This document is for information and instruction purposes. Mentor Graphics reserves the right to make changes in specifications and other information contained in this publication without prior notice, and the reader should, in all cases, consult Mentor Graphics to determine whether any changes have been made.

The terms and conditions governing the sale and licensing of Mentor Graphics products are set forth in written agreements between Mentor Graphics and its customers. No representation or other affirmation of fact contained in this publication shall be deemed to be a warranty or give rise to any liability of Mentor Graphics whatsoever.

MENTOR GRAPHICS MAKES NO WARRANTY OF ANY KIND WITH REGARD TO THIS MATERIAL INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE.

MENTOR GRAPHICS SHALL NOT BE LIABLE FOR ANY INCIDENTAL, INDIRECT, SPECIAL, OR CONSEQUENTIAL DAMAGES WHATSOEVER (INCLUDING BUT NOT LIMITED TO LOST PROFITS) ARISING OUT OF OR RELATED TO THIS PUBLICATION OR THE INFORMATION CONTAINED IN IT, EVEN IF MENTOR GRAPHICS CORPORATION HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

RESTRICTED RIGHTS LEGEND 03/97

U.S. Government Restricted Rights. The SOFTWARE and documentation have been developed entirely at private expense and are commercial computer software provided with restricted rights. Use, duplication or disclosure by the U.S. Government or a U.S. Government subcontractor is subject to the restrictions set forth in the license agreement provided with the software pursuant to DFARS 227.7202-3(a) or as set forth in subparagraph (c)(1) and (2) of the Commercial Computer Software - Restricted Rights clause at FAR 52.227-19, as applicable.

Contractor/manufacturer is:
Mentor Graphics Corporation
8005 S.W. Boeckman Road, Wilsonville, Oregon 97070-7777.
Telephone: 503.685.7000
Toll-Free Telephone: 800.592.2210
Website: www.mentor.com
SupportNet: supportnet.mentor.com/
Send Feedback on Documentation: supportnet.mentor.com/doc_feedback_form

TRADEMARKS: The trademarks, logos and service marks ("Marks") used herein are the property of Mentor Graphics Corporation or other third parties. No one is permitted to use these Marks without the prior written consent of Mentor Graphics or the respective third-party owner. The use herein of a third-party Mark is not an attempt to indicate Mentor Graphics as a source of a product, but is intended to indicate a product from, or associated with, a particular third party. A current list of Mentor Graphics' trademarks may be viewed at: www.mentor.com/trademarks.

End-User License Agreement: You can print a copy of the End-User License Agreement from: www.mentor.com/eula.
Part Number:073171
# Capital Topology

## Table of Contents

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Overview</td>
<td>1</td>
</tr>
<tr>
<td>Getting Started with Capital Topology</td>
<td>2</td>
</tr>
<tr>
<td>Creating a Topology</td>
<td>3</td>
</tr>
<tr>
<td>Design Scopes</td>
<td>3</td>
</tr>
<tr>
<td>Import/Export of Topology Designs</td>
<td>3</td>
</tr>
<tr>
<td>Place a Skeleton and/or Border</td>
<td>4</td>
</tr>
<tr>
<td>Adding Borders</td>
<td>5</td>
</tr>
<tr>
<td>Defining the Topology</td>
<td>5</td>
</tr>
<tr>
<td>Create Slots</td>
<td>5</td>
</tr>
<tr>
<td>Topology Definition</td>
<td>6</td>
</tr>
<tr>
<td>Assigning Rules/Placing Multiple Devices Into a Single Slot</td>
<td>6</td>
</tr>
<tr>
<td>Assigning Wiring Designs</td>
<td>7</td>
</tr>
<tr>
<td>Synchronizing the Wiring Designs with the Topology Design</td>
<td>9</td>
</tr>
<tr>
<td>Physical Object search</td>
<td>10</td>
</tr>
<tr>
<td>Completing the Topology</td>
<td>10</td>
</tr>
<tr>
<td>Multiple Connector mating</td>
<td>12</td>
</tr>
<tr>
<td>Design Validation</td>
<td>13</td>
</tr>
<tr>
<td>Rules and Constraints</td>
<td>14</td>
</tr>
<tr>
<td>Assigning Signals</td>
<td>15</td>
</tr>
<tr>
<td>Splice visualization</td>
<td>16</td>
</tr>
<tr>
<td>Wire Navigation and Reporting</td>
<td>16</td>
</tr>
<tr>
<td>Back Annotation of Wires</td>
<td>17</td>
</tr>
<tr>
<td>Harness Synchronization</td>
<td>19</td>
</tr>
<tr>
<td>Route codes</td>
<td>19</td>
</tr>
<tr>
<td>Bridging data out</td>
<td>23</td>
</tr>
<tr>
<td>Topology Styling</td>
<td>25</td>
</tr>
</tbody>
</table>
Overview

This course will cover the functionality and capabilities within the Capital Topology tool. Starting from the beginning of a design cycle where a user wants to create a topological view of their platform and then sync that topology with a set of wiring designs.

You will cover:

- Creating a simple topology
- Associating wiring designs
- Filling in the gaps
- Routing your wires
- Associating signals
- Reporting
Getting Started with Capital Topology

Capital Topology is a tool that offers a topological overview for the wiring in an entire platform. Using simple techniques it is possible to create a full platform level view displaying the layout of the various harnesses that will make up the topology of the platform. From this the user can then assign a full set of wiring designs to the topology and automatically map the devices and connectors to ‘slots’ and route the wiring through the harness topology.

This course assumes that the user is already familiar with creating detailed wiring designs in Capital Logic, and that you have the Quick Start – Aerospace Interactive project loaded.

This course will be working with the Quick Start – Aerospace Interactive project that is available on the community. The data includes a set of 3 wiring designs for some lighting in and around a platform. There are also 2 sample topology designs here and the exercises are designed to create a similar design to the Business Jet topology.

Concurrency

Many users may work on a single Topology plane at the same time by using concurrency settings. The principle behind this function is that the topology will be split into several harnesses and users will be able to lock the harness they are working on so that others cannot update it at the same time.

Users may use the browser to view which designs are locked or have been recently updated. When working on a locked harness users can chose to hide or dim the other harnesses that belong to the same topological plane.

Wiring synthesis may still be performed on the locked harness, however wiring information will only be generated up to the inline point of that harness and the next harness on the plane.

To lock an object or harness:

⇒ Open the concurrency tab
⇒ Select the required user mode
⇒ Use the filter icons to display the required information in the browser (harness, slots, connectors)
⇒ Use the padlock icon to turn the lock on/off
Creating a Topology

⇒ Select a sub-folder or the project name in the Project Browser
⇒ Right click and select New Topology Design
⇒ A New Capital Topology Design dialog box will be displayed prompting for entry of the design details
⇒ The new design must have at least a name and a revision. The user can optionally provide both a short description and a long description which will help to identify the design and its purpose.
⇒ At this point the user may want to set the release level for the design. It is automatically set to draft by default.
⇒ Creating a new design will also create a new diagram. Set the name of the diagram

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

Import/Export of Topology Designs

Topology designs may be imported and exported from Capital Project or Capital Logic.
To export a design:
⇒ Right click on the design in the Project Browser and select Export design
⇒ Select the ‘as dataset’ button to export any symbol or library data
⇒ Choose a location for the data to be exported to

Note
When importing a design with a dataset attached, any conflicts with library or symbol data will be reported
Place a Skeleton and/or Border

Place a skeleton outline to provide a location reference for placement of slots. To place a skeleton on a Capital Topology diagram:

⇒ Select Graphics / Skeleton

The user can select the desired skeleton from the available libraries. The skeleton will already have been created in Capital Symbol.
Adding Borders

Borders provide a template for printing diagrams according to company standards. They often contain a title block area where information such as Name, Issue and Date can be specified. A border also defines the page size used when printing a diagram. Borders are managed within a symbol library, which means users can set up different libraries for different customers, vehicles, aircraft etc. To place a border:

⇒ Select Graphics/Border
⇒ Select the required border from the library list

Defining the Topology

A topology is made up of a collection of objects such as Slots, Bundles and Connectors. A Slot is a virtual placeholder for a physical device on a platform. Each device within a slot must have a Virtual Connector which represents the actual harness connector that is defined by the wiring designs. Multiple devices can be placed into slots and then their connectors are connected to harness bundles. Bundles define the path of the wiring that connects all of the electrical devices together across the entire plane.

Create Slots

Devices defined in the Wiring Design data are placed into the Slots created in the Topology Design. Capital Topology offers automation and flexibility in the placement of devices within these slots. To create a slot:

⇒ Select the Slot icon from the main toolbar
⇒ Click within the diagram to draw a rectangle defining the slot
⇒ Double click on the slot to bring up the properties window and give it a name.
⇒ Right click on the slot and create a new Virtual Connector
⇒ Double click on the Virtual Connector to bring up the properties and give it a name

Once placed a slot can be rotated or flipped to better suit the topology. A virtual connector can be moved by right click/move virtual connector.

When wiring designs are associated with a topology the tool will try to automatically place a physical device in a slot with the same name (this rule also applies to connectors). By default all placement in Capital Topology is done by name.
It is possible to override the name rule and define new rules by which devices are placed inside slots.
This method also enables the tool to place multiple devices into a single slot.
The tool will automatically create connectors when drawing a bundle up to the outline on a slot (see later notes)
Topology Definition

Once the slots and connectors are placed on the plane they can be joined together with bundles:

The Create Bundle icon is used to create the harnesses that will connect all of the connectors together. Click on the diagram to start drawing, clicking again to place a grip point, double clicking or connecting the bundle to a connector will complete the bundle.

Holding down the Shift key when pressing the toolbar button will force the tool to remain in bundle creation mode.

Connecting a bundle directly to a slot will automatically create a new virtual connector.

Where bundle loops are introduced a reference through node will be created which will aid in the routing of wires, the synchronization with Harness XC and bridging data out to Catia.

Exercise 1
Create the placeholder slots and basic bundle topology

Assigning Rules/Placing Multiple Devices Into a Single Slot:

Rules within Capital Topology will allow the tool to automatically try to match up and place the physical devices in the wiring designs with the empty slots in Topology when the 2 are ‘synchronized’ This is also the only way to get multiple devices into a single slot on the topology. This becomes extremely important when it comes to placing grounds for example.

To add a rule to a slot
⇒ Create a new slot called GND
⇒ Right click on the slot and select ‘Properties’
⇒ Select the ‘Rules’ tab and press the Add Constraint button
⇒ From the list of constraints select ‘Place by Attribute/Property and press ok
Click on the underlined ‘Slot Name’ and change the text to ‘GND.*’

When associating or syncing the wiring designs with the topology all ground devices will be placed inside this slot. This can then be repeated to place other devices into manually created slots, to create for example a bank of switches or relays. The constraint could also be tweaked to use a different attribute or property on the wiring such as the part number rather than the default which is Name.

Note
Rules and constraints are also used for other actions in Capital Topology and will be discussed later on

Exercise 2
Create the constraints as described in the worksheet

Assigning Wiring Designs

Wiring Designs should be associated to the Topology Design via a Build List. Build Lists are very useful when multiple Wiring Design sets are available, or when Design Changes occur.

To Associate Logical Designs select Actions / Associate Logical Designs.

⇒ Select the appropriate build list – this must be pre-defined in Capital Logic or Capital Project

If a change is made to a build list the modification is displayed within the associate logical designs dialog:

Note
Selecting the Build List Name will display the designs held in that build list in the Available Logical Designs Window. The devices found in the Wiring Designs will now be listed in the Wiring Tab of the Diagram Browser, under Devices.
In this example the Power design has been removed from the list.

In this example the Power design revision A has been replaced with Power design revision B. If users toggle the filter the modifications can be displayed.
This example displays the build list content when the changed filter is selected.

If a new design is added to replace an existing design users can select to 'replace the selected removed design with the selected newly added design'. This will preserve the wiring already generated for the existing design if device and net names in the removed and newly added designs match. Note that the designs do not have to be revisions of previous designs.

Synchronizing the Wiring Designs with the Topology Design

Once all of the wiring designs have been associated with the topology the data from the wiring designs needs to be linked up to the topology. This can be done manually by pressing the button on the toolbar. Pressing this will bring all of the devices, connectors, wires and splices into the topology design; they will be displayed under the Wiring tab.

Switching windows between a wiring design and the topology design will automatically force a sync between the two. This way any changes made to the wiring will automatically be brought over to the topology design without the need to manually sync the two.

When syncing with the wiring designs the system will try to route all of the wiring along the available bundles. The user can check to see if any wires are not being routed by looking on the Wiring tab (Unrouted wires appear grayed out) and on the Sync message tab in the output window. The messages that appear here give some indication as to why the wires could not be routed. A good example of this may be wires connected to grounds.

If any ground devices have no harness connectors in the wiring designs by default there will be no connector in the topology for the bundles to connect to. In this case holes can be used. Adding a hole to a slot allows the user to attach bundles to a slot via one of these holes. Capital Topology will then route all wires connecting directly to devices inside that slot that have no connectors through that hole.

⇒ Navigate to the GND slot
⇒ Select the Create Hole tool from the toolbar
⇒ Add a hole to the Slot
⇒ Connect the hole to a bundle
Physical Object search
Users can run a search on objects used on the topology plane to identify which wiring design they have come from:

⇒ Right click the object to run the search on
⇒ Select Show Physical Source

The wiring design containing that object reference will then open and the object will be highlighted.

Filtering signals
A filter can be applied to the signal list to highlight wires with a particular attribute or property. Any signal with the defined attribute/property will be displayed in the signals list in the browser.

Completing the Topology
Looking on the Wiring tab of the Plane browser will give an indication of any devices and connectors that are still unplaced on the topology. The icons for these objects will appear grayed out. A filter can be set to display only objects that are yet to be placed (show remaining objects) to make it easier to identify unplaced devices. These will need to be placed on the topology to complete the topology, this can either be done automatically, using the methods described above by creating empty slots and using rules, or they can be placed manually.

To Manually Place Devices:

⇒ List the Devices by selecting Devices in the Wiring Tab of the Topology Diagram Browser:

⇒ Right Click on the Device Name to be placed
⇒ Select Create Slot from the pop-up menu OR
⇒ Move the mouse cursor over to the diagram and draw a new slot
⇒ The device will automatically be placed inside the new slot and the harness connectors for that device will also appear as virtual connectors on the new slot

When the device has been placed, the device icon in the Wiring Tab of the Diagram Browser will be filled in:
⇒ Repeat device placement until all desired devices have been placed into slots
⇒ Connect the connectors that are generated for these slots to the harness

Now re-sync the wiring design and more wires should now appear routed.

If a device is to be placed in a pre-existing slot (where the name does not match the name of the device):

⇒ Right click the unplaced device
⇒ Select place device
⇒ Select the slot you wish to place the device inside
⇒ Click OK

At this point there may still be unplaced connectors and unrouted wires. In this case the user would still have to manually create the connectors for each device that needs to be placed along the walls of the slot. This can be accomplished by double clicking on the grayed out connector on the Wiring tab and placing the connector down on the diagram against the outline of the slot itself. This will create a new connector on the slot with the name matching the harness connector in the Wiring designs. This 1:1 relationship will then be automatically maintained so that changes to the wiring designs will still sync to the correct places in topology as long as the names of the connectors have not changed.

**Inline Connectors and harness Definition**

To enable correct routing of the wires users need to place inline connectors on the Topology diagram. This will split the topology diagram into individual harnesses. All Inlines on the topology plane are displayed under the Inlines node in the Plane browser.

This example contains 2 pairs of inline connectors that need to be placed on the design to complete the topology and split the single harness into 3.

Highlighting one of the connectors and pressing ‘x’ will take the user back to the physical source and give an indication of where this inline needs to be placed.

In the topology diagram, double clicking on one half of an inline pair will place the entire inline on a bundle and split the harness. The direction of the inline pair will be automatically picked up from the source wiring designs.

⇒ Select one half of the first inline pair in the browser and double left click
⇒ Move the mouse over to the diagram, the cursor will change to indicate placing an inline pair
⇒ Move the cursor to the bundle to split, somewhere along the bottom of the harness
⇒ Click to place the inline at the current mouse position along that bundle
⇒ The inline will now be placed, the direction correct, and the harness split at that bundle
⇒ Re-sync the wiring
⇒ All wires from the Lights, Aircraft design will now appear routed
Multiple Connector mating:

Where there are many connectors mated to a single inline plug connector users can ensure wires are mapped to the correct mating half using constraints.

In Logic:
- Create the plug connector instance
- Add the 1st mating half
- Add the subsequent mating halves

In Topology:
- Associate the logic diagram to the topology plane
- Create the device references
- Add the inline plug and mating receptacle, renaming each to match the names given in Logic

⇒ Add constraints to the receptacle:

⇒ Ensure the constraint uses the receptacle names given in Logic
⇒ Refresh Wiring – notice that all inline connectors are now placed and associated wiring is routed.

Exercise 3
Completing the Topology
Design Validation

Design rule checks exist to enable users to validate their designs in terms of connectivity and consistency with checks for things such as mismatched part numbers and unplaced devices.

It is a good idea to run the design rule checks at regular intervals throughout the design process to enable users to pick up potential problems at an early stage.

The design assistant gives users the option to run a default subset of checks as well as additional design rule checks that users can manually activate within the design assistant.

To open design assistant:
⇒ Click View/design assistant

To select required checks:
⇒ Click the configure checks icon
⇒ Check the boxes that relate to the check you wish to instantiate
⇒ Click ok
⇒ View the results in the output window

The output window has a message area to highlight potential problems on specific objects on the design and hyperlinks to easily zoom to the objects.

Design rule checks can be run independently of any other activity, on an intermittent basis or as part of releasing a design. The severity and frequency of the checks is set in Capital Project under preferences/Capital Topology/Checks.

Examples of Design rule checks

Exercise 4:
Run the Drcs as described in the worksheet
Visualization of Wiring

Once the wires have been successfully routed they will appear as 'Unassigned' signals on the Plane browser. The user can also expand the harness nodes and drive down into the individual bundles to see which wires have been routed through which bundles.

Rules and Constraints

Capital Topology provides the ability to automate device placement and wire routing. Rules and Constraints are used to control how this automation works. A Rule is defined as a collection of Constraints. Placement constraints were covered briefly earlier; this section will concentrate on routing constraints.

In a common scenario where a wire can take multiple routes through a harness/topology how does the tool decide which route the wire should take? By default it will choose the shortest route between 2 points. However this can be controlled at multiple levels using constraints.

Pressing the button on the plane browser toolbar will display the rules/constraints that are applied to the entire design. These rules affect every object within the design but can be overridden at various levels.

The following example explains how the ‘Route by’ constraint is implemented on a design.

Typically the ‘Route by’ constraint would be applied to a harness rather than a plane. Adding the rule to the harness will affect all wires that pass through that harness, where necessary this can be overridden at the bundle level.

In the following case any wires coming through the inline will have a choice of 2 possible routes through the harness, one for primary systems, and one for secondary systems.
Applying the ‘Route by’ constraint to the top and bottom bundles gives the user the control to route different wires through each bundle depending on a condition, that condition might be the result of a property set on the wire, or an option expression.

To assign wires to different bundles along a harness:

⇒ Select the lower bundle and open the properties dialog
⇒ Click the Rules tab
⇒ Add a Constraint
⇒ Select the ‘Route by Attribute/Property’ constraint
⇒ Change the ‘Name’ field to a property (System Type, for example)
⇒ Change the value to ‘Primary’
⇒ Repeat for the upper bundle, changing the value to ‘Secondary’

The result when wires with those properties are routed through this harness should be as follows:

Assigning Signals

In order to help the user visualize the path of a signal through inlines and bundles it is possible to associate wires to signals. Once associated, these signals will appear in the Plane browser under the ‘Signals’ node and disappear from the ‘Unassigned’ node.

The user can choose to assign wires to signals in 2 ways:

- By attribute, such as the name
- By any user defined property

Any wires that have a matching attribute/property will be associated according to the value of that attribute/property. This allows the user to use whatever mechanism they want in the wiring designs (including tracing etc) to assign a signal property and then use this directly in the Topology tool.

⇒ Press the button on the toolbar of the Plane browser to assign wires to signals
⇒ Choose from either a pre-defined attribute or a user-defined property to associate the wires
⇒ Press OK to see the results of the assignment directly in the Plane browser.

If the signals are assigned using property all wires with the same property will be listed under the same signal and will have the same wire name.

**Splice visualization**
To view the location of splices on the topology users should click the toggle splice visibility button at the base of the graphics window.

**Wire Navigation and Reporting**
Selecting any signal or wire will highlight the path that wire takes through the topology. In the case where a signal contains multiple wires this could mean passing through multiple bundles or harnesses highlighting the full path of that signal through the platform.

Right clicking and choosing ‘Show Physical Source’ or pressing ‘x’ on any wire or signal will display the wiring design containing that wire and will highlight it on that diagram.
Report Generation

A report can be generated on any bundle that will indicate the contents of that bundle and give some information about the wires it contains.

⇒ Go to the Reports menu and select Bundle Wires
⇒ Select a bundle on the diagram
⇒ A report will be generated and will appear in the output window

Back Annotation of Wires

Once a topology has been finalized and all of the wires routed correctly Capital Topology allows the user to send wire length information and harness attribute information defined in the topology back to the original wiring designs.

⇒ Open the Actions menu
⇒ Select Back Annotate Wires…
⇒ Open a wiring diagram
⇒ Check that the length and harness attributes have been updated correctly

Signal maps

Once signals and devices have been associated to the topology diagram the slots will have a signal map that indicates the routing of the signals into the connector cavities. This is a useful dialog to view the content of the slot and the mapping of signal information through connectors into the devices within the slot.

To view the signal map:
⇒ Right click the slot and select signal map

The slot content is displayed in the upper half of the signal map dialog (devices, connectors, holes). The lower half of the signal map dialog displays information relating to the signals that are routed into the slot, this includes information such as the harness the signal belongs to, the connector and device that the signal connects to.
Wire Routing Assistant

When wires are un-routed or require re-routing, users can implement the routing assistant. This highlights all potential routes available for the wire and users can select the desired route from the routing table.

If the wire is currently unrouted:

⇒ Open the routing assistant
⇒ Highlight the unrouted wire and drag to the routing assistant window
⇒ Select the desired route
⇒ In the to do list highlight the wire and select the hyperlink to the wire diagram for the wire
⇒ Insert an inline connector on the wire diagram
⇒ Rename the connector to match the name defined in the routing assistant
⇒ The route and wire in the to do list will change status to a green check mark

If a design change is required to accommodate changes to harness definition or to integrate a new system, changes can be made to wires that are already routed:

⇒ Open the routing assistant
⇒ Highlight the routed wire and drag to the routing assistant window
⇒ View the current route for the wire
⇒ Add an inline connector to allow for the new harness definition or integration of a new system
⇒ In the to do list the wire will now show 2 unrouted ends
⇒ Click the first unrouted end in the ‘wires to be routed’ column and select the hyperlink to the wire diagram
⇒ Insert an inline connector on the wire diagram
⇒ Rename the appropriate side of the inline connector on the wire diagram to match the name defined in the routing assistant
⇒ Repeat the steps to route the second wire end to the appropriate side of the inline connector in the wire diagram
⇒ In the topology diagram refresh the wiring

Exercise 5
Complete the exercise described in the worksheet
Harness Synchronization

Topology designs may be transferred to the Harness XC tool using a synchronization process. The synchronization process takes the wire and connector information defined in the Topology plane into the Harness tool.

Poke Home Wires

Where wires are to be assigned to a harness but not terminated within that harness, a poke home attribute must be assigned to the associated connectors. Users assign a poke home attribute to Topology connectors where appropriate. The synchronization is performed and the system identifies which harness connectors have a poke home attribute assigned. These poke home connectors are placed on the harness diagram along with their associated wires, but the system unchecks the ‘included on BOM’ attribute for those connectors.

To assign a poke home attribute to Topology connectors:

⇒ Select the connector to which you wish to assign a poke home attribute
⇒ Right click the connector and select Properties
⇒ Click the routes tab
⇒ Check the poke home box under the poke home column
⇒ Click OK

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 to the XC tool and is not visible within the topology plane. If poke home information is required, this may also be added along with the route code.

To assign a route code to a connector:

⇒ Right click the connector and select properties
⇒ Select the Routes tab
⇒ Highlight the relevant harness row and insert a route into the route code cell
⇒ Click OK

In order to perform a synchronization, a number of steps must be followed:

⇒ Assign a poke home and/or route codes to relevant connectors
⇒ Create a harness build list containing the harness design to be synchronized to (this is done in Harness XC)
⇒ Create a change policy (this is done in Capital Project)

Typically the Topology designers will only need to assign a poke home attribute and route codes to the connectors where necessary, all other aspects of the synchronization will be covered by the Harness engineers. The steps above are covered in detail in the Harness XC training course; here we will discuss how to assign a poke home attribute to connectors in the Topology design.

Topology Build lists

A number of topology designs may exist within one project. The designs may be linked by inline connectors. A topology build list can be created to keep a track of recent revisions of topology designs. Multiple topology designs within a build list may link together via inline or slot connectors. Using mating half information users can navigate between the designs in a build list to view linked designs.
To create a topology build list:
⇒ Open the build lists folder and right click the topology build list
⇒ Select New
⇒ Define a name for the build list and add all relevant topology designs to the build list

**Exercise 6:**
Complete the exercise described in the exercise worksheet

---

**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/Topology 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 (Topology)
⇒ 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
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:

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 have the quick code displayed.

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).
Save a Design

Saving commits the changes made to the design in this session to the database.

To save, execute **File / Save** or toolbar **Save** icon.

Close a Design

This is accomplished by clicking the window close icon.

If the design was modified while it was opened and has not been saved, the user will be prompted to save.

**Exercise 7**

Refer to the exercise worksheets to create an ECO and perform a design change
Data Bridging

Using Capital Harness Bridges it is possible to import data from various external sources such as CATIA v5, Catia v4, Catia v6, Ideas, NX4. The import file is in XML format (.xml extension).

Bridging data out

Topology data can be exported out to other data sources via the bridge facility. A number of file types exist to export the harness information out so that it is recognized in the receiving tool.

⇒ Select the harness to be exported (this might be a single or multiple harnesses)
⇒ Define the file type that is to be exported (dsi, xml, kbl etc)
⇒ Define the location of the resulting file (this will be a zip file if multiple harnesses are selected)
⇒ Click Export

Note on loops in bundles:

Where a loop is introduced in a bundle a through node will be displayed which is rendered as a reference node. Upon bridging out to Catia, this reference node enables users to match integrator bundles to those in Catia, and the reference nodes will be displayed along the relevant bundles.

Change Manager

The Change Manager resource is used to import data into the Topology diagram. The change manager gives users a visible representation of the information that is to be imported. Upon import users can opt to automatically link information to the Topology plane, manually link bundles from the incoming file to the pre-existing topology or automatically place bundles on a Topology diagram.

Data that is imported must be in an XML format.

⇒ 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.

⇒ **Autolink**: Will link objects in the incoming file to the source harness using criteria specified in the bridge options.

⇒ **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.

It is possible to view the data as shown above or in the preview pane by selecting the Display button.

---

**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.

---

**Placing the Branch Configuration**

Place the layout into Topology by selecting the objects to be placed and selecting **Process**.

When a harness model is placed all the objects are linked. A linked object stores a reference to the CATIA 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 CATIA object properties to the object it is linked with.

---

**Note**

Once the Topology has been placed, any changes to the view (flip, rotate etc) will not alter the diagram. After placement, if the view is changed (flipped etc) and a new bundle is added, this new bundle will be positioned as if the Topology diagram has been flipped etc.

Due to changes in the Topology diagram or the CATIA model, some objects may be unlinked i.e. have no link to the data within Capital Topology. 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

![Query Editor](image)

To find new CATIA bundles, select MCAD **Unlinked** and the system will display only new CATIA 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).

---

Topology Styling

Each Topology plane can have its own style set applied. A style set controls the way in which the diagram looks. Many aspects of a diagram can be styled, from the type and color of font to the placement of part numbers, display of borders and additional tables.

**To view a style set and its content:**

⇒ Right click in a space on the topology plane  
⇒ Select Style/Edit Style

The currently applied style set will open up. Users can modify the appearance of objects on the plane by modifying their respective instances in the style set.

**To change a style set on a topology plane:**

⇒ Right click the plane name/Edit/change the style set in the style set field

**To modify the style in a topology design:**

⇒ Open the style set  
⇒ Select the object to be modified (we will take the slot in this example)  
⇒ Open up the decorations sub folder  
⇒ Open Attributes  
⇒ Click Name  
⇒ Make the required changes
Exercise 8

You will now change the font and color of the Name attribute text

⇒ With the Name attribute highlighted in the default topology style
   set click in the Font Name field
⇒ Select Arial
⇒ Click the Color field
⇒ Select Orange or make your own choice
⇒ Change the fit and wrapping to ‘shrink to fit’
⇒ Click Apply

When an object type is to be excluded from the apply style function users can check the ‘exclude from apply style’ check box. This will ensure that each time the apply style is set against the topological plane, that object will remain unchanged. This is particularly useful if manual modifications have been made to the diagram.

More advanced styling features are available but will not be covered during this course.