals,
The cavity component KNOCK-OFF value is subtracted from the wire length.

Note

The **System Parameters** default splice strip length, default solder sleeve strip length and default ultrasonic weld strip length affect the length of wires : in the absence of a component add-on or knock-off, half the value of the parameter is added to the wire length.

Dressed Connector Attributes

If the main component at the node is a dressed connector, the **Dressed Connector Attributes** are examined. The conventions for dimensioning are checked in the **Customer Details**.

If the harness is dimensioned to the FRONT of connectors,
⇒ The CAVITY KNOCK-OFF value for the given route of the wire, is subtracted from the wire length.

If the harness is dimensioned to the BACK of connectors,
⇒ The CAVITY ADD-ON value for the given route of the wire, is added to the wire length.

If the harness is dimensioned to the TIP of the terminals,
⇒ The CAVITY KNOCK-OFF value for the given route of the wire, is subtracted from the wire length.

Note

Some Dressed Connectors can be used on a harness without a route being selected; in this case, the section above "Component Knock-Off and Add-On" applies.

Add-On Per Junction

All junctions which the wire passes through are totaled; the field **Len. Add On / Junc** is checked from the **Customer Details**.

⇒ The value of the parameter is added for each junction on the wire's route.

Note

A junction is a node at which 3 or more branches meet.

Percentage Add-On

The **Customer Details** are checked for the fields **Wire Len. % Add On**, **Wire Len. Max. Add**, **Wire Len. Min Add**.

- ⇒ The percentage add-on is calculated and, provided it is within the boundaries, is added to the wire length.

Wire Length Rounding

The rounding method is read from the **Customer Details**.

- ⇒ The rounding method and value are applied to the wire.

Wire Length Changes

The **Wire Length Changes** details allow to modify the length of wires or to add/subtract an offset value.

- ⇒ The information is read and the calculated wire length is amended.

Multicore Length and Length Changes

Note

Multicore wire cutting lengths are special cases. The length of the longest inner core is not necessarily the length at which the multicore should be cut. It may be longer. The longest length between any inner core ends is taken to be the multicore cutting length.

The **Multicore Wire Length Changes** and **Multicore Sheath Cut Back Length Changes** allow to manually modify/overwrite the calculated multicore cutting length as well as the calculated sheath cut-back length at both ends

- ⇒ The information is read and the calculated multicore wire length is amended.

Appendix 4

Capital Enterprise Reporter

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Getting Started with Capital Enterprise Reporter

Overview

Capital Enterprise Reporter (CER) is a web based application accessible to anyone within the Enterprise. This application is a client server application and provides remote access to the Capital database.

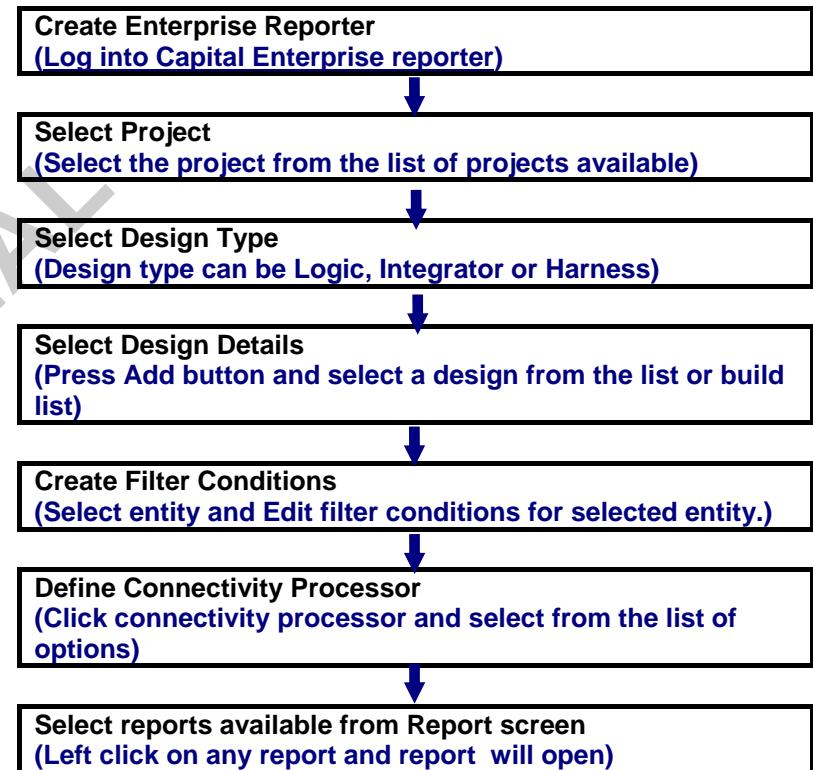
Capital Enterprise Reporter enables users to filter the reports with multiple queries. It enables multiple queries to be grouped and linked by "or, as well as and" operators. A connectivity processor controls what is being reported.

CER supports reports to be generated for the Logic, Integrator and Harness XC designs. The reports available for Harness designs are:

- Harness Component BOM
- Harness Tape BOM
- Harness Tube BOM
- Harness Wire BOM
- Multicore list
- Wire Cutting Chart
- Wire List
- Configured Wire List

Process

The following process flow summarizes the activities to generate a report:



Getting Started with Enterprise Reporter

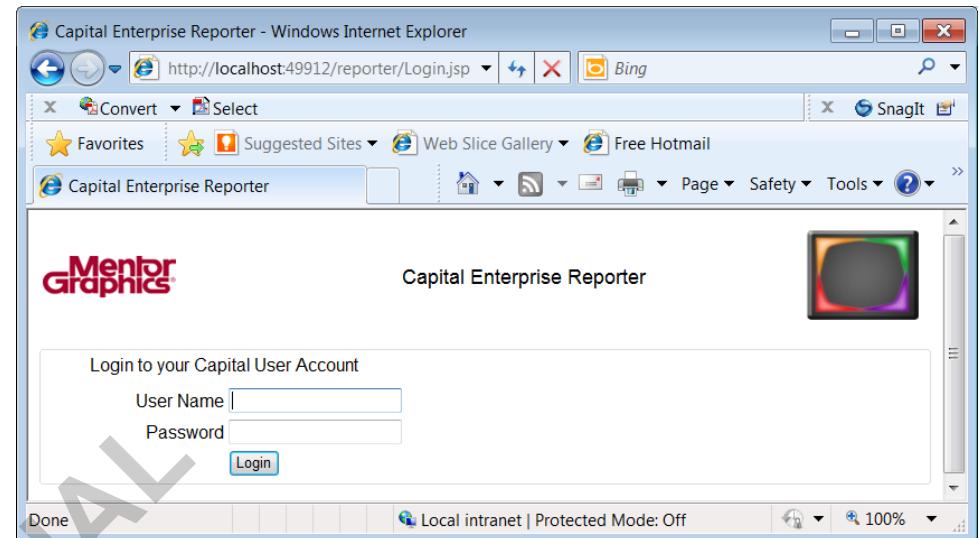
The CER server can start in two modes.

- In Embedded Mode, the server is launched as part of Capital Manager Executable. Start the Capital manager and the server is started along with it.
- In Standalone Mode, the server is started by clicking on windows start button, then select All programs/Capital version [Version number]>Start Capital Enterprise Reporter Server.

Connect to the Reporter Server

Once the server has been started follow the steps to connect to the reporter server.

1. Launch the web browser.
2. In the address field enter the URL:
[http://\[server_name\]:\[port_number\]/reporter/Login.jsp](http://[server_name]:[port_number]/reporter/Login.jsp)
3. Press enter and the following screen appears.
4. Enter the Login and password to open Capital Enterprise Reporter



Scope of Report Generation

Users need to define a scope for the report by defining the data required for report generation. The users must select project, design type and design details for generating any report.

The user can select any of the design types available for selection.

- ⇒ Logic
- ⇒ Integrator
- ⇒ Harness

Design Details contain the list of designs and build lists available for the selected project. The users can select multiple designs or build lists for generating the report.

Capital Enterprise Reporter - Windows Internet Explorer

File Edit View Favorites Tools Help

user:system

Mentor Graphics

Home Refresh Help Log out

Project Report Library Report

Project Details

Project and Design Type

Project : Quick Start - Automotive Generative Design Type : Harness Design

Design Details

Design Type/Build List	Attribute	Operator	Value
Design Selection			

Show Designs

Configuration Details

Design Details	Configurations
	Add Configuration

Next

Filter conditions

Filtering provides a way to find and view objects that satisfy certain conditions. The users can create reports with very rich queries on the scoped data. Filtering provides a greater control on what is being reported. Reports can be generated with only desired objects displayed.

To open the Filter window users must click on **Next** in the previous window.

Entities can be selected from the list and query expressions can be applied for the selected entities.

For example: Filtering on the entity device and creating the filter conditions using queries:

- rating =20 or option expression = A-SBWFR.

This query will show all the devices either with rating 20 or with option expression A-SBWFR.

Capital Enterprise Reporter - Windows Internet Explorer

File Edit View Favorites Tools Help

user:system

Mentor Graphics

Home Refresh Help Log out

Previous

Create Filter Conditions

Entity	Expression
Connector	((attribute Name) = 'C-61332%')

Remove

Connectivity Processor

Next

- ⇒ Once the filter is defined, click **OK** to return to the main window

Note

Defining Filter conditions is optional, users can directly generate reports selecting project, design type and design details.

Connectivity Processor:

The Connectivity processor is a way of filtering the data based on the connection details of the objects in the design. There are different types of connectivity processors available for selection such as

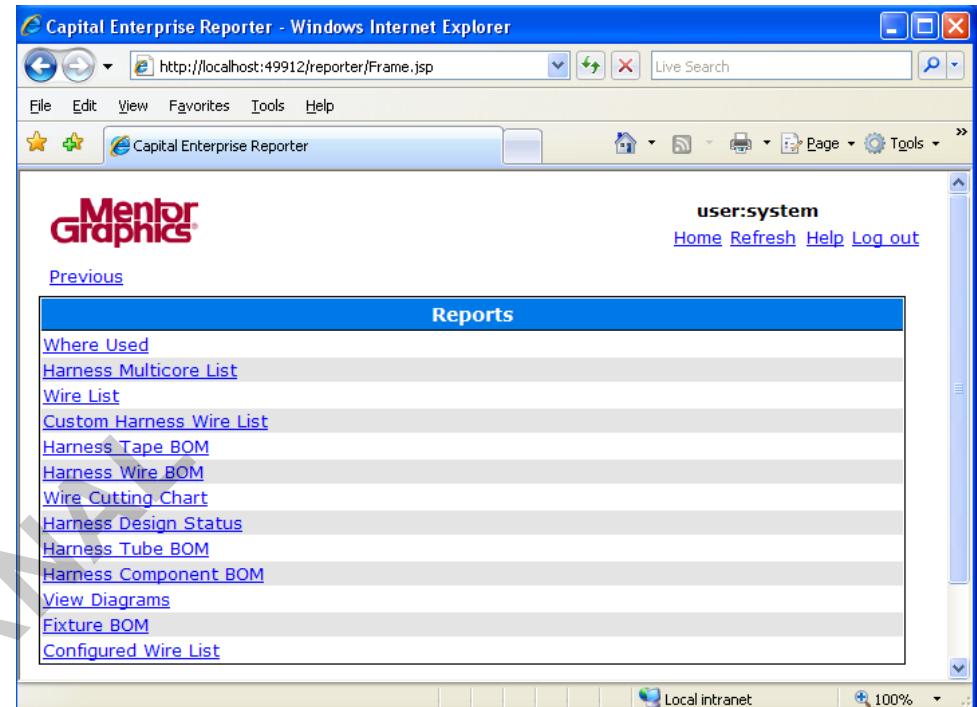
Directly connected: searches for conductors directly connected to filtered pins

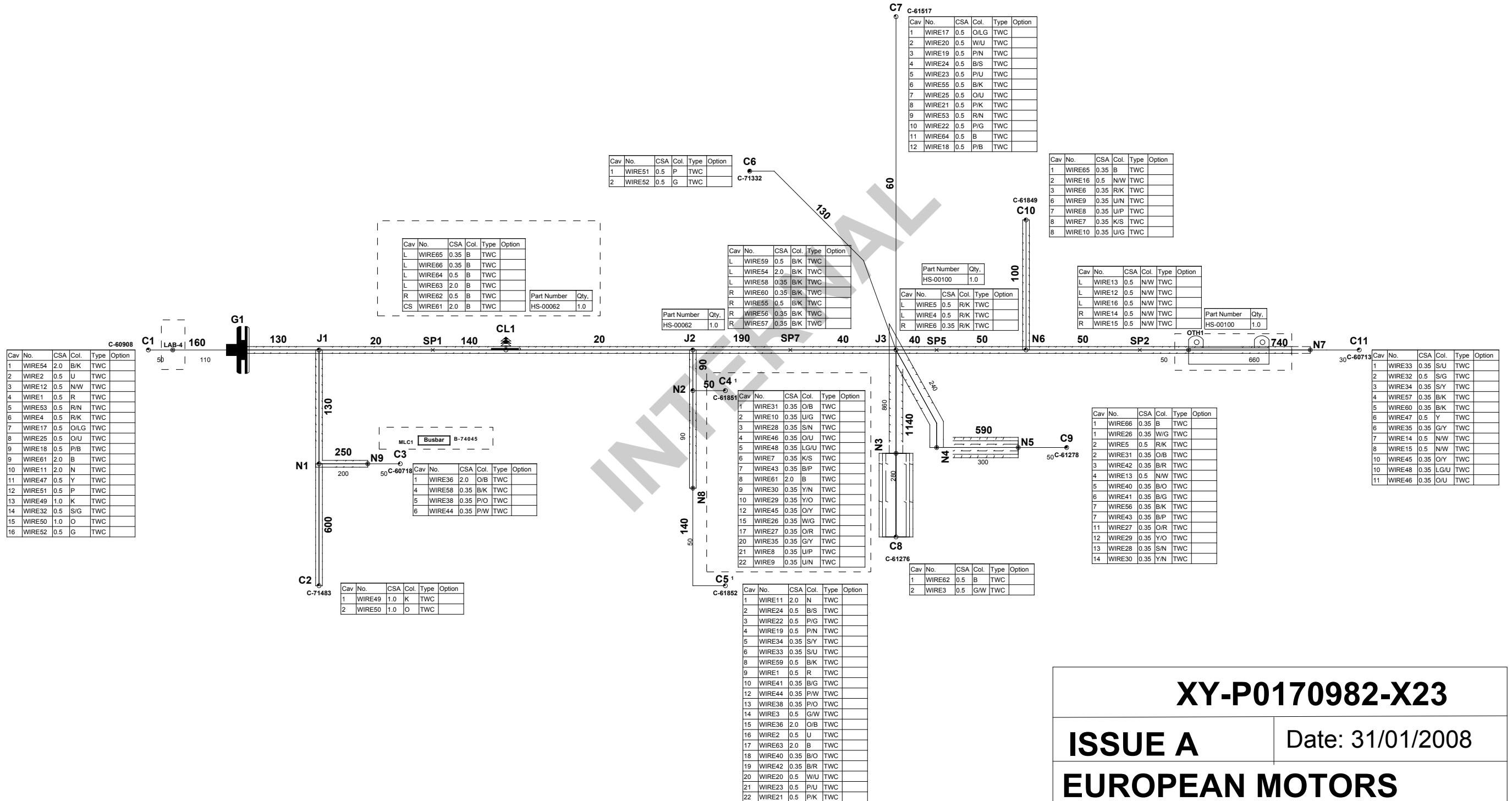
Connectivity between pin lists: Reports on all conductors navigating through splices and inline connectors

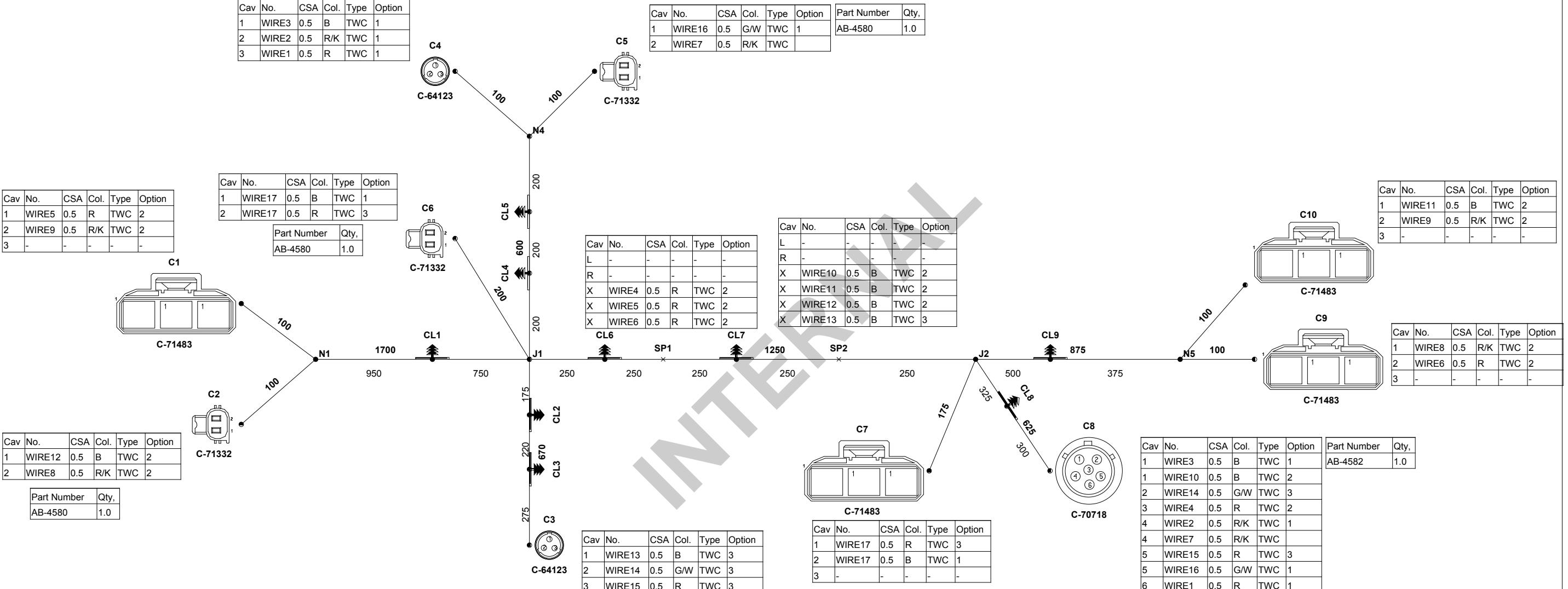
No processor: Reports only on entities satisfying the filter condition

Once the Filters and connectivity information have been defined, click **Next** to move to the following window where users can choose the type of report to be generated.

Once users click on the report type a report is generated.







P022-400-J44

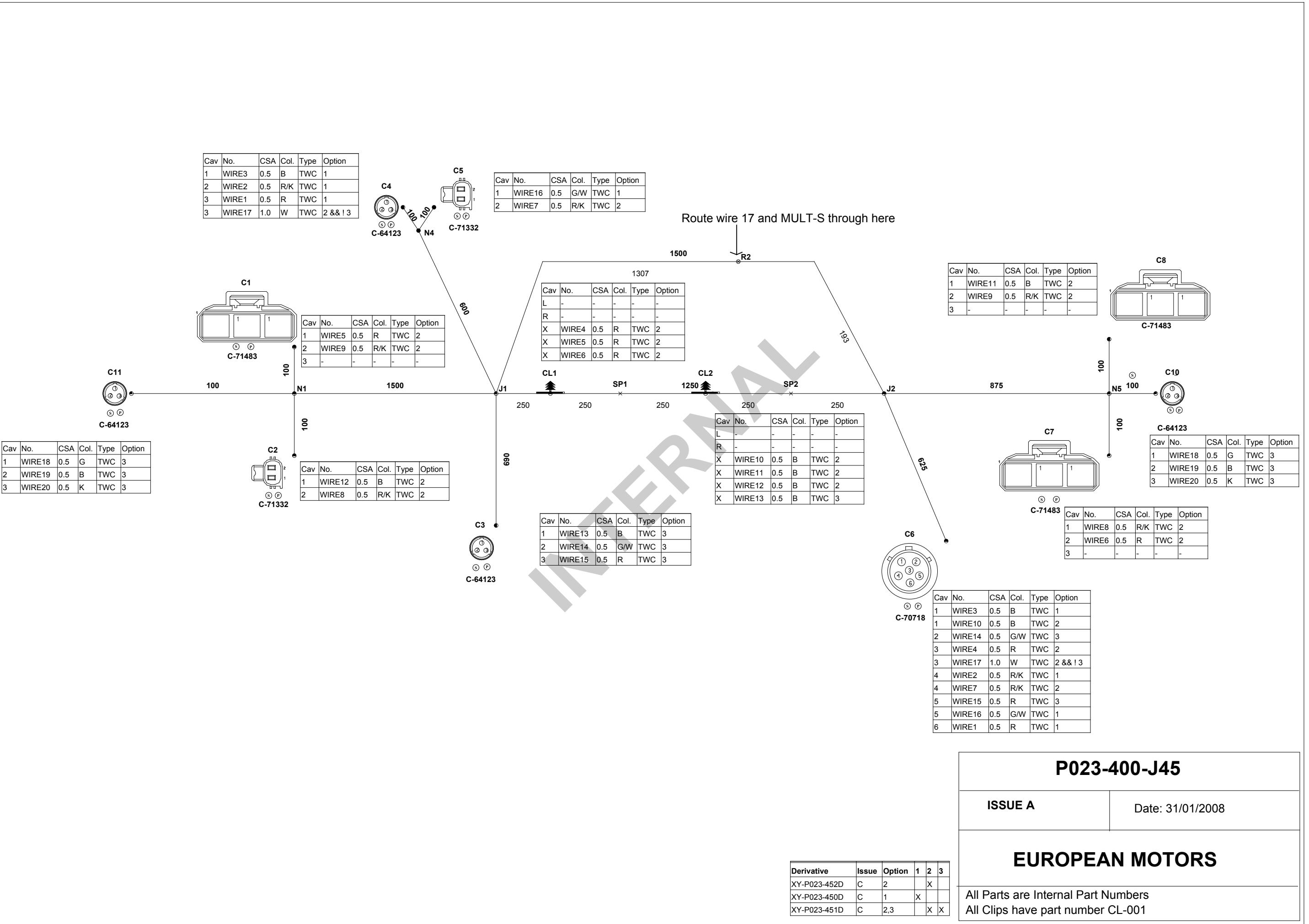
ISSUE A

Date: 31/01/2008

EUROPEAN MOTORS

All Parts are customer part numbers
All Clips have part number CL-001

Derivative	Issue	Option
XY-451D	A	2,3,5
XY-450D	A	1,4
XY-452D	A	2,6



INTERNAL

PART NUMBER: 072633