Pro/ENGINEER ROUTED SYSTEMS DESIGNER

RSD R5 Basic Tutorial

Version –1.0
September 2003

Introduction

This is a basic tutorial is designed to expose the user to the most common functionality within Routed Systems Designer 5.0.
If you have not upgraded from Routed Systems Designer Lite or have not purchased the full version please do not continue with this tutorial. ***RSD 5 Basic Tutorial – Lite.pdf*** is the recommended tutorial to learn about the capabilities of Routed Systems Designer Lite

Please note to become fully proficient with Routed Systems Designer please contact you local account representative for details of an RSD training course.

Conventions

Select – Click with left hand mouse button
LMB – Left Hand Mouse Button
MMB – Middle Mouse Button
RMB – Right Hand Mouse Button
CSV – Comma Separated Variable File
* > * – Task 1 > Task 2 > etc > etc

PRO/ENGINEER ROUTED SYSTEMS DESIGNER

RSD R5 BASIC TUTORIAL

INTRODUCTION

About Routed Systems Designer
- Diagram type separation
- User interface

VIEW CONTROLS

About View Controls

CREATING A NEW DESIGN

SETTING UP THE WORKING ENVIRONMENT

XP Style UI

Cursor options

Turning off RSD Lite

Background Color

Highlight Color

Editing the color pallet

Auto-save

Message Window

Directory Defaults

To improve the usability of saving and retrieving files there is the ability to set the directory defaults.

SETTING UP STANDARD PROPERTIES FOR A DRAWING SHEET

Creating Subtypes

Using properties with catalog templates

Creating a new sheet

Using CSV files

CIRCUIT DIAGRAMS

Creation a new sheet

Placing Shapes
About shortcut options

About Signal Networks
To Instance Signals
To manipulate Signals

About Context-sensitive cursors
Create
Edit

Routing Signals

Branching - Set-up

Branching – Routing Signals

Group Shapes

LOCATION SETS

About Location Sets

To Create or Edit Location Sets
Creating a new location Set
Setting label visibility
Setting the label function
Setting sequential sheet numbering

WIRE AND CORE ASSIGNMENT EDITOR - WIRING DIAGRAMS

Auto wiring

TRAIL FILES

About Trail Files
- To Record a Trail File
- To Play a Trail File
- To Automatically Run a Trail File When You Start RSD
- To Control the Display During Trail File Playback
- To Compare Trail Files

CUSTOMIZING THE USER INTERFACE

ABOUT CUSTOMIZING THE USER INTERFACE

Creating Button

MAPKEYS

About Mapkeys
- To Create a Mapkey
- To Delete a Mapkey

Creating a Mapkey

UPDATING PROTOTYPE INSTANCES

ABOUT UPDATING VARIABLE GROUPSHAPES

Changing the Circuit Diagram

APPLYING DATA SETS

ABOUT ADDING, REMOVING, AND REORDERING MEMBERS IN VARIABLE GROUPS
- To Add a member
- To remove a member

CREATING A CABLE DEFINITION

IMPORTING XML DATA INTO PRO/CABLING WILDFIRE

GLOSSARY OF TERMS

Introduction

About Routed Systems Designer

Routed Systems Designer helps designers and engineers to quickly and easily create complex interconnected routed systems diagrams as an integrated part of the total product development process. Routed Systems Designer allows the creation of the following types of diagrams:

- Wiring
- Piping
- Process and instrumentation design (P&ID)
- Block
- Functional
- Schematic

Routed Systems Designer is a standalone package. However, it is an integrated part of Pro/ENGINEER and allows the reuse of diagrams to drive 3D cabling and piping routing using the Pro/ENGINEER Piping Design and Cabling Design Options. It allows multiple designers to work at the same time, while assuring the consistency of interconnection data and the corresponding 3D routing information.

Routed Systems Designer uses an object-oriented display. You can select any object in the work area and work on it through a context-sensitive command menu. You can search and Select within the database by object type or associated properties.

Diagram type separation

All design shapes and the sheets themselves have a diagram type property **Set** to one of the diagram types.

By default, the catalog explorer shows shapes of the current diagram type of the sheet. However, you can **set** the catalog explorer to show shapes of other diagram types. Routed Systems Designer does not stop you from placing a shape of a different diagram type if you choose one from the catalog explorer.

User interface

Use the tool trays on the left of the screen to create, Add, and edit design elements. The list box contains all the tool tray types, including object and line creation for different design types, the object catalogs, and special Selection tools. The buttons above the list update to show the most recently accessed tool trays.

Use the icons under the main menu to access viewport controls, for example, pan, zoom, and grid display.

Click the main menu commands to show subcommands covering setup and display options, macro recording, layer control, and so on.

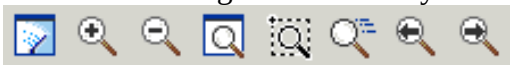
All the pull down menus can also be selected using the keyboard. Press the ALT key and then the letter that has been underlined in the menu name. For example, press ALT, E, A to Open the Appearance dialog box.

View Controls

About View Controls

You often need to magnify or reduce an area of a sheet when drawing complicated areas.
Note: The dimensions of any element drawn on the sheet do not change when you use view controls. They only enable you to view the elements at different magnifications.

Use the following tools to modify the view as required.



In order from left to right, these tools allow you to:

- Refresh graphics
- Zoom in
- Zoom out
- View the full sheet
- Magnify a rectangular area of the sheet
- Adjust the view (dynamic pan and zoom). Left-Click to zoom or middle-Click to pan.
  Alternatively, you can also hold down CTRL and the middle mouse button, move the pointer upward to zoom in and downward to zoom out & hold down SHIFT and the middle mouse button, move the pointer in any direction to center the view on a selected .
- Open the previous window (as defined by magnification).
- Open the next window.

The following additional options are also available from the View menu.

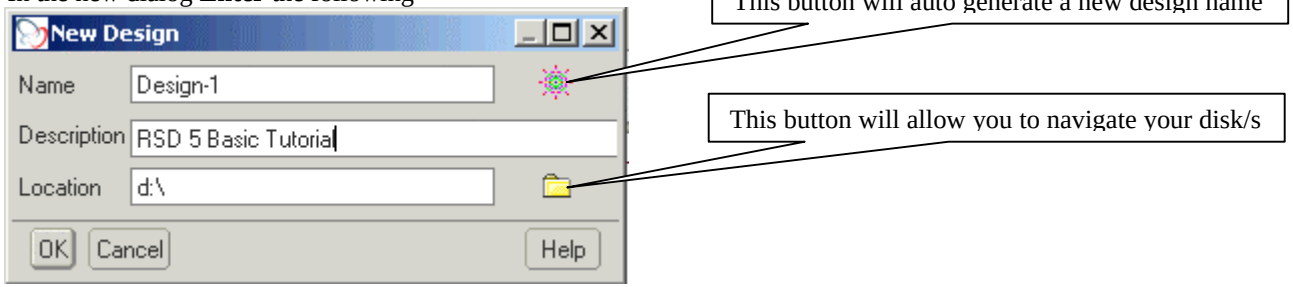
- **Pan** **Centers** the view around the probed point.
- **Pan Fiber** Navigates to alternate ends of a Selected fiber (CTRL – E only in edit mode)
- **Zoom top left** Zooms the top left quadrant of the sheet.
- **Zoom top right** Zooms the top right quadrant of the sheet.
- **Zoom bottom left** Zooms the bottom left quadrant of the sheet.
- **Zoom bottom right** Zooms the bottom right quadrant of the sheet.
- **Zoom maximum** Displays the maximum sheet area, as specified in the Sheet Properties dialog box.
- **Zoom Factors** Sets the zoom factor
- **Zoom half** Zooms the view to half size.
- **Zoom double** Zooms the view to double the current size.
- **Shape Zoom** Zoom to Selected shape or shapes
- **Zoom to scale** Zooms the view about the probed point by a **Set** scale. The scale is specified in the Zoom Factors dialog box.

Creating a New Design

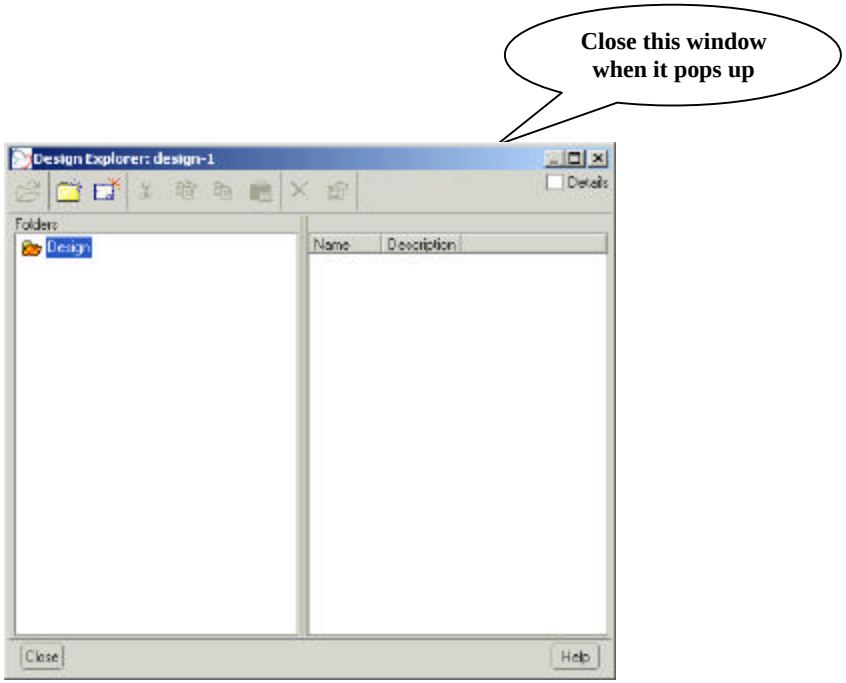
The first thing we need to do is create a new design.

1. **Select**  to create a new design

2. In the new dialog **Enter** the following



3. **Close** the next window



Setting up the working environment

Before we commence any schematic work lets set-up our working environment, this is one time operation.

XP Style UI

With the windows release, there is the option to have a windows XP look and feel.

4. **Select** Options > Switches
5. **Select** XP Style windows
6. **Select** OK from the message prompt
7. Do not **Close** the Switches Control Panel dialog

Do not re-start RSD just yet

Cursor options

There is the option to have either the cross hairs turned either on or off.

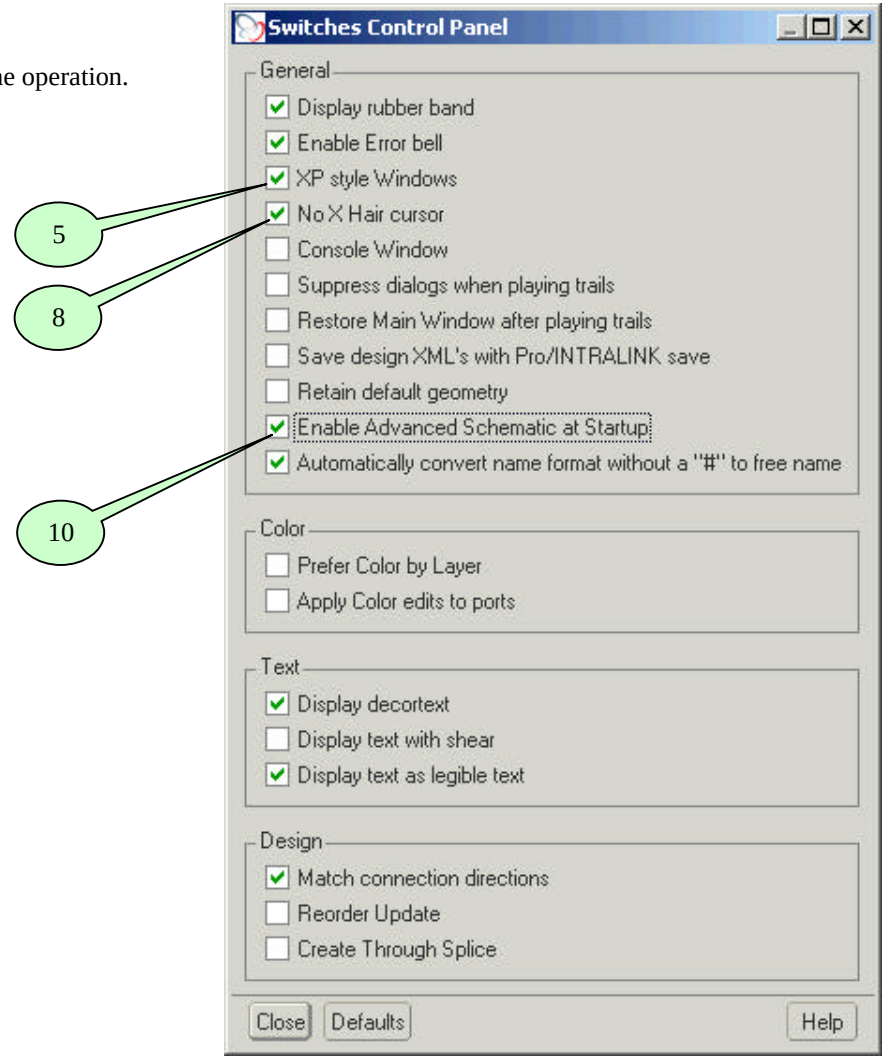
8. To turn off the cross hairs (current default) **Select** No X Hair
9. **Select** OK from the message prompt

Do not re-start RSD just yet

Turning off RSD Lite

There is an option to use the full version of RSD (providing a valid license is available)

10. To use the full version of RSD **Select** Enable Advanced Schematic at Start-up
11. From the Switches Control Panel dialog **Select** Close.



Background Color

Lets change the background color to white

12. **Select** Options > Background color
13. **Select** White from the color pallet
14. From the color pallet dialog **Select** OK

Highlight Color

The highlight color may also be changed

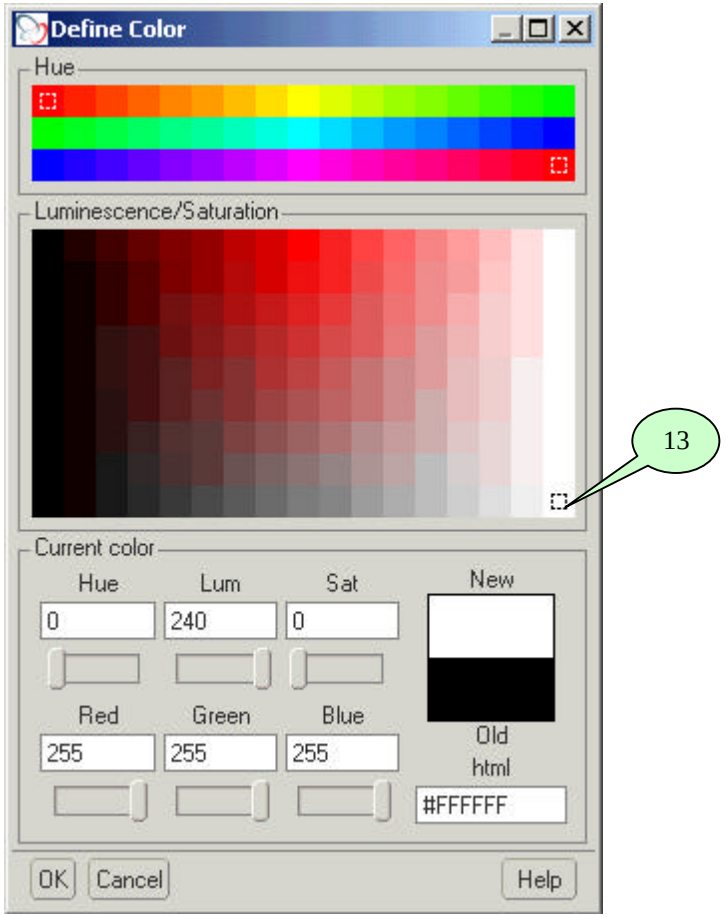
15. **Select** Options > Highlight color > **Select** a desired color
16. From the color pallet dialog **Select** OK

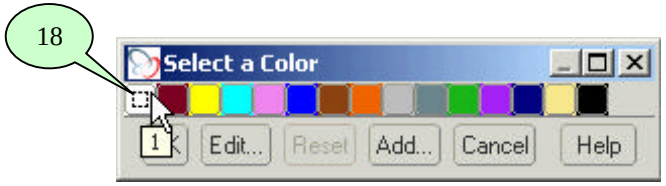
Editing the color pallet

The default color setting for lines and text is white. We have changed the background color to white thus we will not be able to see any of our work. To change the default line and text color we need to edit the color pallet

17. **Select** Options > Edit Color Pallet
18. **Select** color No. 1 > **Select** Edit
19. **Select** the color black
20. From the **Select** a color dialog **Select** OK
21. From the edit color pallet dialog **Select** OK

You will now see the text ***No Sheet Open*** in the center of your screen

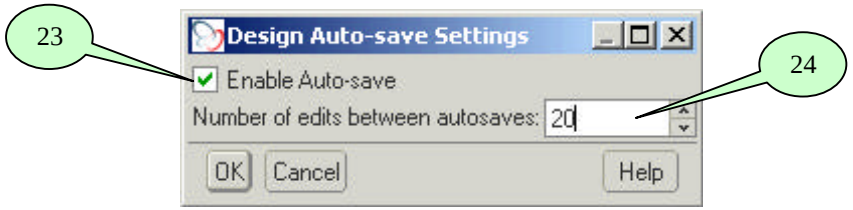




Auto-save

To ensure that we do not loose any work lets ensure that auto-save is enabled

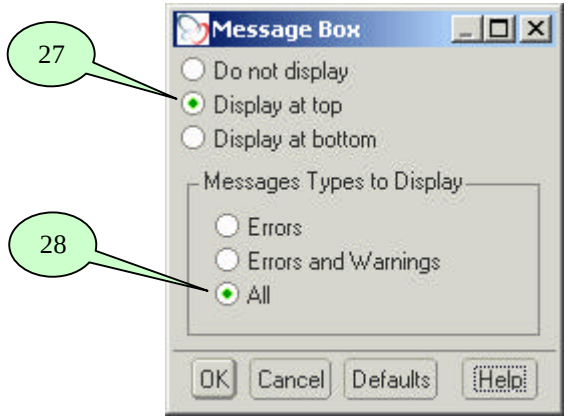
22. **Select** Options > Design Auto-save
23. Ensure enable Auto-save has a tick
24. **Set** the number of edits between auto-saves to be 20
25. From the auto-save dialog **Select** OK



Message Window

To obtain the maximum feedback from RSD lets modify the message box properties

26. **Select** Options > Message Box Properties
27. **Select** Display at top (the message box will appear above the graphics window)
28. **Select** All (display all the messages)
29. From the Message box dialog **Select** OK

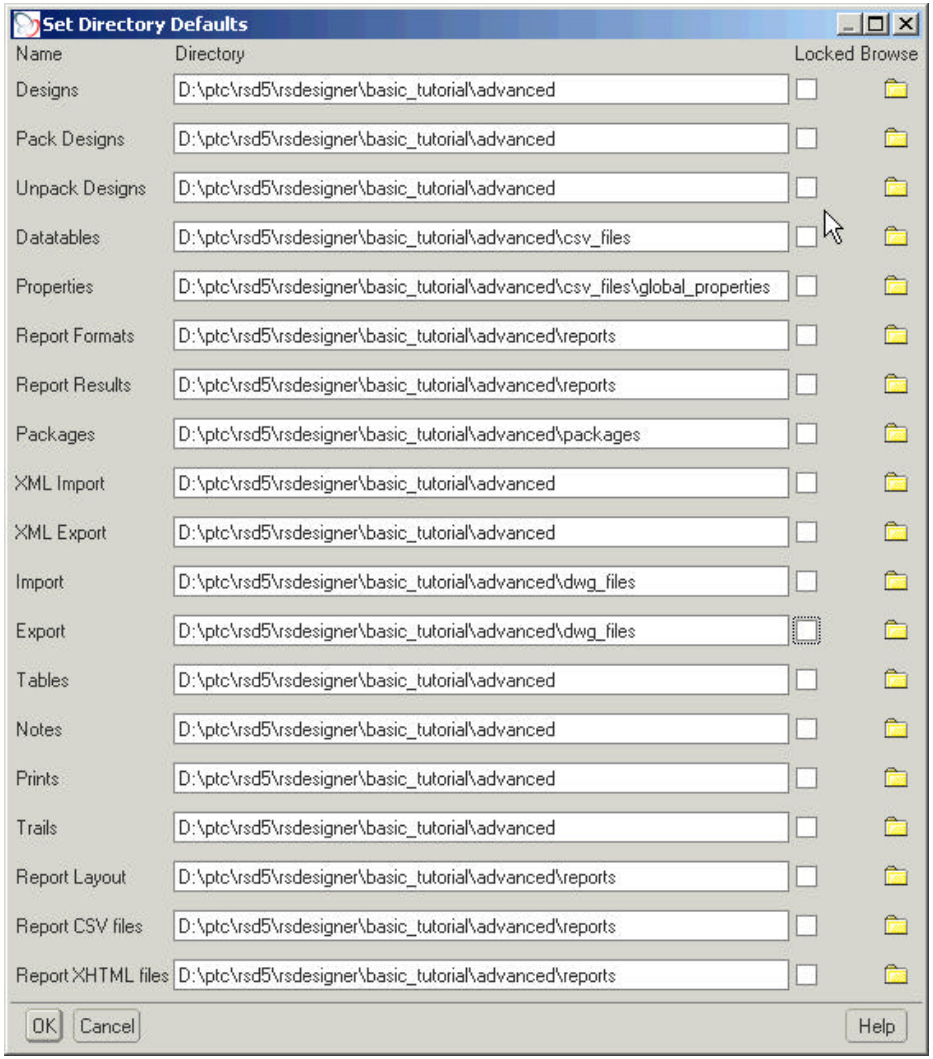


Directory Defaults

To improve the usability of saving and retrieving files there is the ability to set the directory defaults.

30. **Select** Options > Directory Defaults

31. **Set** the working directories to where your files reside. In this example the all the files reside under D:\ptc\rsd5\rsdesigner\basic_tutorial\advanced\



About Probe Specifiers

To the right of the graphics window are the probe specifiers. The system default is Auto Probe how this may be changed by placing the cursor over the desired probe specifier & and Selecting the RMB. To Set a different probe specifier for one mouse action place the cursor over the desired probe specifier & and Select the LMB

Auto

The Auto probe specifier  creates a point by automatically selecting one of the six probes.
The order in which the probe is attempted is:

- Near
- Intersection
- Perpendicular
- Tangent
- Segment
- Free

If there is an existing point within the radius of the cursor, a Near probe is given. If there are no points within the radius, the next type of probe is attempted. If the first five probe types cannot be used then a Free probe is Selected.
As soon as a probe is made, a message appears in the startup window notifying you which type of probe specifier has been given.
The Auto probe specifier usually supplies the probe specifier you need. However, you should note that it is most likely to produce the wrong result when the cursor is in an area of dense lines. You can ensure that the correct type of probe is used by windowing in on the area.
When you start up ROUTED SYSTEMS DESIGNER, the Auto probe specifier is set as the default. Therefore, you do not have to choose the Auto probe specifier from the toolbox. You can position the cursor where you want the point to be placed and Click SELECT. If you wish to change the default, place the cursor over the desired probe specifier, Click Right Hand Mouse Button, & Select Set as Default.

Free

The **Free** probe specifier allows you to create a point at any location on the sheet irrespective of any previously created point.
A **Free** probe specifier creates a point at the current cursor position unless the grid is active. When the grid is active ROUTED SYSTEMS DESIGNER creates a point at the **closest** grid intersection. The coordinates of the point are then taken as the coordinates of the grid line intersection.

Grid Probe Specifier

The Grid probe specifier  lets you create a point that snaps to the nearest grid intersection even if the grid is inactive.

Intersection

The Intersection probe specifier  allows you to create a point at the intersection of two existing lines.

Near Probe Specifier

The **Near** probe specifier  enables you to locate the nearest existing point and to create a new point at that location. This means that a **Near** probe specifier enables you to position a point at exactly the same coordinates as a point in an existing line or at the Datum Point of a text element.

Segment

The Segment probe specifier  allows you to create a point on an existing line so that the new point lies as **Close** to the cursor as possible.

Perpendicular

The Perpendicular probe specifier  allows you to create a perpendicular line between the current point and the line nearest the cursor.
You can use this probe specifier once you have started to create or edit a line.

Center

The Center probe specifier  locates the center point of the nearest curved line segment.
For circles or arcs, the probe locates the defined center point. For other types of arcs a center point is inferred as the intersection of the two perpendiculars from the tangents at the ends of the arc, see below No position is located if the perpendiculars do not intersect inside the arc.

Mid Segment

The Mid Segment probe specifier locates the middle of the straight or curved line segment nearest to the position that you probe.

Tangent

The Tangent probe specifier  allows you to create a point on the nearest arc or circle such that the line segment joining the new point to the previous point is tangential to the arc or circle.
You can only use this probe specifier once you have started to create or edit a line.

Last

The Last probe specifier locates the most recent current point. This enables you to create a number of lines that pass through the same point without having to define that point for each line.

Locating the Datum of an Element

The **Datum** probe specifier  allows you to use the datum point of the Selected element, for example to place a point in a line. To do this:
1. Select a tool, for example a create line tool.
2. Choose the datum probe specifier.
3. Point to the element and Click SELECT.

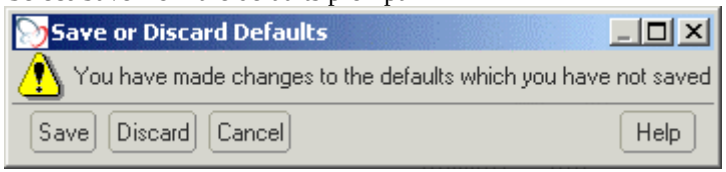
For this tutorial we will be working with a grid, therefore using the RMB we can set the snap default to be to Grid

32. Place the cursor over 
33. Click RMB
34. With the LMB **Select** Set As Default

Saving Default Settings

To enable the options we have just as well as save them as default we must now re-start RSD.

35. **Select** Save 
36. **Select** File > Exit
37. **Select** Save from the defaults prompt



38. Re-start RSD and **Open** Design-1

Importing a Package

Currently we have a new design that does not have any catalog items. We are going to use the packaging functionality to import a catalog.

About the Export Package Manager

About Exporting Parts of a Design

You can export the following parts of a design in Routed Systems Designer to another design. Note that this functionality is not available in Routed Systems Designer Lite.

- Catalog objects
- Datatables and maps
- Design sheets
- Design snippets
- Shapeless artifacts

To export these parts, create an export package that stores the exported objects. Other designs can interpret the contents of the package and import the objects.
Use the Export Package Manager to create the export package.

To Export Parts of a Design

- Click File > Export > Package. The Export Package Manager dialog box opens.
- To add an object to the export package, in the left panel, select the object to be exported and click >>.

The selected objects are added to the export package and are displayed in the top right panel.

- To add the complete contents of the design and catalog to the export package, click >>>>.
- To delete an object from the export package, in the top right panel, select the object to be deleted and click <<.
- To remove all items from the export package, click <<<<.
- To store all the information about the export package in a file, specify the location and the filename in the Export to file box. Alternatively, click Browse to browse to the file location.

The default extension of a package file is .pkg. To save the structure of the export package to a file with a .lst extension, click Contents > Save, specify a file name in the File Select dialog box, and click OK.

- By default, all the dependencies of an object are exported. For example, if a design sheet is exported, all the objects on that sheet have prototypes. By default, the prototype objects are automatically included in the export package. This applies to datatables and maps too. If you do not want to export all the dependencies, click Options > No dependents.
- Click Export.

About the Import Package Manager

About Importing Parts from a Design

You can import the following parts of a design into an existing design:

- Catalog objects
- Datatables and maps
- Design sheets
- Design snippets
- Shapeless artifacts

To import these parts, you need an export package file that stores all the exported objects. Use the Import Package Manager to import parts from the export package file.
An unique identifier of an object, SYS_ID, allows Routed Systems Designer to identify objects that have already been imported and exported and report this information. If an object already exists when importing, a difference and merge operation is performed between the two sets of data, resulting in one of the following situations:

- The original object is retained and the new one is discarded.
- The original object is discarded and the new one is used.
- The properties of the original and imported object are merged.
- While importing:
- If two designs are created separately using the same imported catalog, they can be merged later and the catalogs are not duplicated.
- If a subsystem is imported into two or more designs, and the designs are subsequently merged, the artifact representing the subsystem can be identified as having two representations.
- If two designs are merged and names clash (two objects with the same name, one originally existing, the other being imported), Routed Systems Designer automatically reassigns names based on the sequence numbers, or informs you of such a clash in names.

To Import Parts from a Design

- Click File > Import > Package. The Import Package Manager dialog box opens.
- In the Import from file box, specify the location and filename of the package file from which you want to export objects. Alternatively, click Browse to browse to the file location. The default extension of a package file is .pkg.
- To import an object, select the object in the left panel and click >>. The selected objects are displayed in the right panel.
- To import all objects from the package, click >>>>.
- To delete an object from the import list, select the object from the right panel and click <<.
- To delete all objects from the import list, click <<<<.
- If you do not want dependent objects to be automatically imported from the import package, click Options > No dependents.
- By default, objects in the import package or the current design are automatically imported and new objects are merged. If you do not want the new objects to be merged, click Options > No merge. The new objects appear as conflicts in the Import package comparator dialog box.
- Click Import.

If the objects being imported already exist, the Import package comparator dialog opens. Routed Systems Designer automatically merges the objects if possible. If not, it displays the differences and allows you to select the version of the object to be used. The differences are displayed in a tree format, showing the original design structure versus the structure after the import. The differences and similarities are indicated differently allowing you to identify what to keep and what to discard.

Click Next to move to the next unresolved difference. Accept the original value (or object) or the value from the imported package. If the properties of an object are listed as UNSET, the object was not present in the original design or in the import package, depending on which side of the list they appear on.

When all differences are resolved, Import on the Import Package Manager dialog box is enabled.

Note: The Import package comparator dialog box may open several times, depending on how many sheets are being imported and whether the sheets have differences.

.

1. **Select**  from the main tool bar to Open the Catalog Explorer

Notice that the catalog explorer is currently empty.

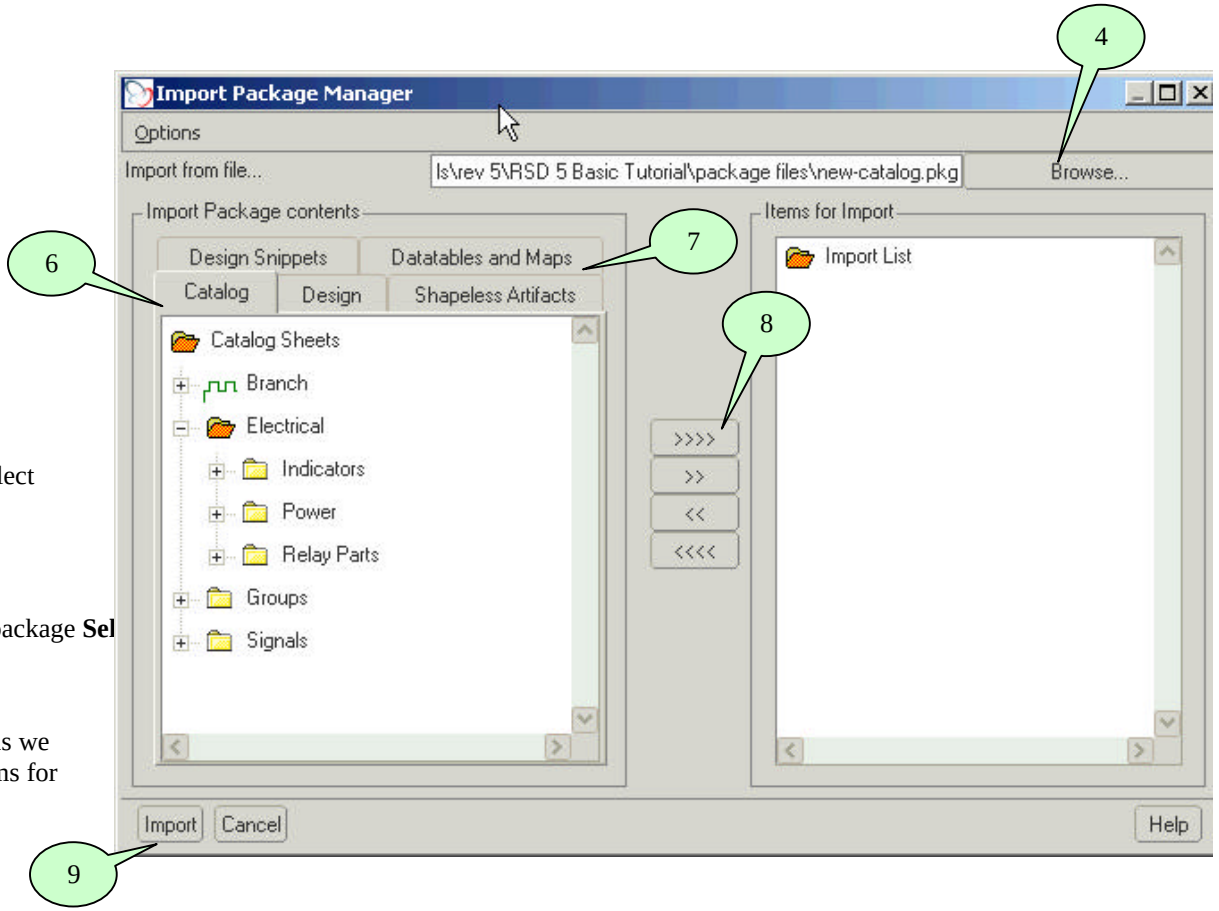
2. **Close** the catalog explorer
3. **Select** File > Import > Package.
4. **Browse** to the package directory and **Select** new-catalog.pkg.
5. **Select** OK from the file selector dialog.
6. **Select** the Catalog tab.

The catalog tab displays a folder structure; it is from here that you may select individual items that you wish to import or a complete folder structure.

7. **Select** Datatable and Maps Tab.
8. We also have some datatables. To ensure that we import all of the package **Sel**  .

The import area on the right hand side has now populates with all the items we wish to import. Drag and drop of items between package contents and items for import is also available.

9. Select Import.

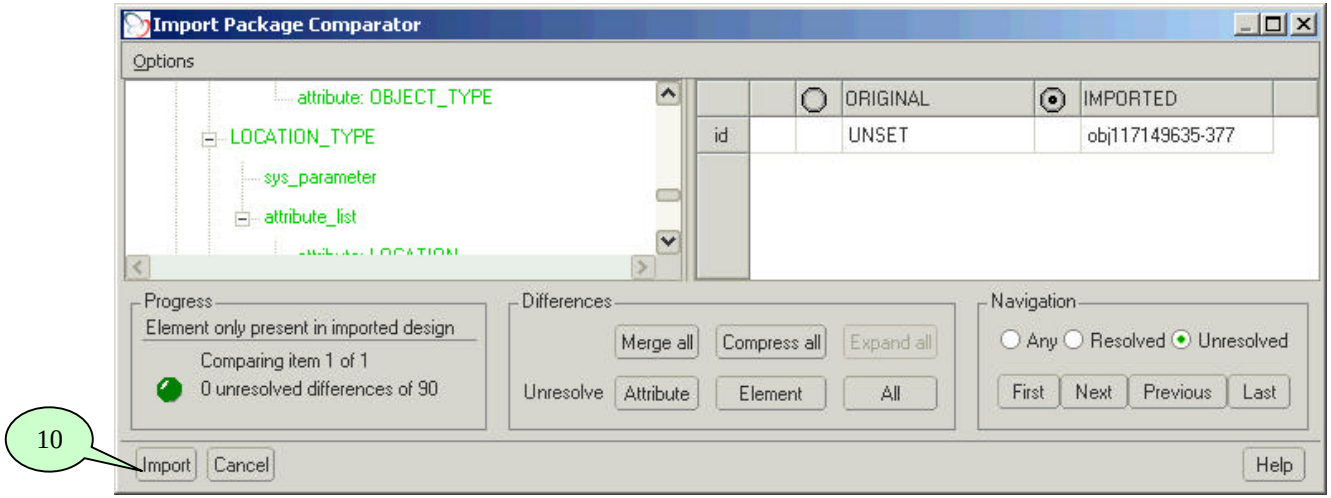


The import package comparator will now open. This is telling you that there are number of differences between the base design (the design you have created) and the imported data. We would expect there to be a number of differences between the import data as the base design is empty.

10. **Select** import. The dialog will disappear once the import is complete and a message will indicate that the transaction is complete.

11. **Select**  to Open the Catalog Explorer.

Notice that we now have some catalog items.



Global Properties

We now want to set-up Routed Systems Designer so that when a new design sheet is created it contains information pertinent to the project and the designer. To do this Routed Systems Designer uses properties.

About Properties

About Object Types

An object type is a hierarchical property used to identify the family to which an object belongs. The predefined types are: component, connection, dataset, document, spool, and port. You can define new subtypes of the predefined types. A set of default properties and values are associated with each type.

Blocks, groups, fibers, ports, sheets, and datasets have types. The type of a block must be a subtype of the component type. The type of a fiber must be a subtype of the connection type. The type of a group must be a subtype of either the component type or the connection type. The type of a port must be a subtype of the port type. The type of a sheet must be a subtype of the document type. The type of a dataset must be a descendent of either the dataset type or the spool type

To Set an Object Type

- Select the object. You can select a block, group, fiber, port, or a sheet.
- Right-click and select Properties from the shortcut menu.
- Click Set beside the Type box. The Select type dialog box opens.
- Select the type and click OK.

You can also set the type of an object in the Model Explorer.

To Define an Object Type

- In the Diagramming tool tray, click  . The Define types dialog box opens.
- Click  in the left panel to create a new subtype of the currently selected type. Specify a name for the new type and click OK.

The new type has a copy of the properties of the parent type. The top right panel, Type Properties, shows the properties of the current type.

- Click OK. The new type is added to the global list. The top right panel, Type Properties, shows the properties of the current type.
- Add properties to the type, as required.

To Rename an Object Type

- In the Diagramming tool tray, click  . The Define types dialog box opens.
- Select the type to be renamed.
- Click  . The Rename type dialog box opens.
- Specify the new name and click OK.

Note: You cannot rename predefined types.

To Delete an Object Type

- In the Diagramming tool tray, click  . The Define types dialog box opens.
- Select the type to be deleted.
- Click  .

Note: You cannot delete predefined types.

About Object Properties

Blocks, fibers, groups, and ports can have a user-defined set of properties and values associated with them, at both the definition and the instance level. When you instance a catalog definition, the instance inherits the definition's properties. You can change the value of properties on the instance, if necessary.
Sheets and datasets can also have a user-defined set of properties and values.
Before you can assign a property to an object, you must first create the global definition of the property for use in the design. Any property can be added to any type.

Click  in the Diagramming tool tray to view or edit global properties. The Define types dialog box opens.
The left panel displays existing types in the design. You can add properties to these types using the options in the right panel.

Options in the top right panel

The options in the top right panel of the Define types dialog box are:

- **New** Launches the Property definition dialog box to create a new property in the property pool.
- **Delete** Deletes the selected properties, in either the type properties or the property pool, from the design.
- **Modify** Launches the Property definition dialog box to modify the selected property.
- **Copy** Launches the Property definition dialog box to create a copy of the selected property**.**
- **Import** Imports a .csv property definition file.
- **Value options** Sets a default value for a type property.

Options in the lower right panel

The lower right panel, Property Pool, shows properties that are defined but not used in the current type. You can either Add properties to the current type or remove properties from the current type using the appropriate options.

- **Add Property** - Adds the Selected property to the current type in the property pool.
- **Remove Property** - Removes the Selected property from the current type and returns it to the property pool.

To Add Property Definitions

- the Diagramming tool tray, click  . The Define types dialog box opens.
- Click New in the right panel of the Define types dialog box. The Property definition dialog box opens.
- Specify the following property definition items in the Property definition dialog box:

  - **Name**—Identifies the property definition in the design. Examples include Nphases, MaxPowerRating, Color, and Standard.
  - **Value Type**—Integer, Real, String, or List. Examples are 3, 3.142, Red, and [Iso, DIN] respectively. A list can consist of a list of strings.
  - **Description**—Describes the property.
  - **Units**—Specifies the units of the property.
  - **Derived property**—Sets a property as derived. Use a function format that is similar to a label format to calculate such properties.
  - **Constrain to value list**—Constrains property definitions to a list of values.
  - For a **Value Type** of **String** or **List**, you can constrain the property to a Set of values specified in the Value List field.
  - For a **Value Type** of **Real** or **Integer**, you can constrain the property to a list of values specified in the Value List field and also a range of values specified in the **Value Range box.**

- Click OK. The new property is added to the global list.

To Add, Modify, or Delete Properties

After you have defined a property globally, you can add it to a selected block, fiber, group, or port, and set a value for it. The properties can be added when the object is created and added to the catalog, when the object is instanced or after it is instanced.

- Select the object. To select multiple instances of the same object type, hold down the SHIFT key and select the objects as required.
- Right-click and select Properties from the shortcut menu. The Properties dialog box opens. The current values are displayed in the respective boxes.
- To add a new property:
  - Click Add. The Property selector dialog box opens, listing all previously defined properties.
  - Select a property from the list to be added to the selected objects, and click OK or Apply. The property is added to the object's property list.
  - With the property still highlighted in the list, type a value for it in the Value box. Click  to confirm the value.
- To delete an existing property:
  - Select the property to be deleted.
  - Click Remove.
- To modify a property:
  - Select the property to be modified.
  - With the property still highlighted in the list, specify a value for it in the Value box. Click  to confirm the value.
- Click Apply or OK.

Depending on the type of object selected, you can also change the following object properties:

- Name Format— Specifies the format for the name of the object.
- Serial Number— Specifies the serial number to use when generating the name from the name format
- Global Name— Specifies that the name must be globally unique
- Autogen/Serial/Free Format/Unset— Specifies how the name of the object should be generated.
- Has Direction— —(for a fiber) Specifies that fiber has a direction, which means that internal connectivity of a fiber is one-way only. For example, consider a fiber going from A to B. If the fiber is without direction, A is connected to B through the fiber, and B is connected to A through the fiber. If the fiber has direction, then A is connected to B through the fiber, but B is not connected to A through the fiber.
- Internally Connected— —(for a block) Specifies that all the ports on selected block are connected together through the block.
- Fix member identities— for a group) Specifies that the group is fixed, not parametric. It can only be changed on a catalog group without instances.

Depending on the type of shape selected, you can change the following shape properties:

- Layer—Specifies the layer on which the shape is placed.
- Color—Allows you to choose a color for the shape.
- Cross ref shape—Specifies that the shape is a cross reference shape, so will not show in cross reference labels.
- Diagram Type—Specifies the diagram type of the shape and can only be set on a catalog shape without instances.
- Linked to cross-ref shape— (for a parametric group) allows the shape to be linked to a cross-reference shape, which displays graphically the locations within the design of shapes of other members of the group. It can only be changed on a catalog group shape.
- Associated cross-ref type— (for a group) Specifies the type of cross-reference shape, which is linked with the group shape. Options are:
  - **Other instanced members** Shows the locations of all other shapes of this group, which have been placed in the design.
  - **Member map** Shows all other members of the group as defined by a member map, including those, which have no shapes (yet) in the design.
  - **Master / Slaves** Shows the location of the master shape within the design or, if this shape is the master shape, show the locations of all other shapes (its slaves) within the design

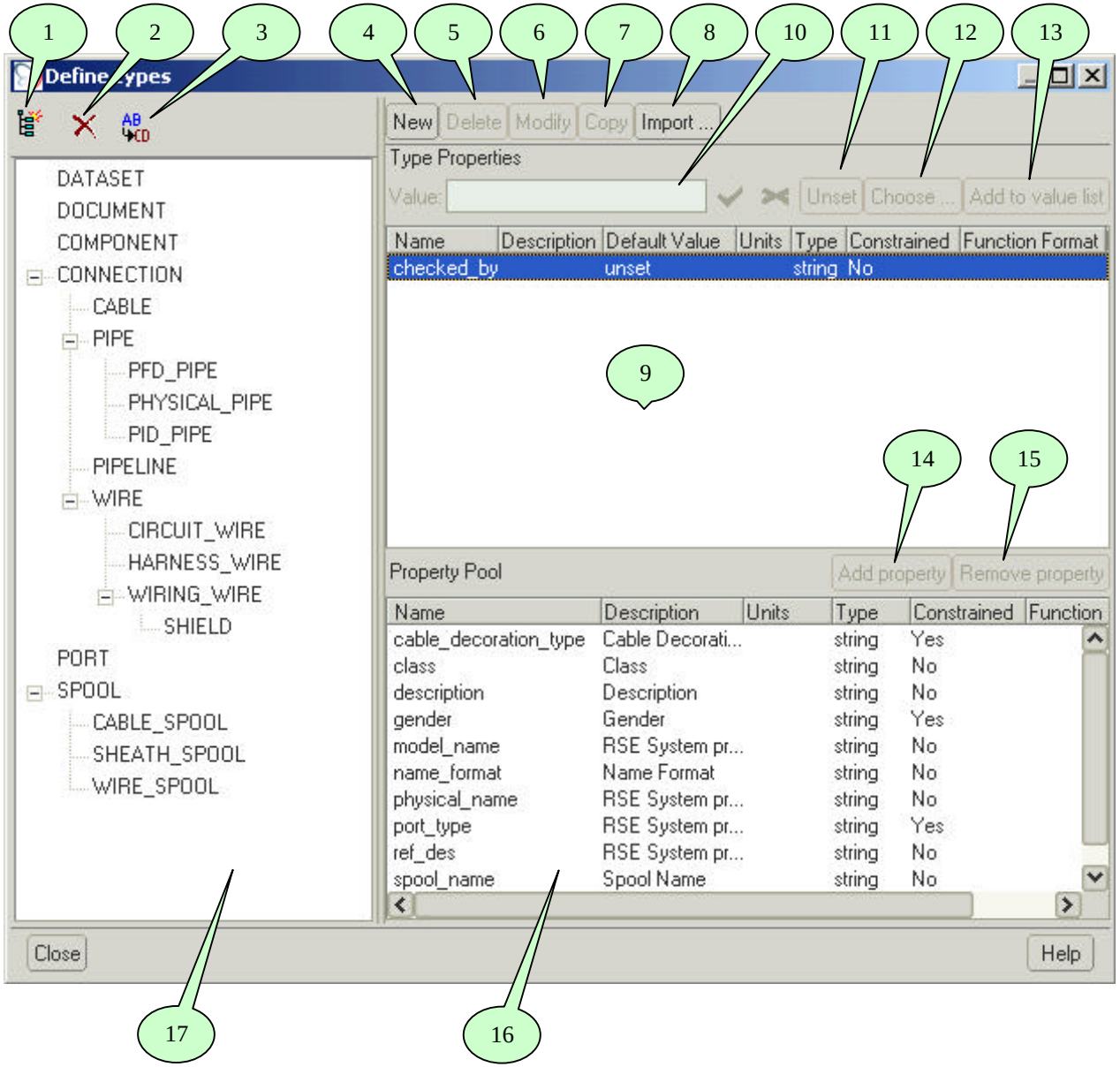
In addition to the previous properties, for a fiber, you can specify shape properties such as line type, pitch, point function, and width. You can also specify label properties such as the position, format, type, layer, font, height, angle, and color.
.

Note:

- The object property dialog boxes work across all modes. You can keep them open as you go from selection to selection. If you switch object types, the Properties dialog box changes accordingly.
- If you have selected multiple objects of a type, the dialog box shows a list of all properties defined for all the selected objects. If the value of a property is the same for all selected objects, the relevant box shows that value. Otherwise, the box is empty.
- When editing properties of a single selected shape, the shape's attributes are displayed at the bottom of the dialog box

Global Properties Dialog Overview

1 Creates a new subtype.
2 Deletes created subtypes.
3 Re-names created subtype.
4 Adds a new property to the pool.
5 Deletes a user-defined property.
6 Modifies a user-defined property.
7 Copies Properties.
8 Imports properties from a *.CSV file.
9 Type property list.
10 Default value for Selected property.
11 Unset's Selected property value.
12 Choose value from a pre-defined list.
13 Adds a new value to property list.
14 Add Selected properties to types from property pool.
15 Remove Selected property from type.
16 Property Pool.
17 Types –Default types are as follows;
New Block COMPONENT
New Group COMPONENT
New Port PORT
New Fiber CONNECTION
New Sheet DOCUMENT



Setting up standard properties for a Drawing Sheet

We shall use properties to ensure that for every time a new sheet is created within the design explorer Routed Systems Designer automatically includes the properties Drawn by, Checked by & Project name

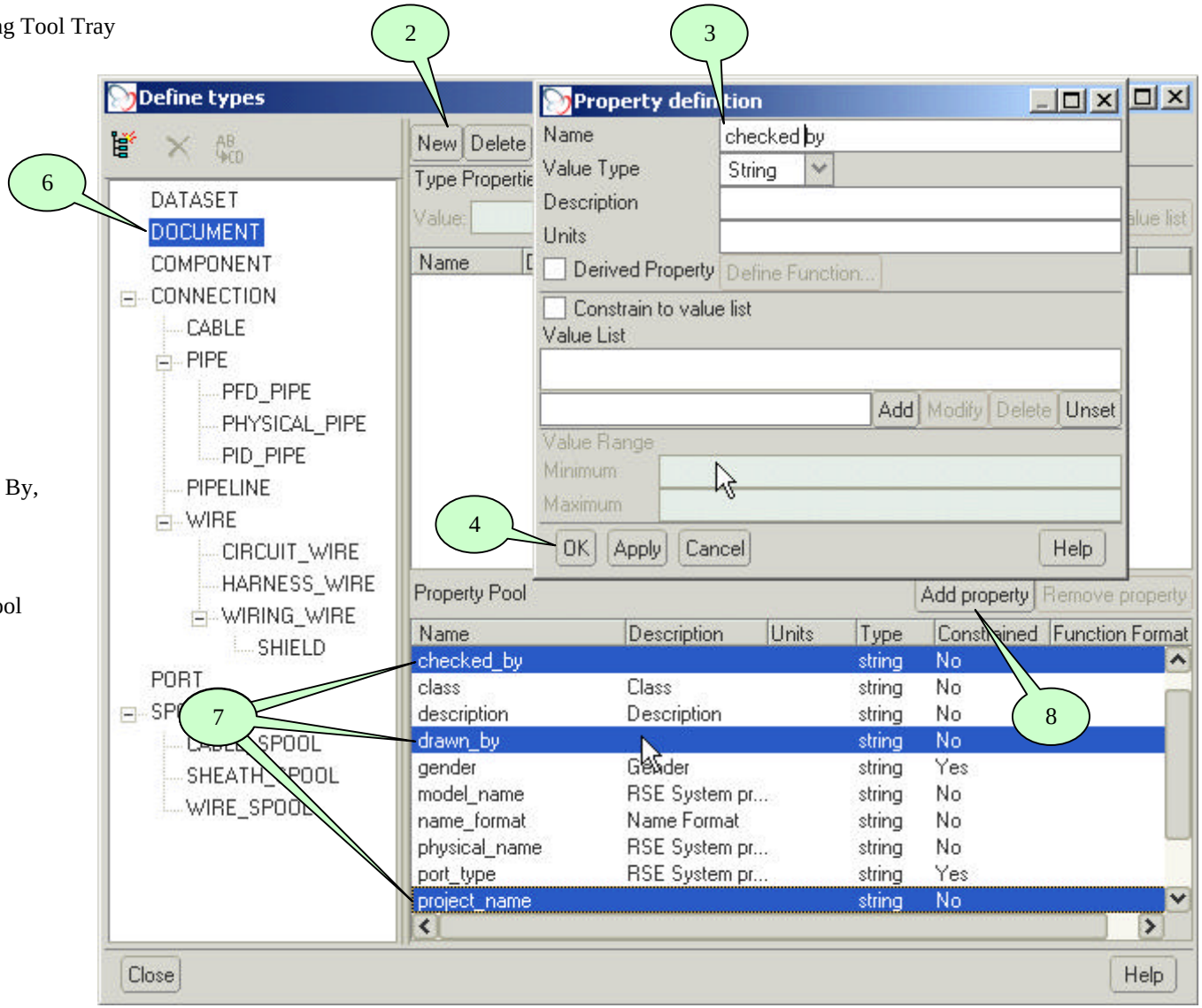
Note: This only has to be done once and may be stored within the company standard catalog.

1. Select Global Properties from the Diagramming Tool Tray
2. Select New
3. **Set** the Name to Checked by
4. **Select** OK to Close the property definition dialogue
5. Repeat steps 2 to 4 for to create two new properties
   - Drawn by
   - Project name

The three new properties we have just created now exist within the Property Pool

6. From the Type menu **Select** DOCUMENT
7. Using **CTRL Select** three new properties (Checked By, Drawn by, Project name)
8. **Select** Add property,

The three properties shall now move from the property pool into the type list



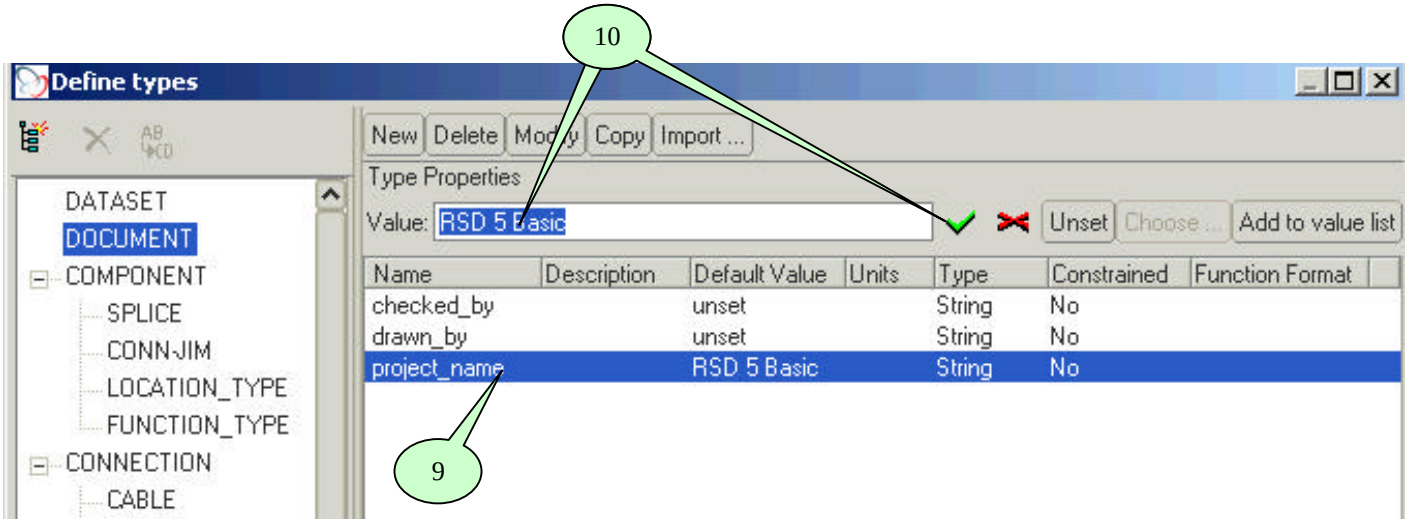
9. From the Type Properties pool **Select** project_name

10. In Value **Enter** RSD 5 Basic > **Select**  to confirm the entry

11. RSD 5 Basic is now set to the default selection

We have now set up 3 new properties that shall always be added when a sheet with type document is created. In this example we have Set RSD 5 Basic to be the default setting for Project name.

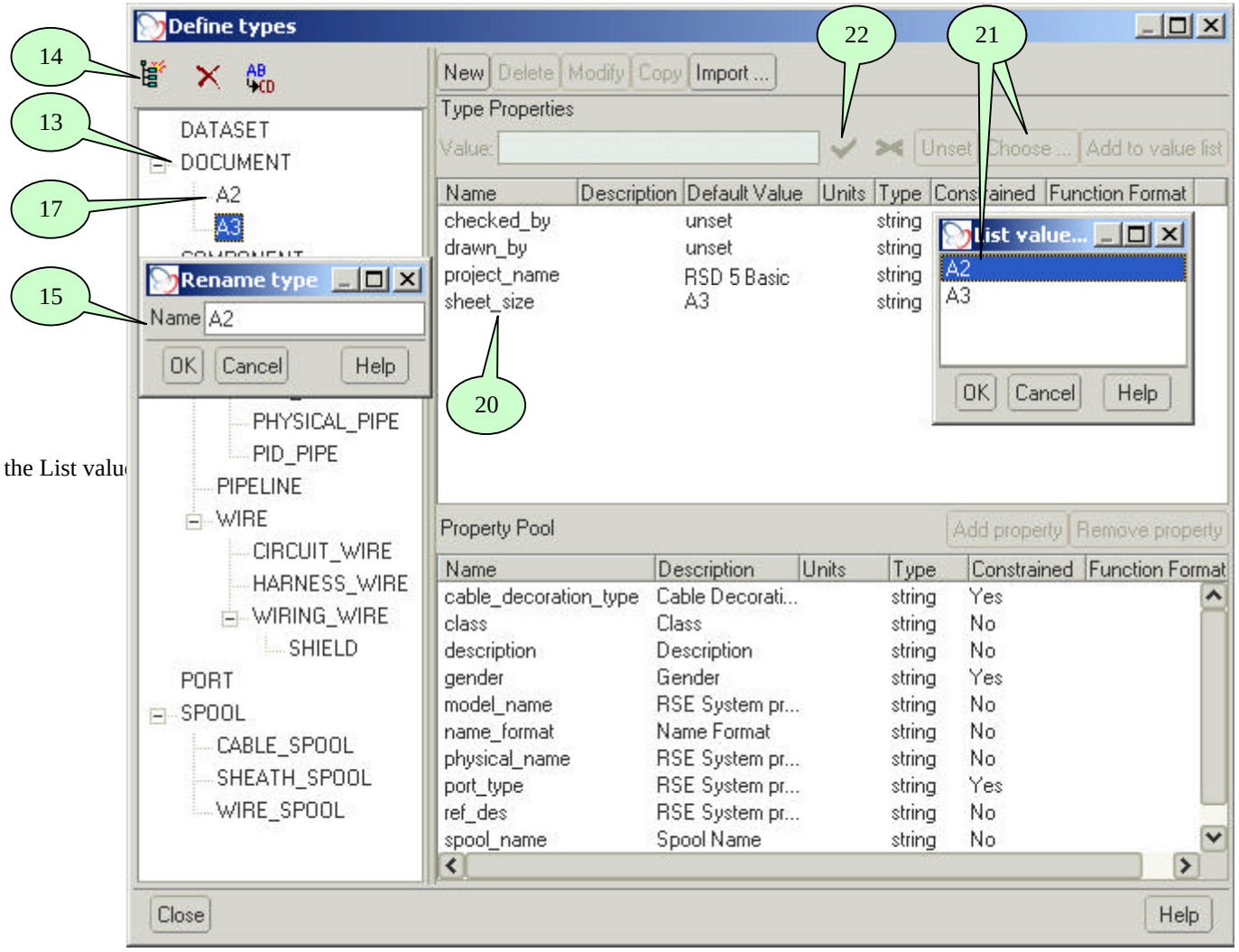
Later we shall create a new sheet and find that the properties of the sheet will have 3 new properties added automatically, and the property project name will automatically have its value set to RSD 5 Basic



Creating Subtypes

We can take global properties a stage further and have subtypes. We shall now create some subtypes for Document that would only be relevant for sheets of a particular size.

12. **Select** Global Properties from the **Diagramming Tool Tray**
13. From the type list **Select** Document
14. **Select** Add new subtype
15. **Enter** A2 for name
16. From the rename type dialog **Select** OK
17. **Select** subtype A2
18. **Select** sheet_size from the property pool
19. **Select** Add Property
20. **Re-Select** sheet_size from the type list
21. **Select** Choose > **Select** A2 > **Select** OK to Close the List valu
22. **Select** to confirm the value
23. Repeat steps 14 – 22 and create a subtype A3
24. From the Define types dialog **Select** Close



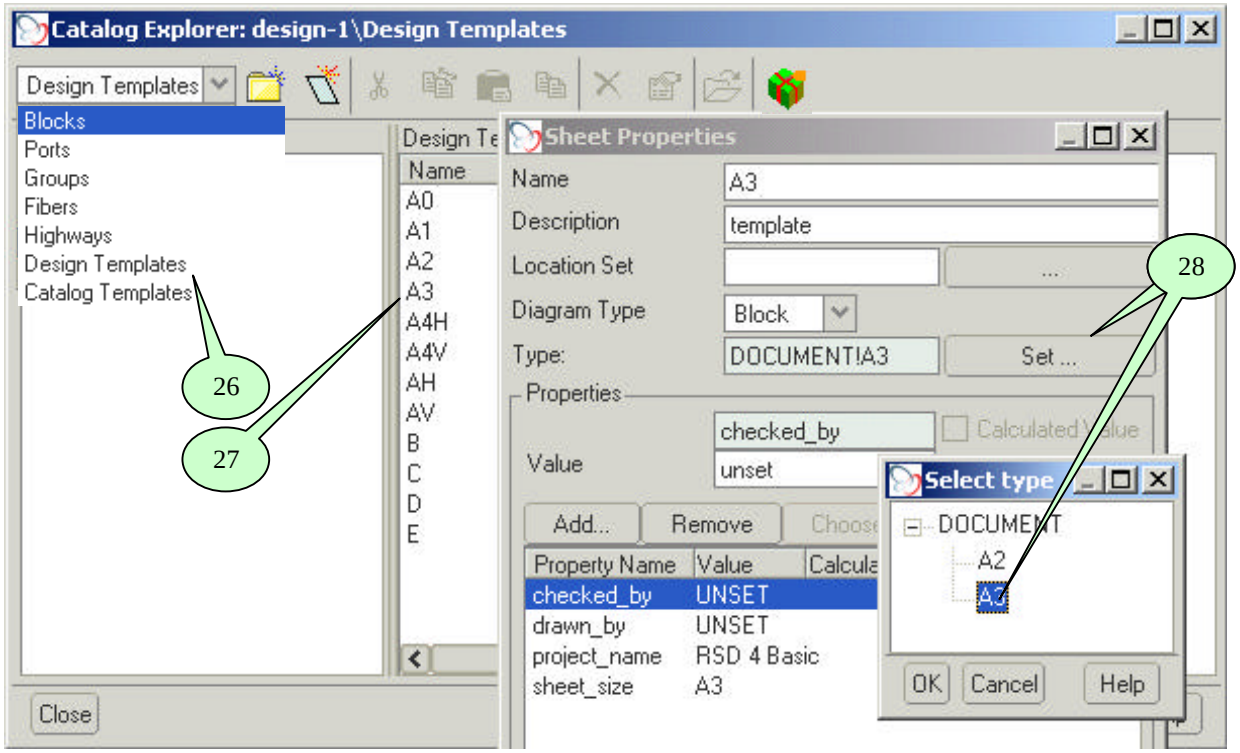
Using properties with catalog templates

We have setup 3 new properties for type DOCUMENT and one new property for subtypes A2 and A3. We shall now setup our catalog template to ensure that each time a user creates a new A2 and A3 sheet the new properties are always included.

25. **Select**  from the main tool bar to Open the Catalog Explorer
26. **Select** Design Templates from the pull down menu
27. **Select** A3 > **Click** RMB > **Select** Properties
28. Select Set > Select A3
29. From the select type dialogue **Select** OK

Notice that 4 new properties are added.

30. From the properties dialogue **Select** OK
31. **Close** the catalog explorer



Creating a new sheet

Now we have setup our A3 design template lets create a new A2 and A3 sheet to check our settings.

32. From the main tool tray **Select**  to Open the Design Explorer

33. From the design explorer **Select**  to create a new sheet

34. **Select** template size **A3**

35. **Select** Sheet_1 > **Click** RMB > **Select** Properties

You will see that Sheet 1 has 4 properties 2 of which have values that are unset

36. Repeat Steps 33 – 35 but this time use template **A2**

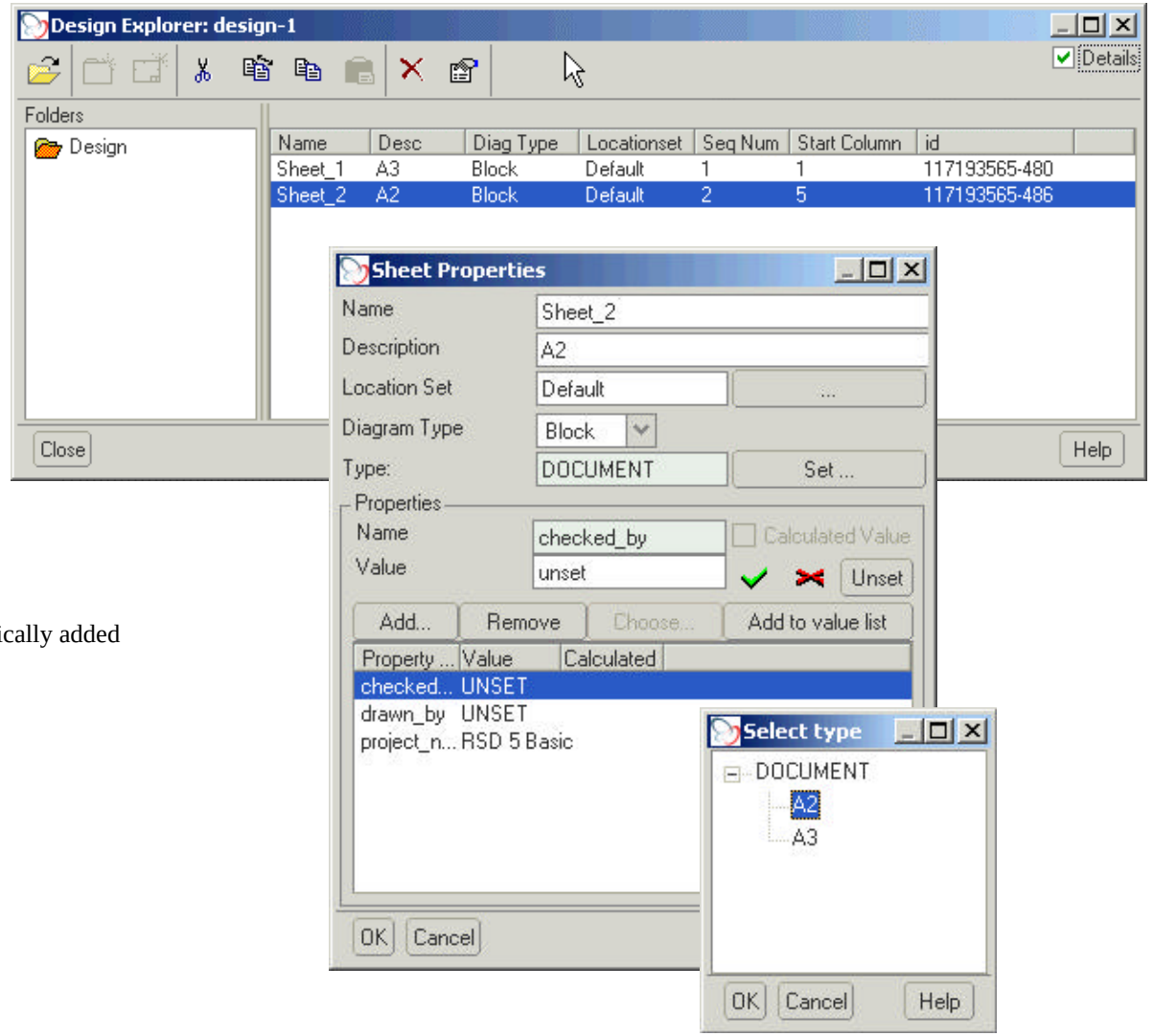
You will notice that sheet_2 has only 3 properties,

37. Select Set

38. Select A2

You will now see that property sheet size has been added with a value of A2 automatically

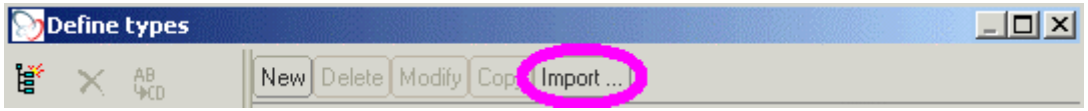
39. **Set-up** the A2 template sheet so that property sheet size is automatically added when an A2 sheet is created



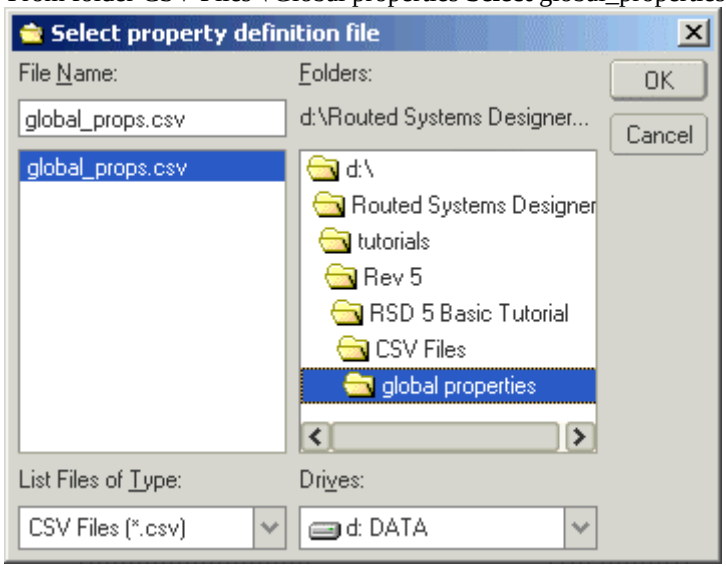
Using CSV files

We can also use CSV files to import global properties.

40. **Select** the Global Properties  from the **Diagramming Tool Tray**
41. **Select** IMPORT



42. From folder CSV Files \ Global properties **Select** global_properties.csv



43. From the select property dialogue **Select** OK.

You will now notice that the global properties has been populated with information from the CSV file

44. **Close** the Global Properties Dialog

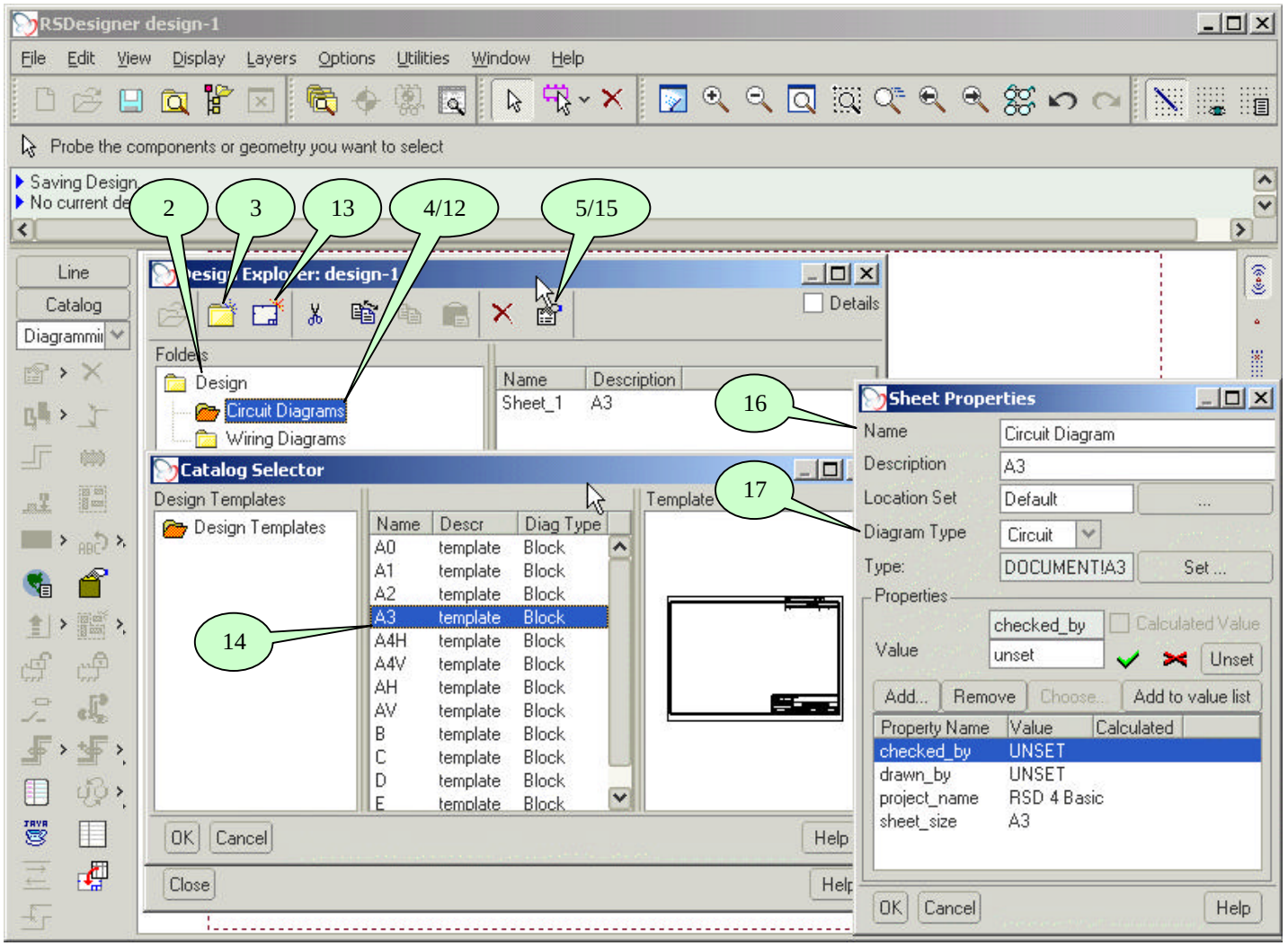
Circuit Diagrams

Creation a new sheet

In this diagram type you can perform the following activities:

- Create fiber networks, often called signal networks
- Inline internally connected blocks without breaking the signal network

1. **Select**  to Open the Design Explorer
2. **Select** folder Design
3. **Select**  to create a new folder
4. **Select** the new folder
5. **Select**  to edit the folder properties
6. **Set** the folder name to Circuit Diagrams
7. **Select** folder Design
8. **Click** RMB > **Select** New Folder
9. **Select** the new folder
10. **Click** RMB > **Select** Properties
11. **Set** the folder name to Wiring Diagrams
12. **Select** Folder Circuit Diagrams
13. **Select**  to create a new sheet
14. **Select** A3 & **OK** from the Catalog **Select**or
15. **Select** the new sheet > **Select** 
16. **Set** the sheet name to Circuit Diagram
17. **Set** the Diagram type to Circuit
18. **Select OK** from the sheet properties dialog
19. **Double Click** with LMB on the sheet to Open
20. **Close** the Catalog Explorer

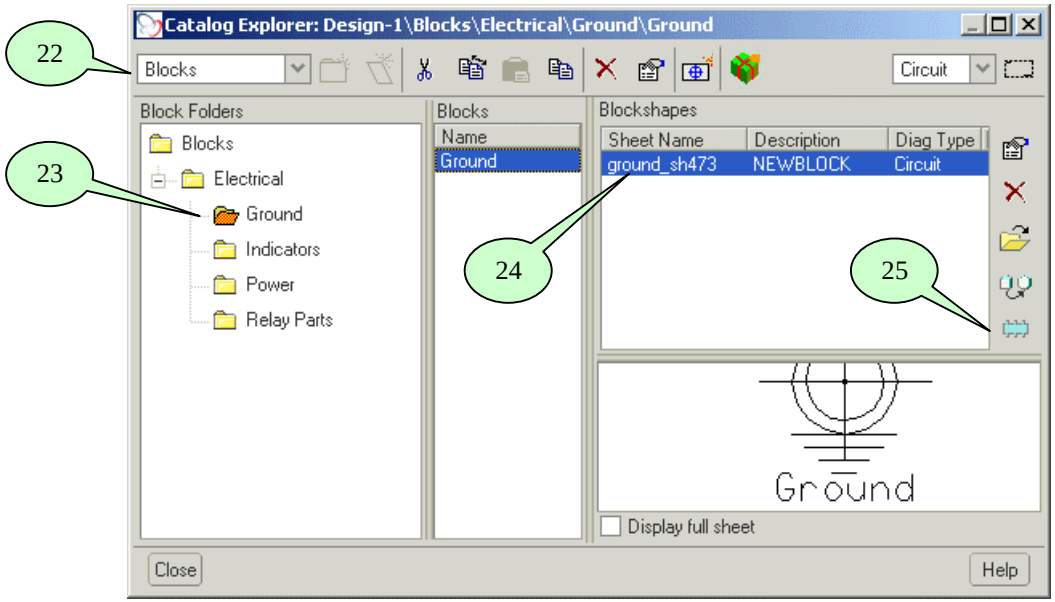


Placing Shapes

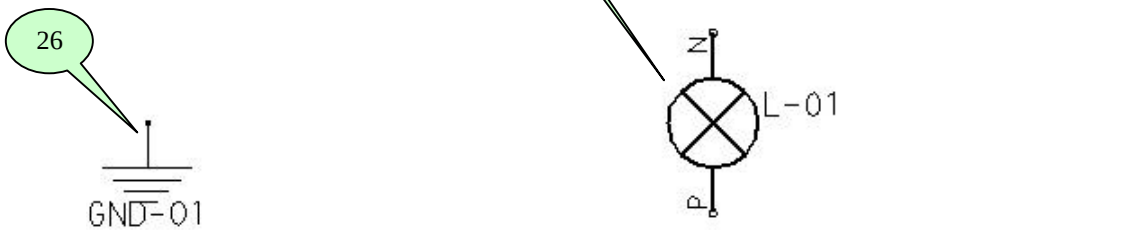
About shortcut options

During the placement of shapes the following shortcuts are available
The **MMB** allows free rotation
On the **RMB** you have the option to

Turn Left	Rotates the shape left
Turn Right	Rotates the shape right
Straight	**Set** the shape to the default orientation
Set Rotation	0, +90, -90,180, user defined value
Magnify	LMB Proportional / RMB Free
Mirror H	Mirror the shape horizontally
Mirror V	Mirror the shape vertically
Drag by port	**Select** a port to attached cursor to for placing
Exit Tool	Cancels placement



21. **Select**  to Open the Catalog Explorer
22. **Select** Blocks from the pull down menu
23. **Select** folders Electrical > Ground
24. **Select** block Ground
25. **Select**  place an instance of the shape
26. The ground shape is now on your cursor, **place down** using the LMB
27. **Click** RMB > **Select** Exit Tool
28. Repeat steps 22 to 27 for Power box from the Power directory (rotate 180 duri placement), and Signal Lamp, from the Indicators directory.
29. **Select** Groups from the pull down menu
30. **Select** the Relays Folder
31. **Select** the Coil shape form the Variable Relay group
32. **Select**  to place and instance of the coil
33. **Select** Cancel from the New Reorder Dialog
34. **Place** the coil down
35. **Click** RMB > **Select** Exit Tool
36. **Select** Close from the catalog explorer dialog



About Signal Networks

A signal network is made up of multisegment fibers created in circuit diagrams when a network shape has been specified. A fiber network is a signal network. Any fiber shape with a diagram type of circuit is regarded as a signal network.
A circuit diagram consists of components, represented either by block graphics or group graphics, connected together by signal networks. A signal network represents the logical connection between the component's ports. Generally, a network of wires is created in the wiring diagram that is equivalent to the signal networks in the circuit diagram. Signal networks can be simple or complex.

- Simple signal network—A circuit diagram consisting of only two pins with a single fiber connecting the two ports. A simple signal network may correspond to multiple wires connected via intermediate connectors.
- Complex signal network—A circuit diagram signal network consisting of three or more ports is a complex network.

The minimum number of wires required to connect a number of ports is one less than the number of ports connected. For example, two ports require one wire, three ports require two wires, four ports require three wires, and so on.

To Instance Signals

In the Diagramming tool tray, Click  . The Catalog Selector dialog box opens.

- Click a fiber definition and then a fiber shape.
- Click OK in the Catalog Selector dialog box or double-Click the fiber. The cursor assumes a circle and dot shape, ready to trace the fiber onto the sheet. Specify the start point by Selecting a port or clicking Free Point on the right mouse button menu to start the fiber anywhere. The fiber is shown tracked in a dotted line.
  - To use automatic routing, click the terminal port. Routed Systems Designer Selects a route for the fiber. Automatic routing routes the fiber to avoid crossing the two blocks being connected, but does not necessarily avoid any intervening blocks or other fibers.
  - To use manual routing, left-Click to install a corner where desired. Click to terminate on a port. To terminate "in space," Click Terminate from the right mouse button menu.

While placing a fiber, use the right mouse button menu to switch sheets or to change to a new instance.

- If you route a fiber shape on a sheet, then Select Same instance from the right mouse button menu, and then route another fiber shape (on the same sheet), a single fiber artifact is created. This artifact contains three ports, two segments, and two fiber shapes associated with it. The first fiber shape shows the first segment and its two ports. The second fiber shape shows the second segment and its two ports. The port that is common to both segments has portshapes in both fibershapes.
- If you route a fiber shape on a sheet, then Select Change sheet from the right mouse button menu, and then route another fiber shape (on another sheet), a single fiber artifact is created. This artifact contains three ports, two segments, and two fiber shapes associated with it. In this case, the fiber shapes are on different sheets.

If you choose new instance before routing a fiber shape, a new fiber artifact is created.

Note:

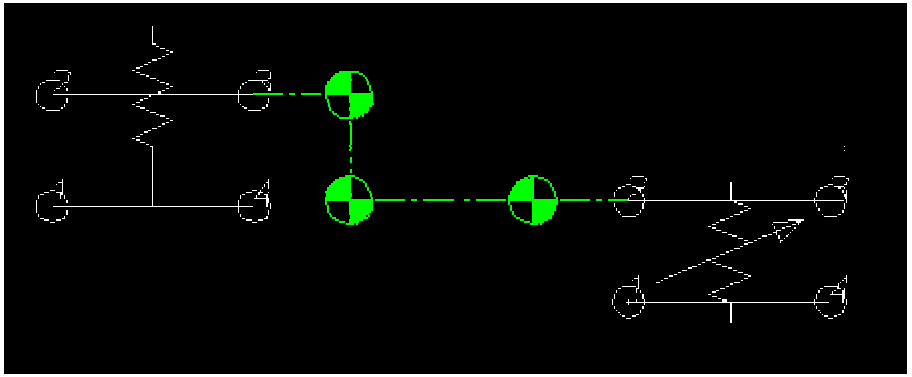
- You can use manual routing with automatic routing. For example, if you know that you want to route in a particular direction, you can start to manually route a fiber, placing several points in space. At any time you can position the cursor over the destination port, and the router will illustrate the autoroute from the current point in space to the destination port as a solid line. If you are satisfied with the remainder of this route, Click on the port and the fiber will be routed.
- Fiber end geometry is always created if a fiber port is connected to a non-fiber port.
- You can customize a shortcut tool to instance a specific fiber and shape each time you Select the shortcut tool, rather than selecting it from the catalog each time.

To manipulate Signals

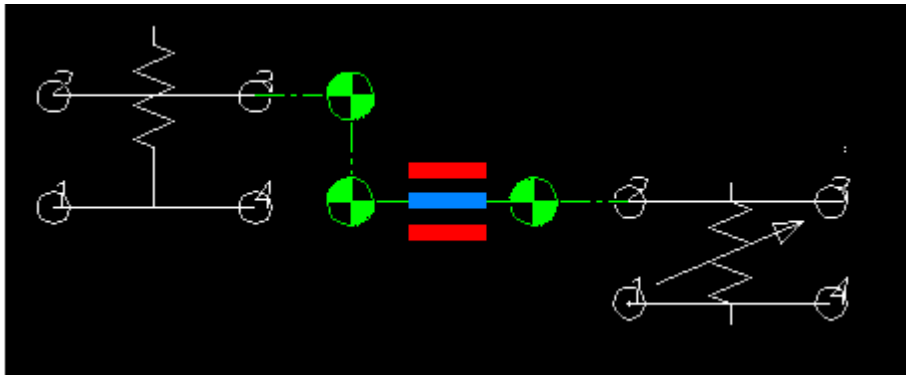
To manipulate a fiber either double Click the fiber with the LMB or Select  from the diagramming tool tray.

To edit a fibre path;

- Double-Click the target fiber path. The path is shown as a centerline with dots at each corner. When the fiber is in this mode, you can:
  - Use the right mouse button menu to Add points (corners).
  - Drag Selected segments as necessary.
  - Delete a point by moving it to coincide with another point.
  - Move a fiber from one port to another by placing the cursor over the port until you see a red box with a blue dot in it. Press the left mouse button and drag the fiber to the new port.
- To finish, Click Exit Tool from the right mouse button menu.



In this figure, the fiber path is in Edit mode.



When the cursor is over a segment, you can drag it. You can also add a crossing (gap or hoop) where the fiber crosses another fiber.

About Context-sensitive cursors

Create

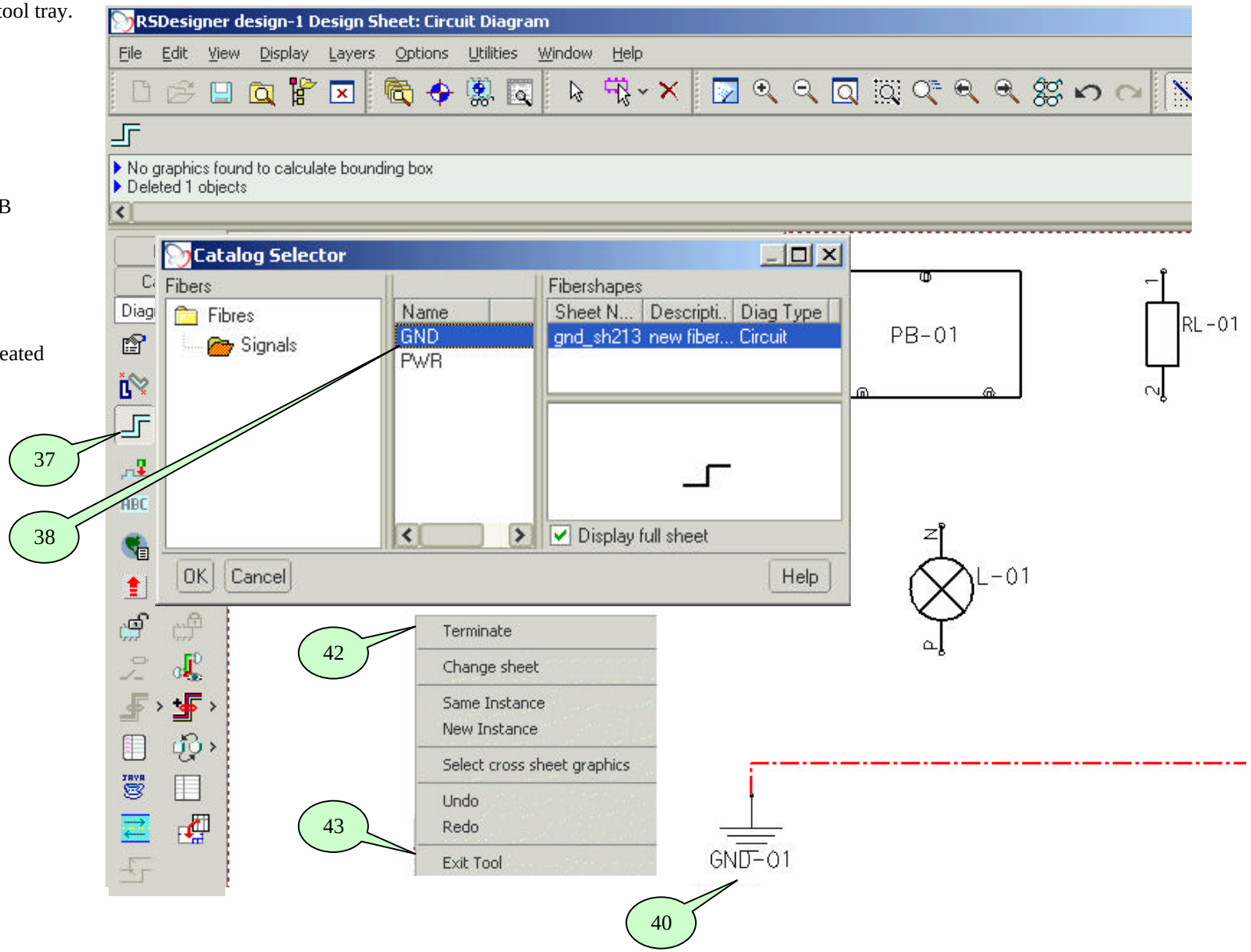
Unix	Win	Meaning
		Auto-portshape
		Connect to portshape immediately to the right of the cursor
		Connect to the portshape under the cursor
		Cannot connect to the portshape under the cursor because its diagram type does not match that of the fibershape prototype
		Cannot connect to the fiber under the cursor because the branch-shape being used won’t fit into the line segment
		Cannot insert a point under the cursor because free-point has not been enabled
		Cannot connect to the portshape under the cursor because such a connection is disallowed by the user-defined Java
		Insert a network shape
		Insert a branch shape
		Free-point has been enabled

Edit

Unix	Win	Meaning
		Slide network portshape along the fiber
		Move a point on a fiber
		Move a point on a fiber such that the two adjacent line segments are both at angles that are multiples of 15 degrees
		Move a line segment
		Move and reconnect a fiber end, set when the cursor is over the Fiber’s end-geometry
		Select fiber segment
		Cursor not over a selectable part of the fiber being edited
		Only add-point is available because the cursor is over the first or last line segments – not including the end-geometry

Routing Signals

37. **Select**  from the diagramming tool tray.
38. **Select** Signals > GND
39. Select OK
40. Using **LMB** start at GND-01
41. **Move** to the right > **Place** with LMB
42. **Click** RMB > **Select** Terminate
43. **Click** RMB > **Select** Exit Tool
44. **Select** label G01 on the fibre just created
45. **Click** RMB > **Select** Delete



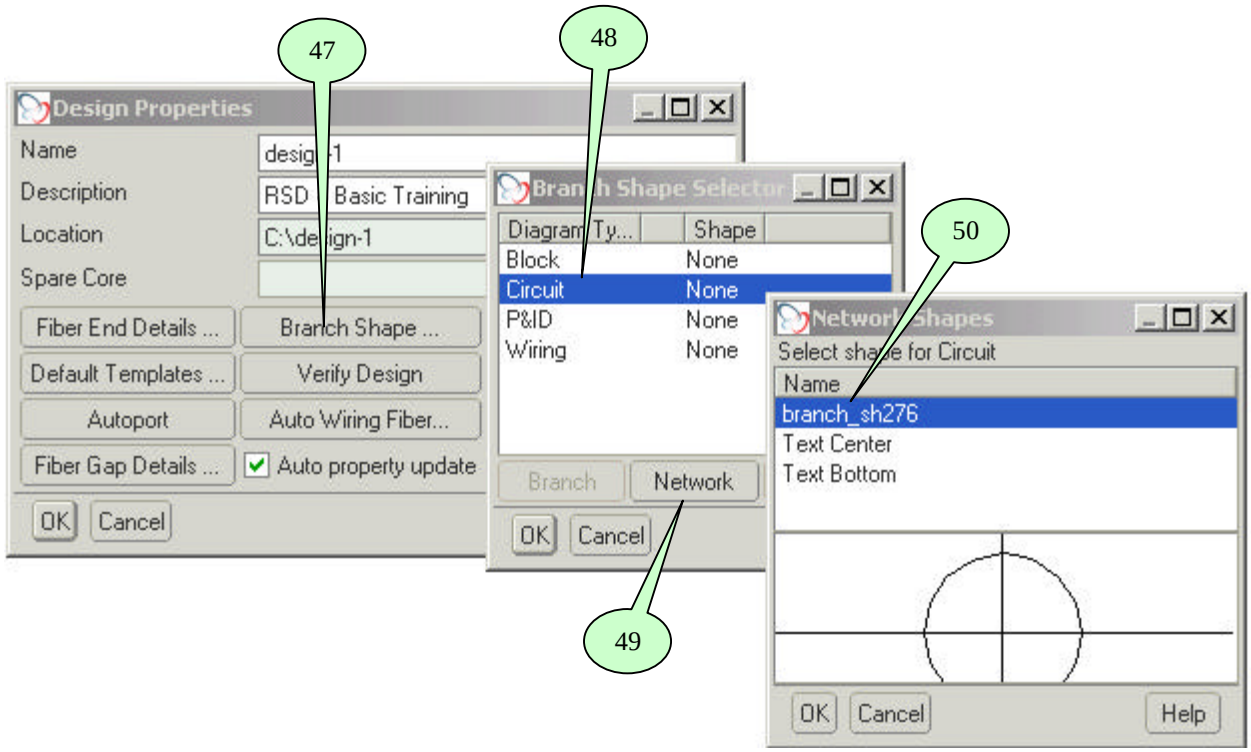
Branching - Set-up

We now need to branch into the G01 Before we can do this we need to set the Network shape

You can set a branch or a network shape for a diagram type.

The Branch Shape Selector dialog box displays the branch shape types and the names that you have selected. Only valid branch shapes are available for selection and are displayed in the Branch Shape Selector dialog box. A valid branch shape is the one that matches the selected diagram type and has three ports, at 0°,-90°, and 180°. The 0° and 180° ports must be on the same horizontal line. The -90° port must not be above the other two ports.
If a branch shape is already set for a diagram type, it is already selected and it appears in the viewer. If the branch shape is not set, the viewer is empty. Click Branch or Network on the Branch Shape Selector dialog box to select a branch or a network shape for a fibershape in the catalog. The shape is used when branching a fiber from another fiber.
You can also select Branch or Network in the Fiber Properties dialog box to set the branch or network shape for a fibershape in the catalog.

46. **Select** File > Design Properties
47. **Select** Branch Shape
48. **Select** Circuit
49. **Select** Network
50. **Select** branch_sch276
51. From the Networks shape dialog **Select OK**
52. From the Branch shape Selector dialog **Select** OK
53. From the design properties dialog **Select** OK

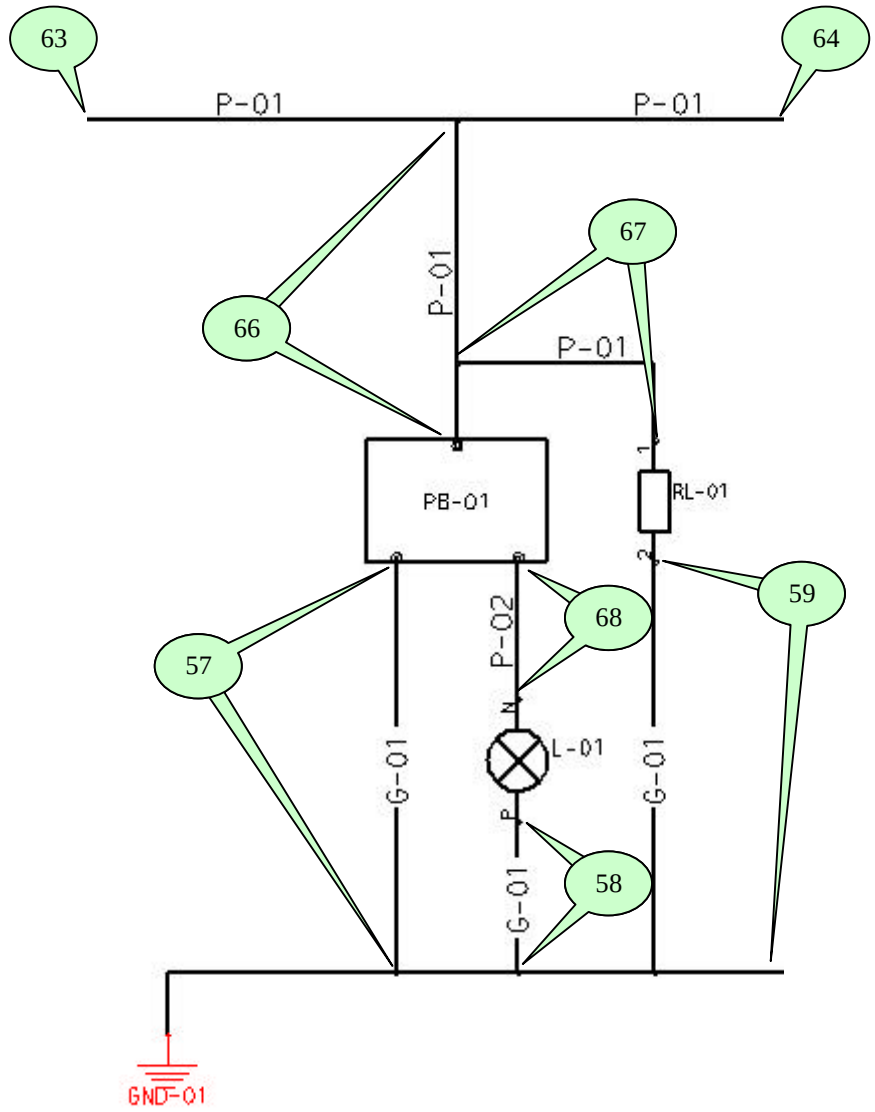


Branching – Routing Signals

54. Select  from the Diagramming tool tray.
55. From folder Signals **Select** GND
56. Select OK
57. **Route** from PB-01 Pin 3 & **branch** into G-01

Note: If you cannot branch check that you have set the branch shape correctly

58. **Route** from L-01 and **branch** into G-01
59. **Route** from RL-01 and **branch** into G-01
60. **Click** RMB > **Select** Exit Tool
61. **Select**  from the diagramming tool tray.
62. **Select** Signals > PWR > **Select** OK
63. **Click** RMB > **Select** Free Point > **Place** with LMB
64. **Move** to the right > **Place** with LMB
65. **Click** RMB > **Select** Terminate
66. **Connect** Pin 1 of PB-01 and **branch** into signal P-01
67. **Connect** Pin1 of RL-01 and **branch** into the Vertical section of signal P-01
68. **Connect** Pin 2 of PB-01 to the remaining pin on L-01
69. **Click** RMB > **Select** Exit Tool
70. **Delete** any unwanted labels



Group Shapes

Groups are used to associate multiple blocks, fibers, groups, or ports together in a single object, for example, to represent several logical components that are all in the same part. An example of this is the individual contacts and coils in a relay

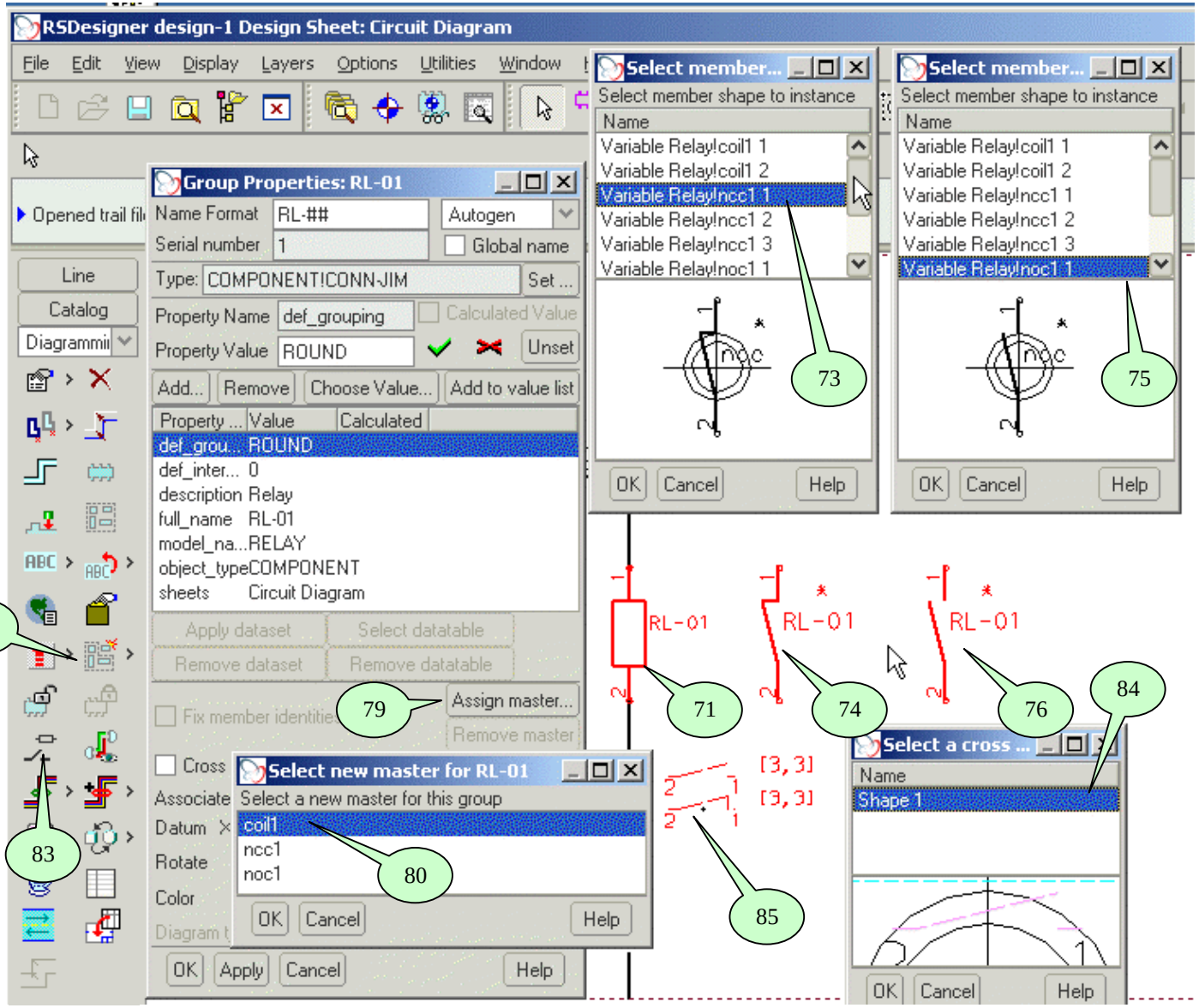
We shall now place down a NCC and a NOC to complete RL-01 our group shape

71. **Select** RL-01
72. From the Diagramming Tool Tray Select
73. **Select** Variable Relay!ncc1_1 > **Select** OK
74. **Place** to the right of RL-01
75. **Select** Variable Relay!noc1_1 > **Select** OK
76. **Place** to the right of NCC
77. **Select** cancel from the **Select** member shape to place dialog
78. **Select** the coil > **Click** RMB >**Select** Properties
79. **Select** Assign master
80. **Select** Coil1 > **Select** OK
81. From the group properties dialog **Select** OK

Notice that the NCC and NOC label is now reporting the position of the coil. You may need to **Select** to update the labels

82. **Select** the coil
83. From the Diagramming Tool Tray Select
84. **Select** Shape1 > **Select** OK
85. With the LMB **Place** the Cross Reference Shape near the coil.

Cross-reference shapes are visual aids that help keep track of the relative locations of the various members of a variable unconstrained group throughout the design. Cross-reference shapes show those members of the group that are placed at the time the cross-reference shape is created or last updated.



Location Sets

About Location Sets

Routed Systems Designer uses location sets to control how sheet columns and rows are defined and sequenced on sheets. A location set is used to group sheets together for cross-reference purposes. Three location setup tools (under one button set) are available in the Catalog tool tray:

- Create and Manage Location Sets
- Set Sheet Column and Row Width
- Set Sheet Column and Row Labeling Datums

Routed Systems Designer has a default location set. All design sheets are placed in this location set by default. You can use the Location Set Explorer to create new location sets and to move a sheet from one location set to another. A sheet can exist only in one location set at a time.
For each sheet in the location set, you can define:

- The number of columns per sheet
- The number of rows per sheet
- The width of each column
- The width of each row
- The start and end positions to place the column or row labels on the sheet

Define these settings for all the design template sheets in your catalog. You can subsequently modify the settings for a design sheet, if required.

To Create or Edit Location Sets

Click  in the Catalog tool tray to create location sets or modify the behavior of an existing location set. The Location Set Explorer dialog box opens.
The Location sets list displays all location sets defined for this design. The Sheets list displays the sheets that are currently assigned to the selected location set, its order in the set, and the starting column number for the sheet. The entry <<No location set>> shows the sheets that are currently not assigned to a location set.

Creating a new location Set

To create a new location Set, Click Create new. A new entry is listed in the location sets, with a default name. Type a new name in the Name box and Click Apply.

Setting label visibility

To specify whether the sheets must display row or column labels and to specify where on the sheet border the labels should be displayed, select the appropriate check boxes under Label position.

Setting the label function

To specify user-defined label formats for a row or column using a Java function call, select the appropriate check box under Label function, and specify the function to be called in the box.
The following default function is provided:
{rsdesigner/component/Label.alpha()}
This method results in the row or column text **Set** as A, B, C through Z corresponding to row or column numbers 1, 2, 3, through 26 respectively.
Other example label functions are:

- {rsdesigner/component/Label.alpha(ROW)}. This method precedes the alphanumeric string with a user-specified string. In this example, the row text is ROW A, ROW B, ROW C and so on.
- {rsdesigner/component/Label.NFPA(name)}. This method precedes the alphanumeric string with the value of a specified string property of the current sheet. In this example, the row text is Sheet_1 A, Sheet_1 B, Sheet_1 C and so on.

Setting sequential sheet numbering

Select the Sequential Sheet Numbering check box to number the columns starting at 1 for the first column of the first sheet, then increasing continuously across all sheets in the sheet set.

To remove gaps in sequential numbers, **click**  .

To Set the Column Width or Row Height

- Click  in the Catalog tool tray. The Set Sheet Columns and Rows dialog box opens.
- Type the number of columns and rows in the Num columns and Num Rows boxes respectively, and Click  to update the list with the specified number of columns or rows. The dialog box lists each column and row with its current width or height. The labeling datum's define the area that the rows and columns are defined for, so the sum of the column widths must equal the distance between the labeling datum's. If the sum does not equal the distance then the difference is shown in the Deficit label. Ensure that the number is zero before you Click OK. To get a deficit of zero, you can:
  - Click Equal widths or Equal heights. The column widths or row heights are distributed equally among all columns or rows.
  - Select a row in the list and Click Add to Selected. The deficit value is added to the Selected row, the row is updated to reflect the new value, and the deficit is changed to zero.
  - Manually alter the value of each width. Select the column to be altered, type a new value in the Width box, and Click  .
- Click OK or Apply. The new column and row definitions are stored for the sheet. The labels are displayed around the borders of the sheet, between the two labeling datum positions, dependent on the Label Position choices in the Location set Explorer.

To Set the Labeling Datum's

- Click  in the Catalog tool tray.
- Select the point of the first labeling datum and a position for the last one. These two points are treated as defining a diagonal line. The column labels are positioned along horizontal lines at the top and bottom of the diagonal line; the row labels are positioned along vertical lines at the left and right of the diagonal line. When you select the second point, the set Sheet Columns and Rows dialog box opens.
- Complete the set Sheet Columns and Rows dialog box as described in "To set the Column or Row Width".

To Move Sheets Between Location Sets

- Click  in the Catalog tool tray. The Location Set Explorer dialog box opens.
- In the Sheets list, Select the sheet that you want to move and Click one of the following:
  -  Moves the sheet up in the sequence
  -  Moves the sheet down in the sequence
  -  Moves Selected sheet to another location set. A list of the possible location sets into which the sheet could be moved is displayed. Select a location set from the list and Click OK. The sheet is moved into the new location set. Alternatively, you can select a location set into which you want to Add a sheet, and Click Add Sheet. A list is displayed showing all sheets that are not already in the Selected set. Select a sheet and Click OK. The Selected sheet is moved from its previous set into the Selected one.

Note: Moving sheets between location sets can affect the column numbers of the sheet itself, as well as column numbers of other sheets in the affected location sets. This in turn can affect cross-references on the sheet itself, as well as any other sheet that contains cross-references to the affected sheets. Click  to update cross-references on the Selected sheet. The sheet is updated only if it is Open. Update All Sheets updates all sheets in the Selected location set. Closed sheets do not need to be updated because they are automatically updated when they are opened in Routed Systems Designer.

To Delete a Location Set

- Click  in the Catalog tool tray. The Location set Explorer dialog box opens.
- In the Sheets list, select the sheet that you want to delete.
- Click Delete. The Selected location set is deleted. All the sheets that were in the location set are moved to the <<No Location set >> list.

To Remove a Sheet From a Location Set

- Click  in the Catalog tool tray. The Location Set Explorer dialog box opens.
- In the Sheets list, Select the sheet that you want to move and Click  . The select location set dialog box opens.
- Select <<No location set >> and Click OK.

Row & Column Text

To control the height, color and font of row and column text Select File > Design Properties > Set Row/Column text

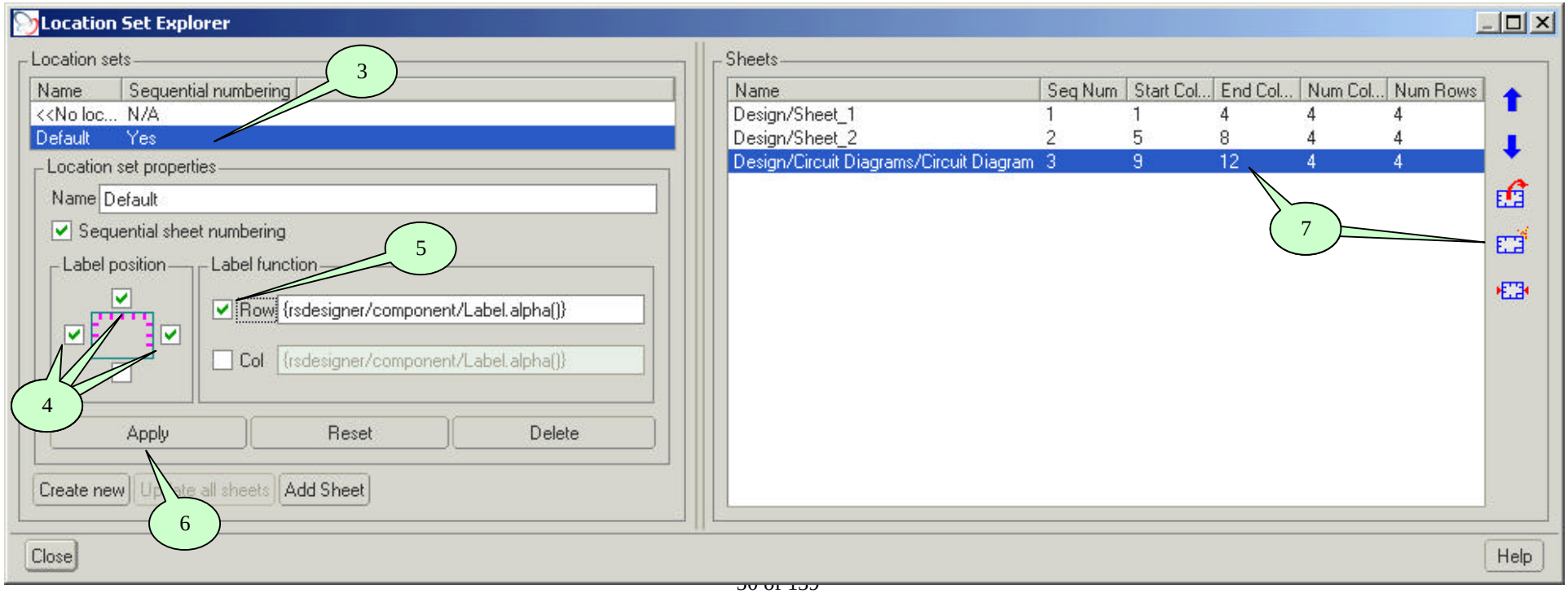
Using Java in location sets

Using Java it is possible to have a user defined sequence for example alphanumeric. We are going to set our rows to an Alpha sequence using Java alpha sequence that is supplied with RSD (not available in RSD Lite).

1. Select the catalog tool tray
2. **Select** the location set explorer icon 
3. **Select** location Default
4. **Set** the label position to be Left, Top and Right
5. **Turn on** the label function (this is calling the Java alpha sequence) for the rows
6. **Select** Apply
7. **Select** Sheet Circuit Diagram >**Select**  to update the cross reference information > **Select** close

You will now see that you have alpha rows and numeric columns and that your relay members and xref shape our now reporting the updated row column locations

.

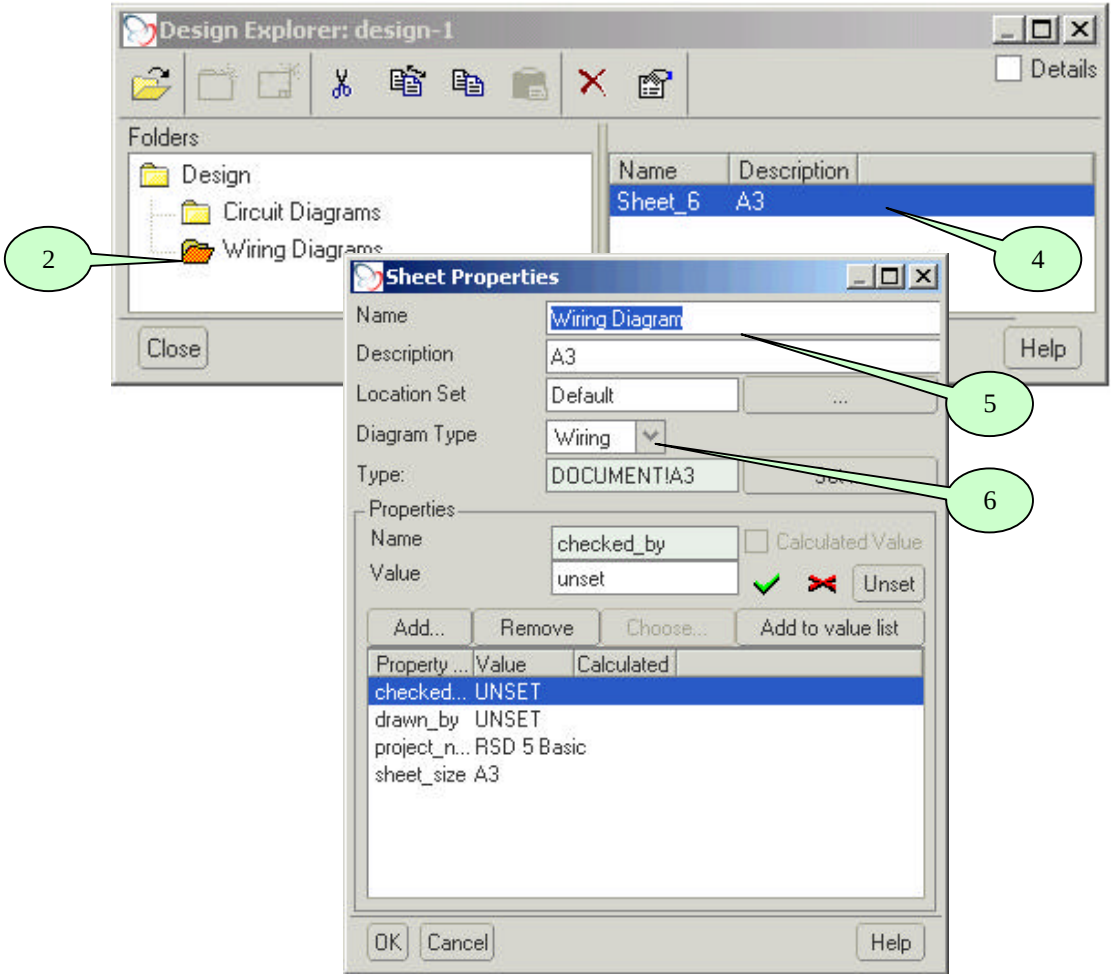


Wiring Diagrams

Now that we have created a part of a circuit diagram lets start to create the wiring diagram.

Creating a sheet

1. **Select** the Design Explorer Icon 
2. **Select** Folder Wiring Diagrams >**Click** RMB > **Select** New Sheet
3. From the catalog Selector dialog **Select** A3 > **Select OK**
4. Within the Design Explorer **Select** the new sheet > **Click** RMB > **Select** Properties
5. **Set** the sheet name to Wiring Diagram
6. **Set** the Diagram type to Wiring
7. From the sheet properties dialog **Select** OK
8. **Select** the renamed sheet > **Click** RMB > **Select** Open Sheet
9. **Close** the Catalog Explorer



Placing Groups with the report tool

We are now going to use the report tool to place the wiring shape

About the report tool

You can generate reports in Routed Systems Designer. You can include the layout of the report and the items to be included in the format. You can also name and save the format for future use.

You can generate the report as a dialog box display, or a text file that you can print, or as a table in the current sheet. If you generate the dialog box display, you can use the dialog box to probe the diagram to find and highlight reported objects.

Click  in the Diagramming tool tray. The Format report dialog box opens. At the top of the dialog box is a toolbar with the following tools:

 creates a new report format.

 opens a file selector that is used to select a previously saved format.

 saves the current report format.

 deletes the file of the current report format. The dialog box is reset to its default state.

 generates the report, sending the report to the specified target.

The tabs below the toolbar allow you to switch between the different panels described below. Use the tabs that are appropriate for your report: General, Column, Sort, Criteria, and Destination. The Trace and Trace Criteria tabs are enabled when creating connection reports only. The Table Style tab is enabled when the destination is sheet.

General

Use this panel to set the report title, type of report, column separator, and classes of object on which to report. There are three report types:

- Properties—generates a report with a line for every object that satisfies all the criteria. This type of report may result in identical lines in the output.
- Inventory—generates a report in which every line is unique and adds a column to indicate the number of objects that generated the line.
- Connection—generates a report showing the connectivity between objects. The Trace and Trace Criteria panes are used to control the output of this type of report.

Line number

To show line numbers in the report, check the Show checkbox beside the Line Numbers Column box. To set the line number style in an inventory report, select Incremental or Hierarchical from the Style list box.

Select General > Item Selection > Criteria to include items that satisfy the specified criteria in the report. Select General > Item Selection > Selected Items to include the currently selected items in the report.

Column

Use the Column panel to set which fields are to be shown in the report, along with up to three heading lines and the width of the column. Click Add Property Column to select a property to be displayed as a field in the report, and Delete to remove a field from the report. The Move Up and Move Down control the order of the fields in the report.

Click Add <Label> Column to add a new column whose value is generated from a label format. Use the Property Format text box and the Column Heading text box to edit both the property format and the column heading after adding the new column.

Click [fn(x)] to incorporate a predefined function into the label format text of a label column.

Note: The Property Format label/text and the [fn(x)] buttons are disabled when you select an item in the column list that is a normal property column. These buttons are enabled when you select a label format type column or add a new column of this type. Values change automatically in the list as you type in either of these text boxes.

Sort

Use the Sort panel to set properties that are used for sorting, and specify whether the order is ascending or descending..

Criteria

Use the Criteria panel to set the criteria by which objects are selected. Only objects that satisfy all the criteria are selected.
Note: The properties named in the criteria used to qualify an object for inclusion in a report do not need to appear in the columns list.
Most properties can be used as criteria in reports. The criteria that can be used depend on the type of the property.

Criteria for Integer and Real Properties

Criterion	Description
Equal	Property value is equal to the specified value
Not Equal	Property value is not equal to the specified value
Less Than	Property value is less than the specified value
Less Than or Equal	Property value is less than or equal to the specified value
Greater Than	Property value is greater than the specified value
Greater Than or Equal	Property value is less than or equal to the specified value

Note: The value of an integer or property can be null (or unset) value. If the specified value is unset (that is the unset check box is selected), only the Equal and Not Equal criteria are available.

Criteria for String Properties

Criterion	Description
Equal	Property value is equal to the specified value
Not Equal	Property value is not equal to the specified value
Less Than	Property value is less than the specified value
Less Than or Equal	Property value is less than or equal to the specified value
Greater Than	Property value is greater than the specified value
Greater Than or Equal	Property value is less than or equal to the specified value
Match	Property value matches the specified pattern in which wild cards can be used. Wildcard characters are: ***** denotes any number of any character. **?** denotes any single character. \ is the escape character.

Note: The value of a string property can be null (or unset) value. If the specified value is unset (that is the unset check box is selected), only the Equal and Not Equal criteria are available.

Criteria for List Properties

Criterion	Description
Equal	List equals specified list
Not Equal	List does not equal specified list
Contains	List contains elements in specified list
Not Contains	List does not contain elements in specified list
Match	List matches the specified list with which you can use wildcards. Wildcards that can be used are: ***** denotes any number of any character. **?** denotes any single character. \ is the escape character.

Some built-in properties have their own, specific, criteria.

Criteria for "sheets" Property

Criterion	Description
On	The artifact that owns the property has shapes on one or more of the specified sheets.
On Any	The artifact that owns the property has at least one shape on a sheet.
Not On	The artifact that owns the property does not have shapes on any of the specified sheets.
Not On Any	The artifact that owns the property has no shapes, so is not represented on any sheet.

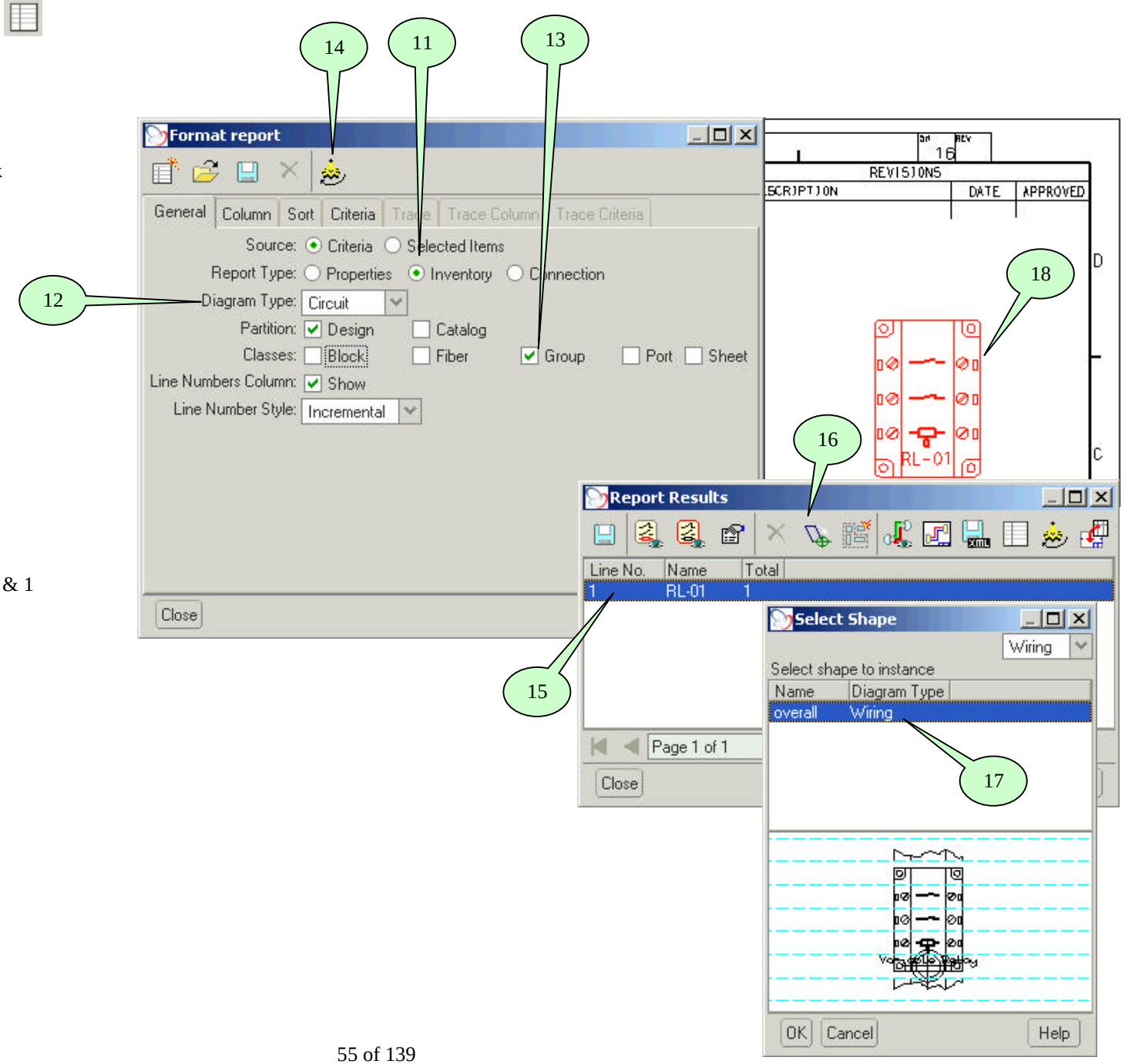
Criteria for the "type" Property

Criterion	Description
Equal	The type of the artifact that owns the property is equal to the specified type.
No Equal	The type of the artifact that owns the property is not equal to the specified type.
Ancestor	The type of the artifact that owns the property is equal to, or is a supertype of, the specified type.
Not Ancestor	The type of the artifact that owns the property is not equal to, and is not a supertype of, the specified type.
Descendent	The type of the artifact that owns the property is equal to, or is a subtype of, the specified type.
Not Descendent	The type of the artifact that owns the property is not equal to, and is not a subtype of, the specified type.
Related	The type of the artifact that owns the property is equal to, or has a common ancestor with, the specified type.
Not Related	The type of the artifact that owns the property is not equal to, and does not have a common ancestor with, the specified type.

Placing Groups with the report tool

10. From the Diagramming Tool Tray Select 
11. **Set** the report to Inventory
12. **Set** the Diagram type to Circuit
13. **Set** the class to Group & **de-Select** Block
14. Select 
15. From the report results **Select** RL-01
16. From the report results **Select** 
17. **Select** Overall > **Select** OK
18. **Place** the Relay
19. **Close** the report results
20. **Close** the Report tool

Notice that the wiring shape has 1 coil, 1 NCC & 1 NOC as defined in the circuit Diagram.

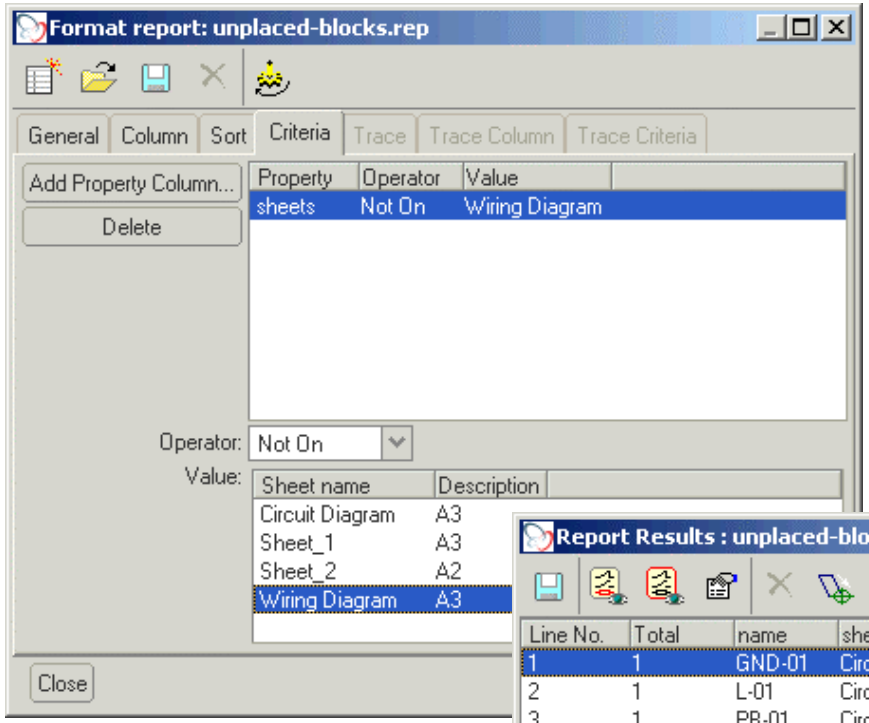


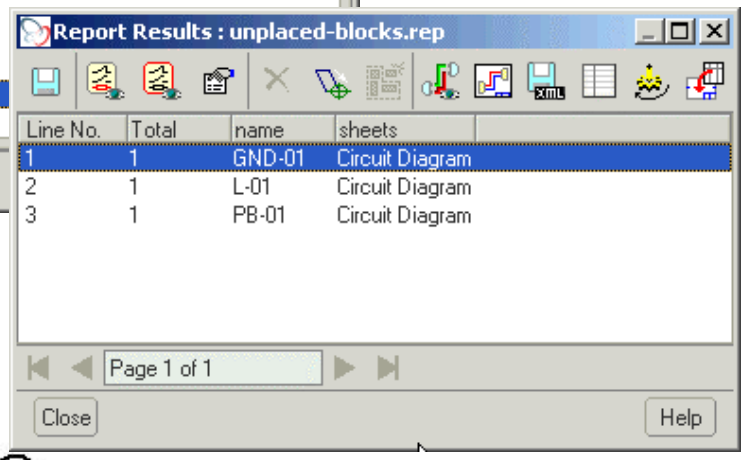
Placing Blocks with the report tool

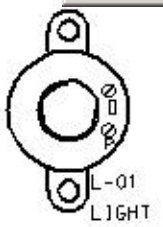
21. **Select**  from the diagramming tool tray
22. **Select** Open 
23. From the reports directory **Select** Unplaced-blocks.rep
24. **Select** the Criteria tab
25. **Select** Sheets
26. **Select** Not On
27. **Select** Wiring Diagram
28. **Select**  to generate the report
29. **Close** the Format Report Dialog
30. From the report results dialog **Select** GND-01
31. From the report results dialog **Select**  to place the wiring shape.
32. **Select** ground_sch480
33. **Place** in the bottom left hand corner of the sheet
34. From the report results dialog **Select**  .

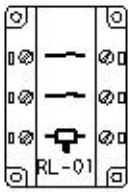
Notice that due to the criteria settings GND-01 is no longer in the report results.

35. Place the remaining shapes









Now that we created the wiring view of the circuit diagram lets now create our own catalog items to finish our design.

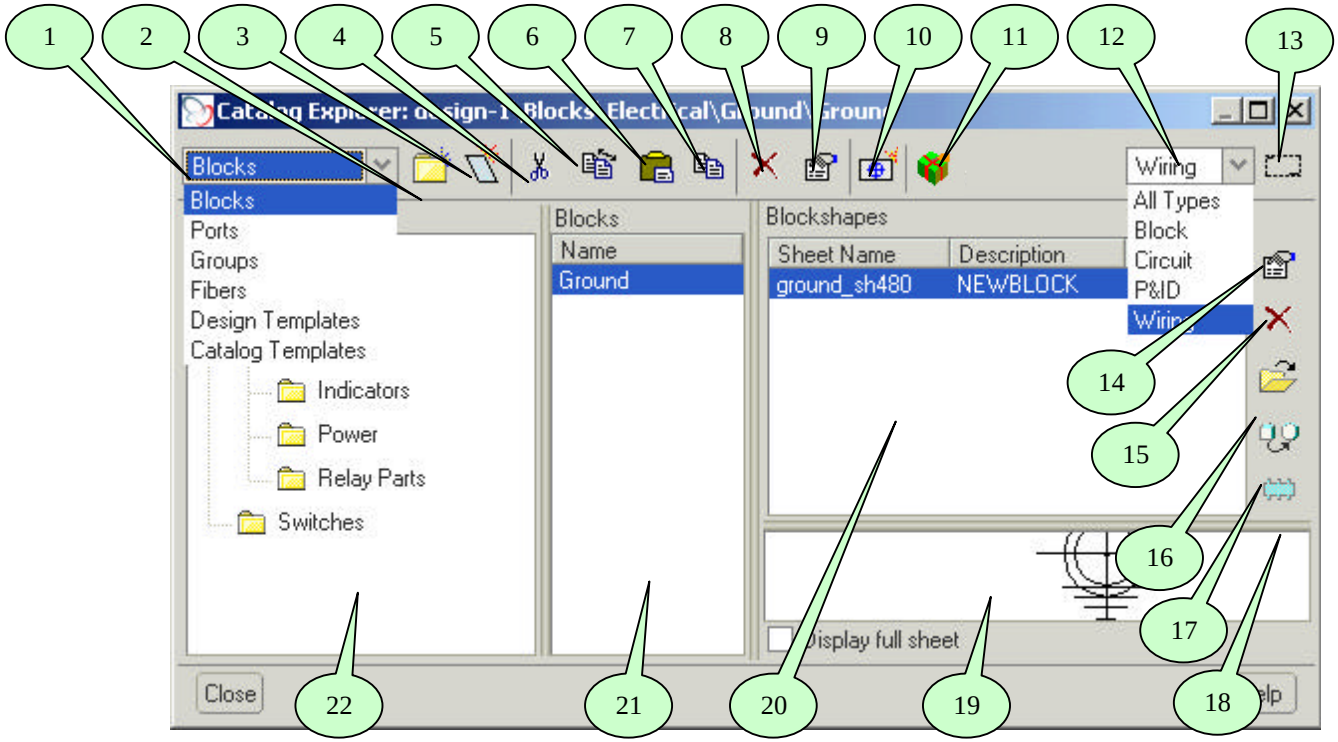
Defining Catalog Items

About the Catalog Explorer

The Catalog Explorer is the tool for creating and adding design element definitions to catalogs. These defined elements can then be added to the design.
In the Catalog Explorer, element definitions are classified in a hierarchy of folders. You can add or delete folders, or cut, copy, and paste elements between folders.
All designs have a catalog, which is an integral part of the design. A new design has an empty catalog. Once you start creating items in the catalog, these items are automatically stored with the design. You must have a design open during this process.

The Catalog Explorer icons

1. Catalog filter
2. Creates a new folder
3. Creates a new prototype artifact.
4. Cuts either a folder or a prototype artifact
5. Links either a folder or a prototype artifact
6. Pastes either a folder or a prototype artifact
7. Copies either a folder or a prototype artifact
8. Deletes either a folder or a prototype artifact
9. Properties of either a folder or a prototype artifact
10. Creates a new prototype shape that represents the currently Selected prototype artifact.
11. Package Tool
12. The Selection filter defaults to the current Open sheet type (e.g. wiring) and filters out components of types that do not match.
13. Sets the default template settings
14. Properties of prototype shape
15. Deletes prototype shape
16. Opens prototype shape for editing
17. Updates instanced prototype shapes
18. Instancing tool
    - o  create an instance of a block
    - o  create an instance of a port
    - o  create an instance of a group
    - o  create an unconstrained instance of a group
    - o  create an instance of a fiber
    - o  create an instance of a cable
19. Preview
20. Artifact shapes
21. Artifact list
22. Folder structure

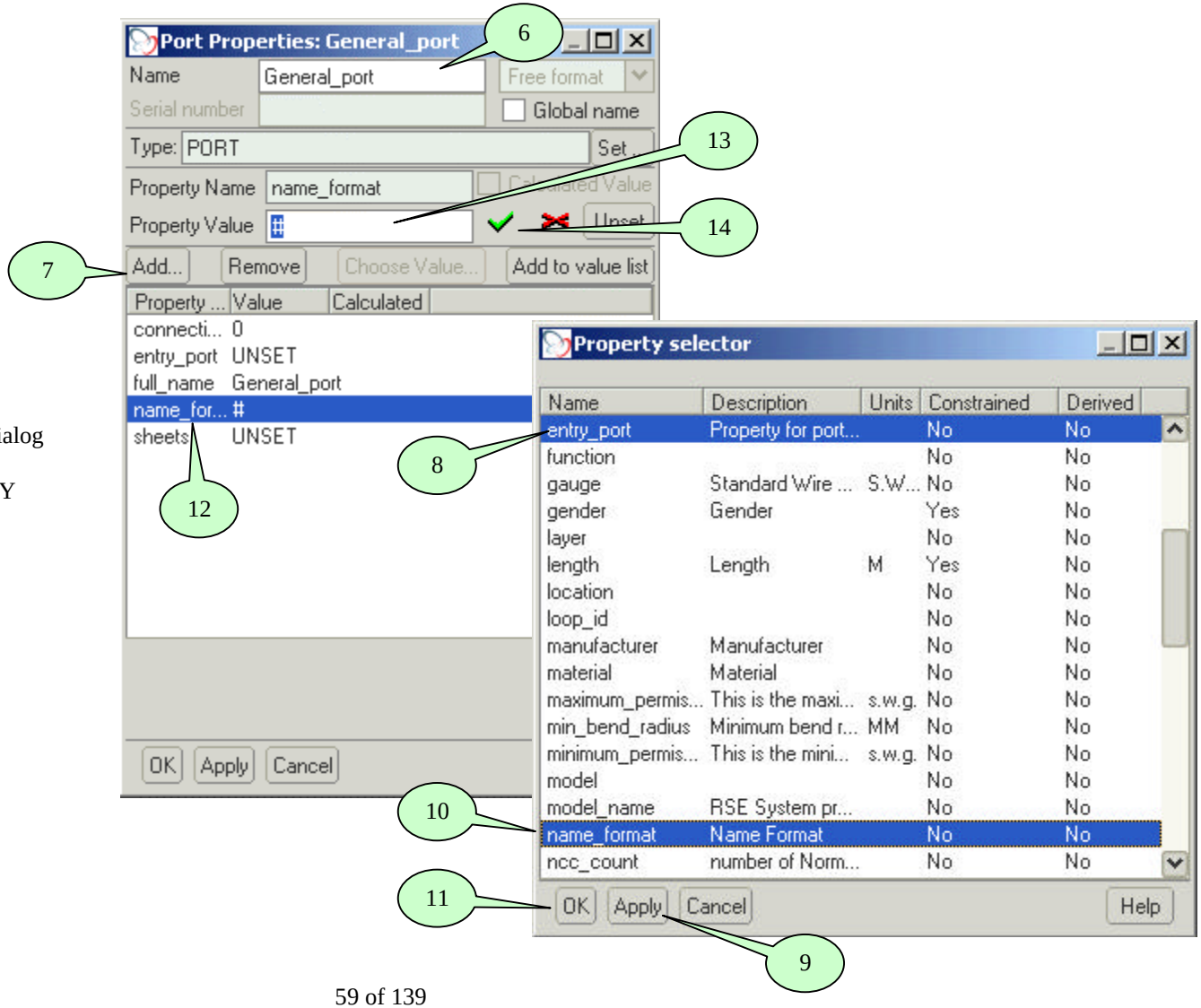


Defining a port artifact

An artifact is a block, group, fiber or port that can have properties. It is graphically represented on a sheet as a shape. An artifact need not always be represented on a sheet. For example, fibers created using the auto wiring dialog box do not have shapes, and so, do not appear on any sheet

A port is a terminal or pin that can be connected to zero or more ports. A port always has a parent or an owner. The parent can be a block, a group, or a fiber.
A port can connect to any port irrespective of the parent. For example, you can connect a port on a block to a port on another block, a port on a fiber, or a port on a group. A port from a block, a port from a fiber, and a port from a group can all be connected together in a single connection.
Ports are of two types, rigid and scaleable. A rigid port does not scale when the parent is scaled. A scaleable port scales when the parent is scaled.

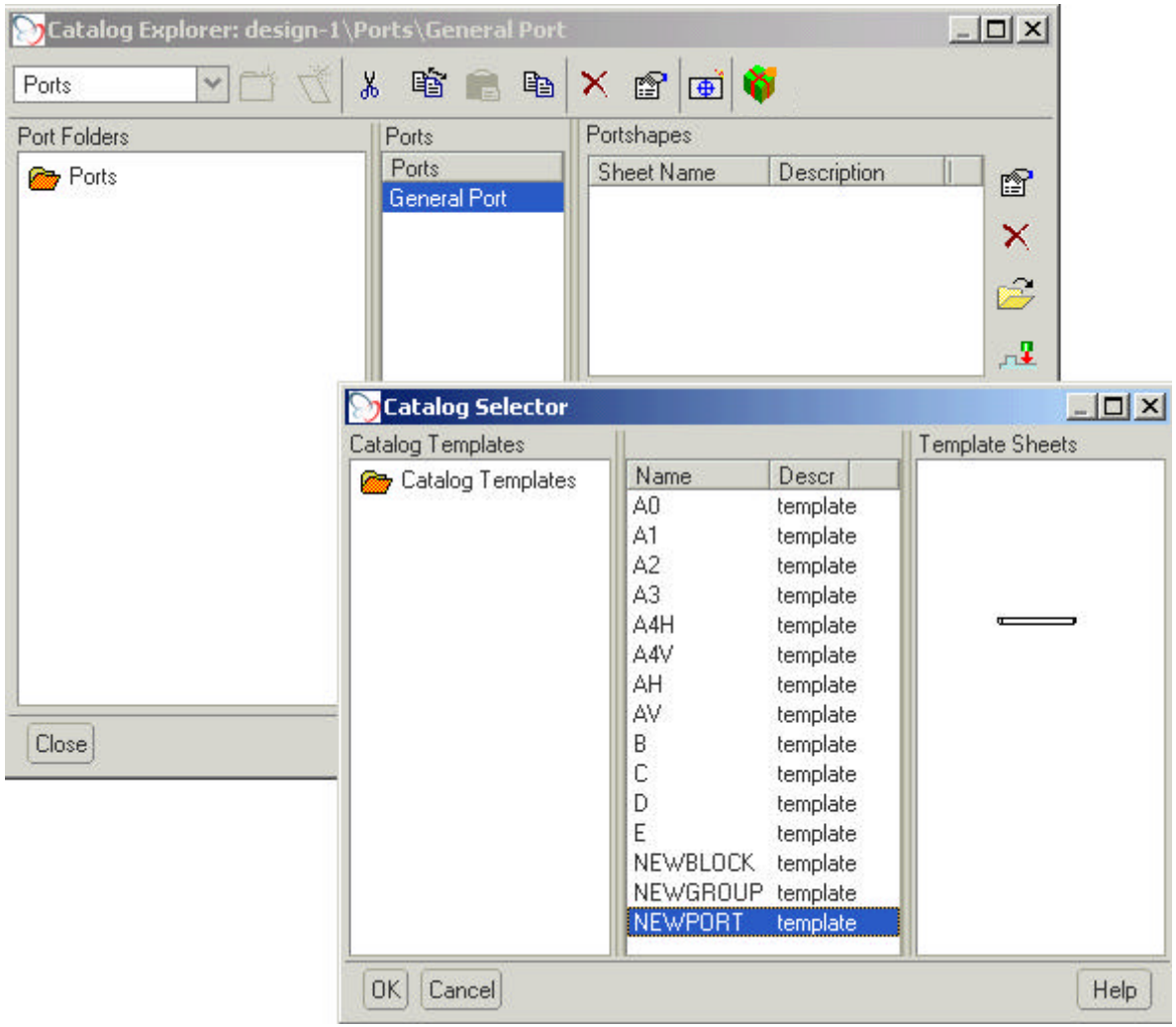
1. **Open** the Catalog Explorer.
2. **Select** Ports from the pull down menu.
3. **Select**  to create a new Port Artifact.
4. **Select** the newly created Artifact PORT_1
5. **Click** RMB > **Select** Properties
6. **Set** the name to be General_Port
7. Select Add
8. **Select** entry_port from the property Selector Dialog
9. From the property selector dialog **Select** APPLY
10. **Select** name_format
11. From the property selector dialog **Select** OK
12. **Select** name_format
13. In Property Value **Enter #**
14. **Select**  to confirm the entry > **Select** OK.



Defining a port shape

A shape is a graphical representation of an artifact on a sheet. A block shape represents a block, a group shape represents a group, a fiber shape represents a fiber, and a port shape represents a port.
A shape has a prototype and an instance.

15. **Select**  to create a new port shape.
16. **Select** NEWPORT for the shape template.
17. From the Catalog selector dialog **Select** OK
18. **Select**  to Open the port shape for editing
19. **Close** the Catalog Explorer



Grids

About Grid Icons

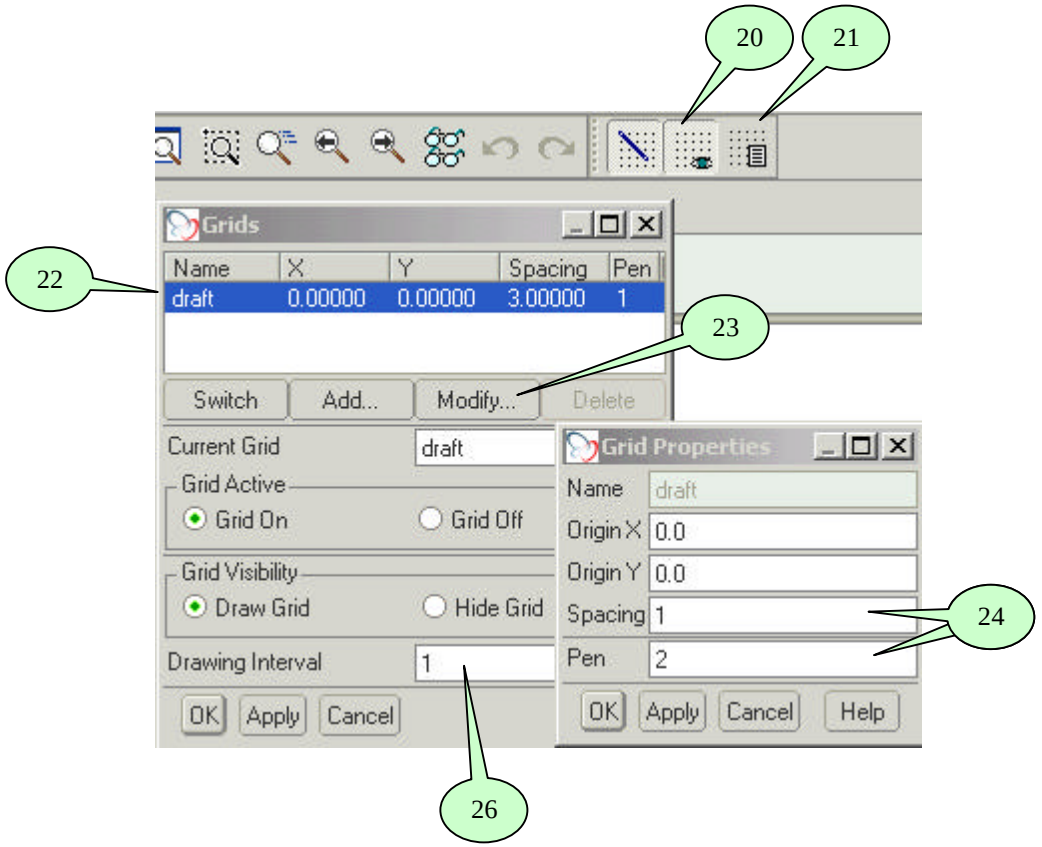
To modify the grid properties select from the main toolbar.

To view the grid select from the main toolbar.

To snap to the grid select from the main toolbar.

Setting the grid

20. **Select** to display the grid.
21. **Select** to open the grid dialog.
22. **Select** draft.
23. **Select** Modify.
24. **Set** the grid Spacing to 1 & the Pen to 2.
25. From the grid properties dialog **Select** OK
26. **Set** the grid Drawing Interval to 1
27. **Select** OK to exit the Grid Dialog



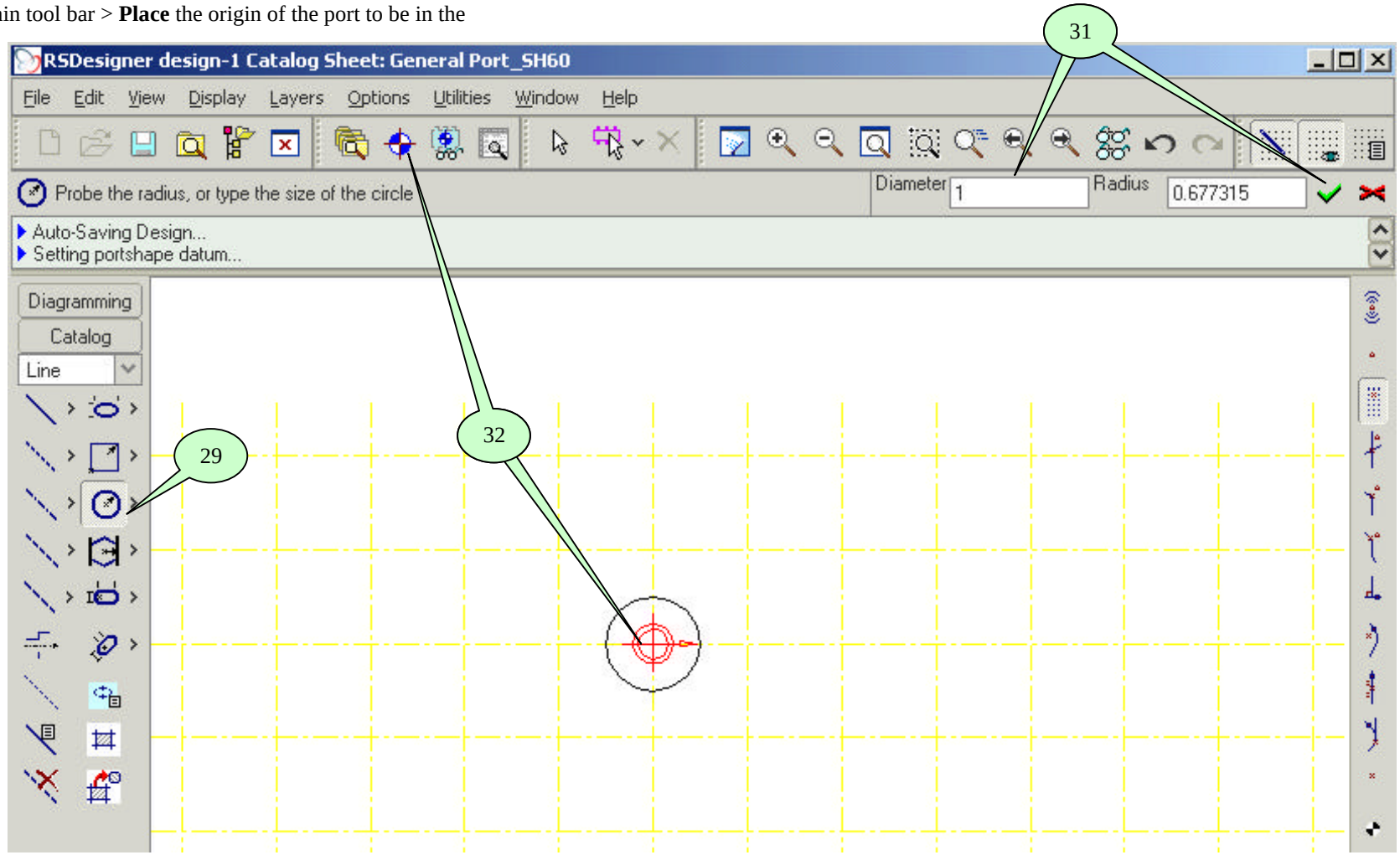
28. Select the Line tool tray

29. **Select**  to draw a circle

30. **Probe** with the LMB on the sheet

31. **Enter** 1 for the diameter > **Select**  to confirm the entry.

32. **Select**  from the main tool bar > **Place** the origin of the port to be in the centre of the circle

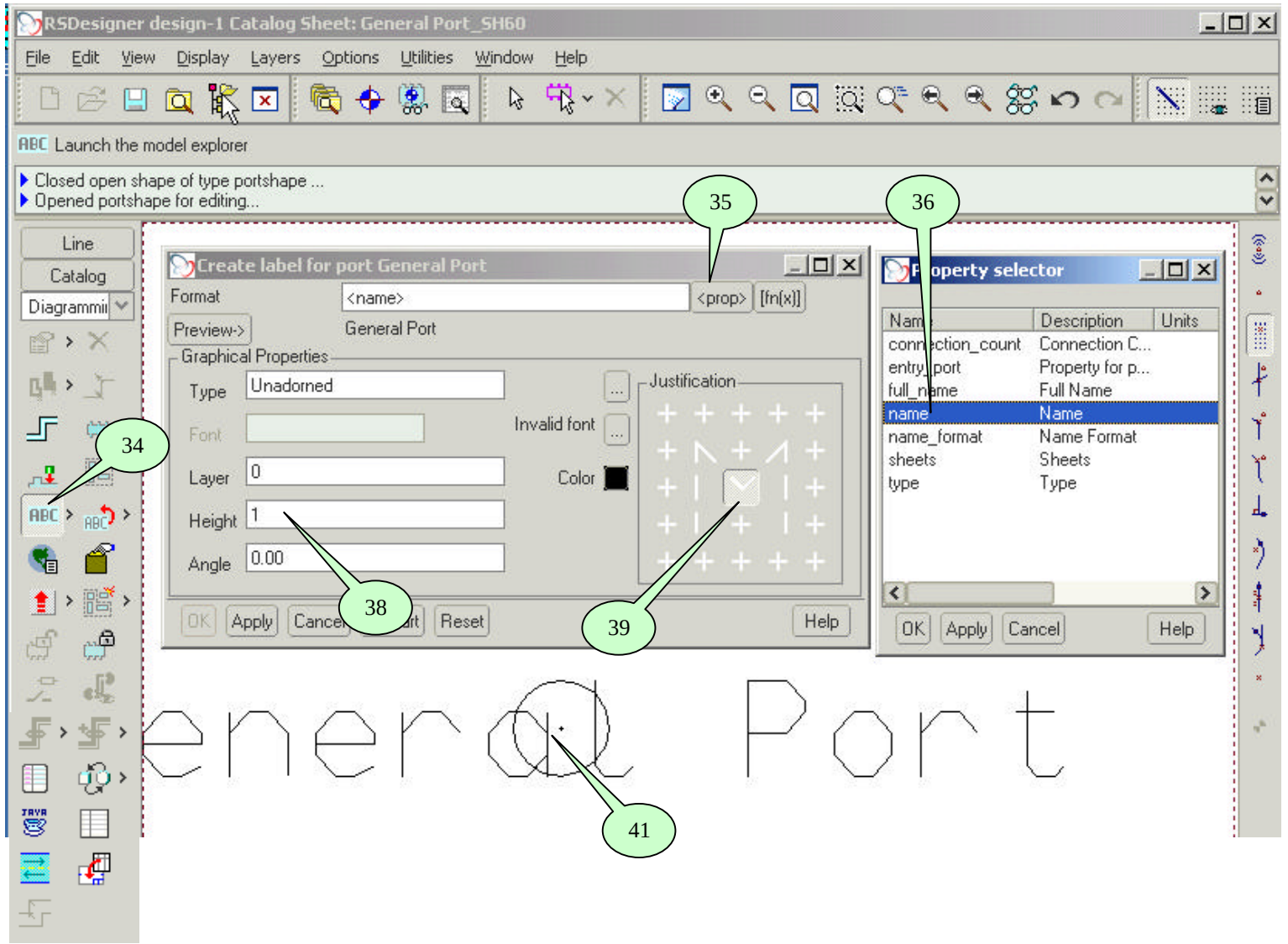


Creating Port Labels

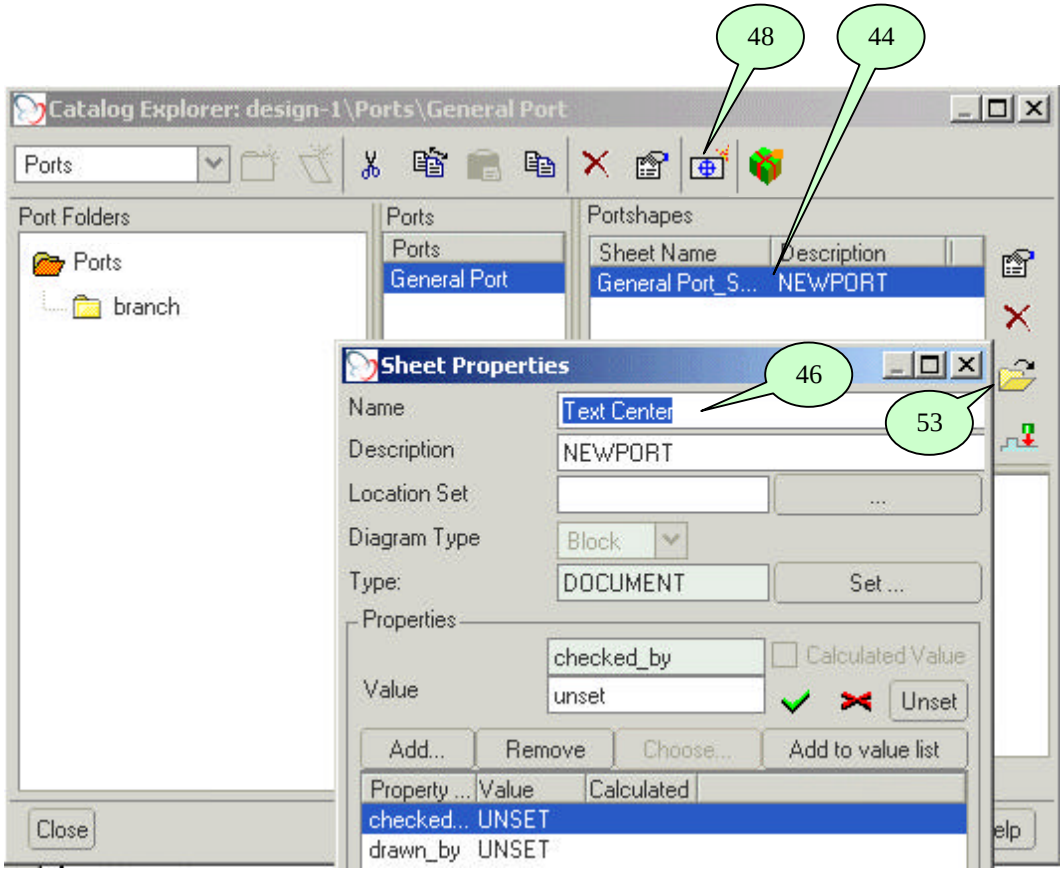
Use labels to display text for blocks, ports, or fibers on a sheet. The label can display values of properties of the object it is associated with, as well as those of associated objects. A configurable format string governs the content of the text that is displayed in the label. You can add labels to catalog definitions, objects, and sheets.

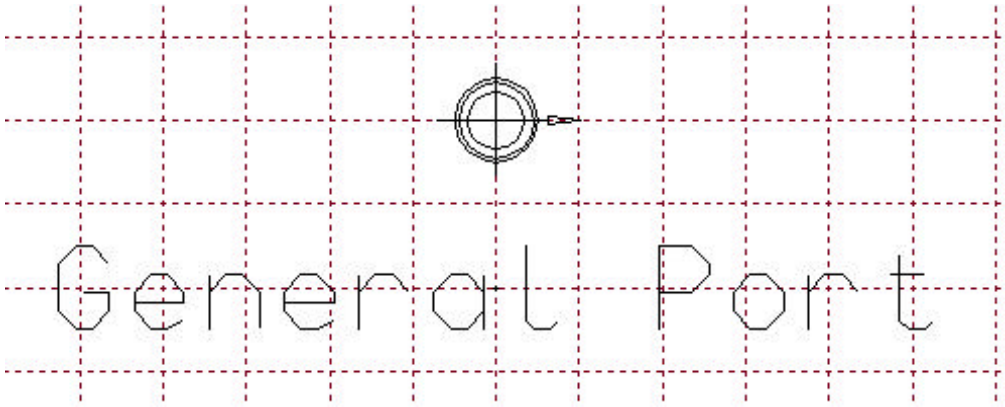
Labels can be configured as either rigid or scalable. Rigid labels retain their original shape when their parents are scaled. Scalable labels scale along with their respective parent components. All labels are rigid by default

33. Select the Diagramming tool tray.
34. **Select**  to add a label.
35. Select  .
36. **Select** name from the property selector dialog.
37. From property selector dialog **Select** OK
38. **Set** the Height to be 1.
39. **Set** the text justification to be centre.
40. Select Apply
41. **Place** the text in the selector of the circle.
42. **Select**  to close the shape.



43. **Open** the Catalog Explorer
44. **Select** the new port shape
45. **Click** RMB > **Select** Sheet Properties.
46. **Set** the sheet name to Text Centre
47. From the sheet properties dialog **Select** OK.
48. **Select** the  to create a new shape
49. **Select** NEWPORT for the shape template
50. From the catalog selector dialog **Select** OK
51. **Select** the new port shape > **Click** RMB > **Select** Sheet Properties**.**
52. **Set** the sheet name to Text Bottom > **Select** OK
53. **Select**  to open the new shape.
54. **Close** the Catalog Explorer.
55. **Select**  to edit the grid properties.
56. **Set** the grid spacing to 1 & the pen to be 2.
57. **Select** the Line tool tray and draw a circle with a diameter of 1.
58. Select  from the Diagramming Tool Tray to add a label.
59. Ensure format is Set to <name>
60. **Place** the label below the circle
61. **Select**  from the main tool bar > Place the origin of the port to be in the centre of the circle.
62. **Select**  to close the shape.

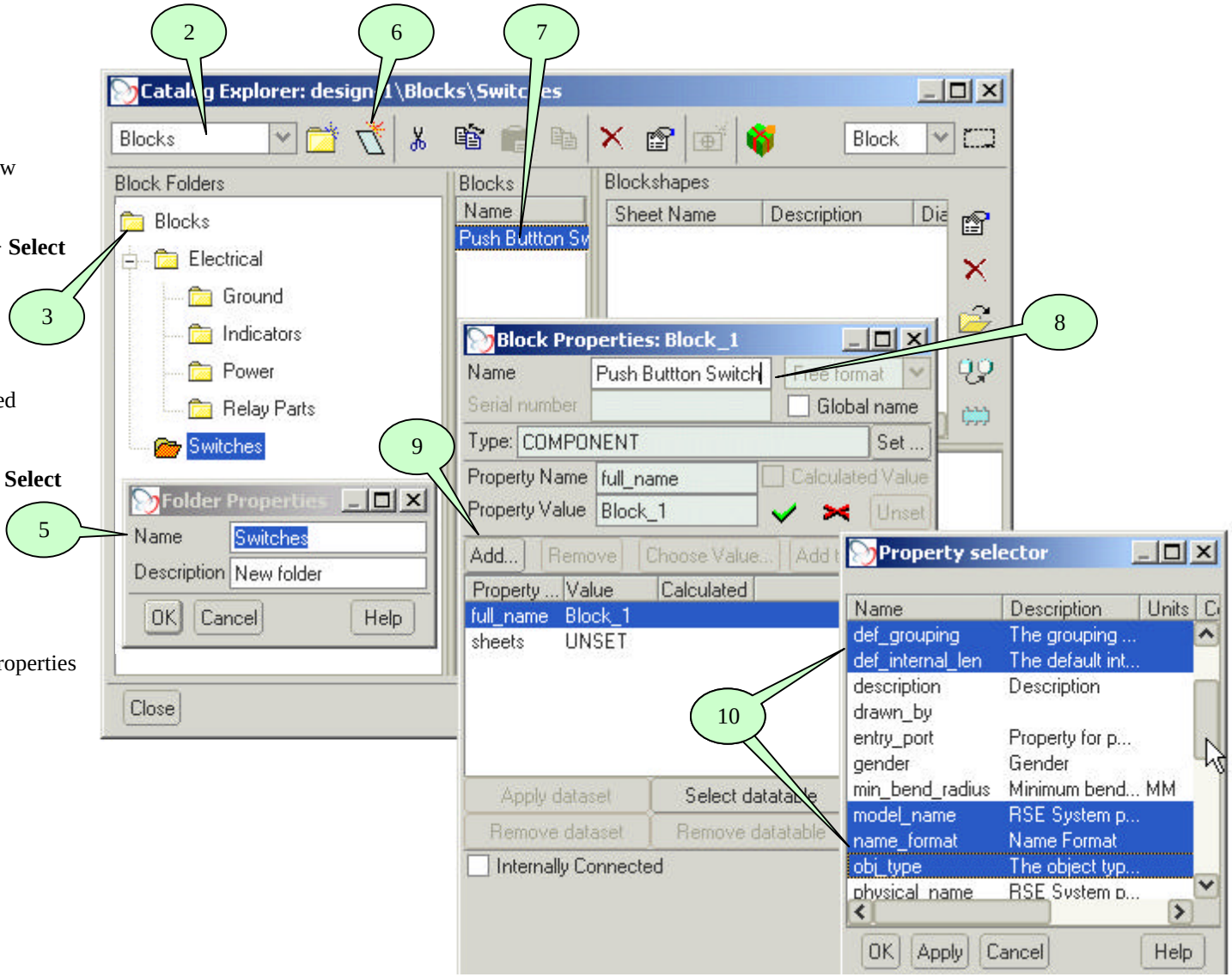




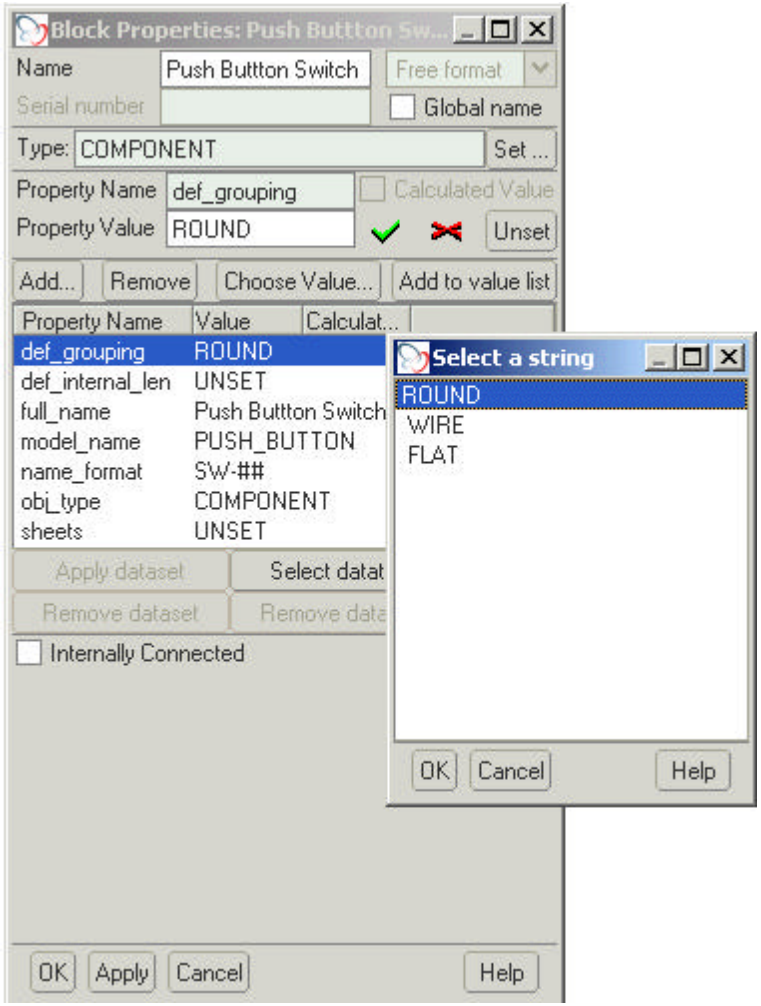
Defining a Block Artifact

A block is an object that can contain ports. The ports can accept connections

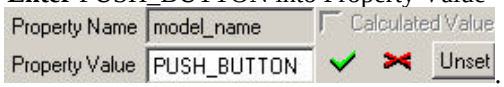
1. **Open** the Catalog Explorer.
2. **Select** Blocks from the pull down menu.
3. **Select** folder Blocks > **Click** RMB > **Select** New Folder.
4. **Select** the newly created folder > **Click** RMB > **Select** Properties
5. **Set** the folder name to Switches.
6. **Select**  to create a new block in the switched Directory
7. **Select** the newly created block > **Click** RMB > **Select** Properties**.**
8. **Change** the name to Push Button Switch.
9. **Select** the Add button
10. Using the **CTRL** button **Select** the following properties
    - def_grouping
    - def_internal_len
    - model_name
    - name_format
    - Object_type
11. **Select OK** from the property selector

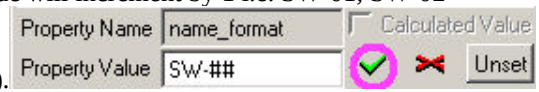


Modifying the artifact properties



12. **Select** property def_grouping.
13. **Select** Choose Value.
14. From the Select a string dialog **Select** ROUND.
15. From the Select a string dialog **Select** OK.
16. **Select**  to confirm the entry.
17. **Select** property def_internal_len
18. **Enter** 0 into Property Value 

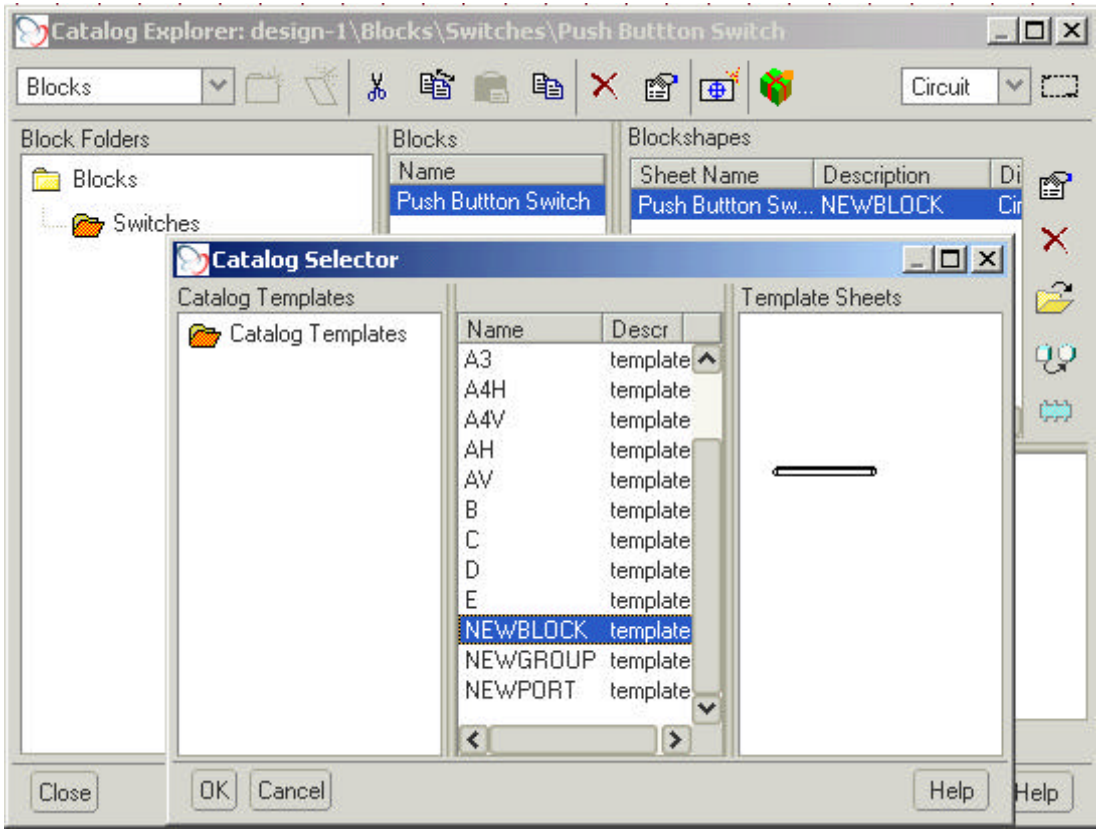
19. **Select**  to confirm the entry.
20. **Select** property model_name
21. **Enter** PUSH_BUTTON into Property Value


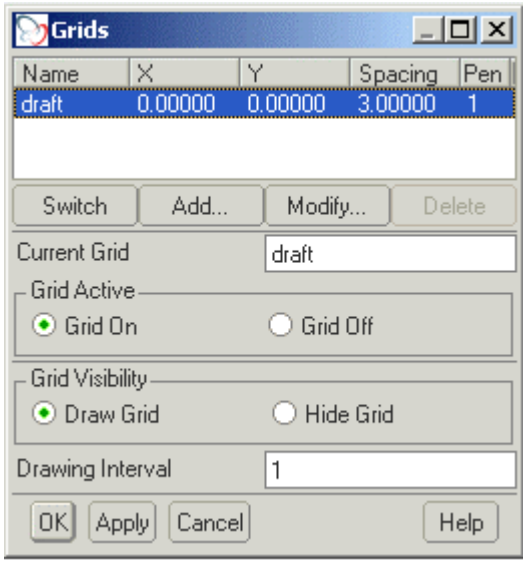
22. **Select**  to confirm the entry.
23. **Select** property obj_type.
24. **Select** Choose Value.
25. From **Select** a string dialog **Select** COMPONENT > OK
26. **Select**  to confirm the entry
27. **Set** property name_format to be SW-## (Each time the switch is instanced the value will increment by 1 i.e. SW-01, SW-02 etc).


28. **Select**  to confirm the entry
29. **Select** OK to close the block properties dialog.

Creating a Circuit shape

1. **Set** the diagramming filter to Circuit**.**
2. **Select**  to create a new shape.
3. From the Catalog selector **Select** catalog template NEWBLOCK**.**
4. From the Catalog selector **Select** OK.
5. **Select**  to open the new shape for editing.
6. **Close** the Catalog Explorer.
7. **Select**  to edit the grid Properties.
8. **Set** the drawing interval to 1
9. From the grid dialog **Select** OK.





Creating Lines

Before we can define the shape we need to understand how to create and edit lines within Routed Systems Designer.

About The Line Tool Tray

 Thin, medium & thick solid lines

 Thin, medium & thick dotted line

 Thin, medium & thick chain lines

 Thin, medium & thick short dash lines

 Thin, medium & thick long dash lines

 Centre lines

 Construction lines

 Lines of any type

 Delete construction lines

 Ellipses using working line type and base angle

 Boxes using working line type and base angle

 Circles using working line type and base angle

 Polygons using working line type and base angle

 Slots constrained to 0/90 degrees from base angle using working line type

 Free slots using working line type

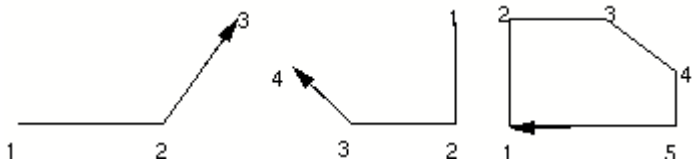
 Query or change properties of **Close**d geometry tools

 Create hatching

 Modify Selected hatching

About Creating a Line

A line should be thought of as a sequence of points, beginning with point1. When you create a line, the sequence in which you specify the points defines the direction of the line. This is shown below. The current point marker indicates the direction of the lines.



Line Direction

Knowing the direction of a line is vital when you want to Add a new point to the line as it affects the result that you achieve.

Positioning the cursor at the first point in the line and clicking LMB creates the first point of a line. The next point in the line is created at the second position of the cursor and a line is drawn between the first point and the new point. As you work you can use any of the edit line tools, you can adjust the XY-coordinates of each point by choosing Move Point or Line Point Properties... from the popup menu, you can also change the point function by choosing either Line Point Properties from the popup menu or Point Functions from the Utilities menu

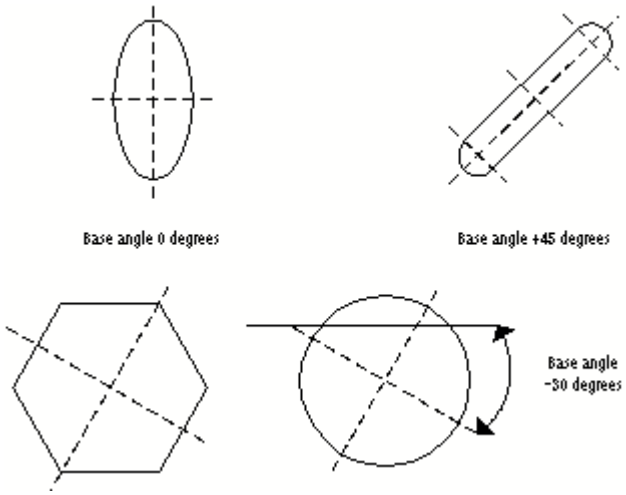
When you have finished, you can choose New Line from the popup menu to draw another line, or choose Exit Tool to leave the line Selected ready for the next operation. If you choose none of these, then any further points you create will be joined to the existing line.

When creating closed geometry, you can also specify the following properties for the closed geometry tools:

- Working line type
- Centerline and the amount of overlap beyond the boundary of the geometry
- Base angle (not free slot tools)

About Base angle

The base angle, **Set** to +0.00000 degrees by default, determines the angle of **Close**d geometry elements to be drawn on the sheet as shown below.



Elements Created Using Different Base Angles

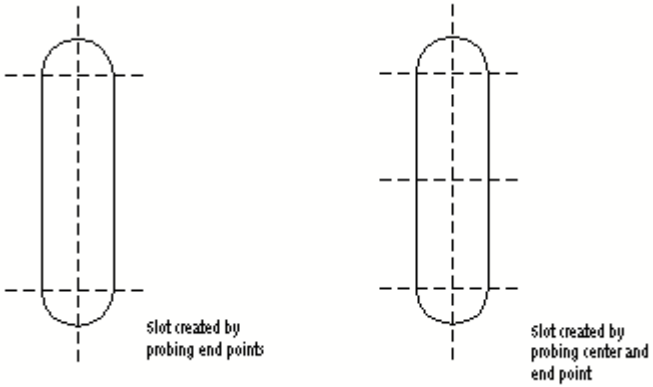
You can change the angle either by:

- Choosing Base Angle from the popup menu and entering a new angle in the Angle field. The Closed geometry element you are currently creating will use this base angle, as will any subsequent element while the tool remains active.
-  Choosing the Closed geometry properties tool to display the Closed Geometry Properties box, typing a new angle (either positive or negative) in the Base angle field, and then clicking OK**.** (This sets it for all the closed geometry tools.)

Please note: You can prefix the angle with either a plus or a minus sign (optional).

About Centerlines and centerline overlap

You can choose to display centerlines on the closed geometry that you create. The centerlines indicate the horizontal and vertical axis of the geometry rotated by the amount of the base angle. For the slot tools, the centerlines also pass through the center of the round-slot ends as shown below



Displaying Centerlines on Slots

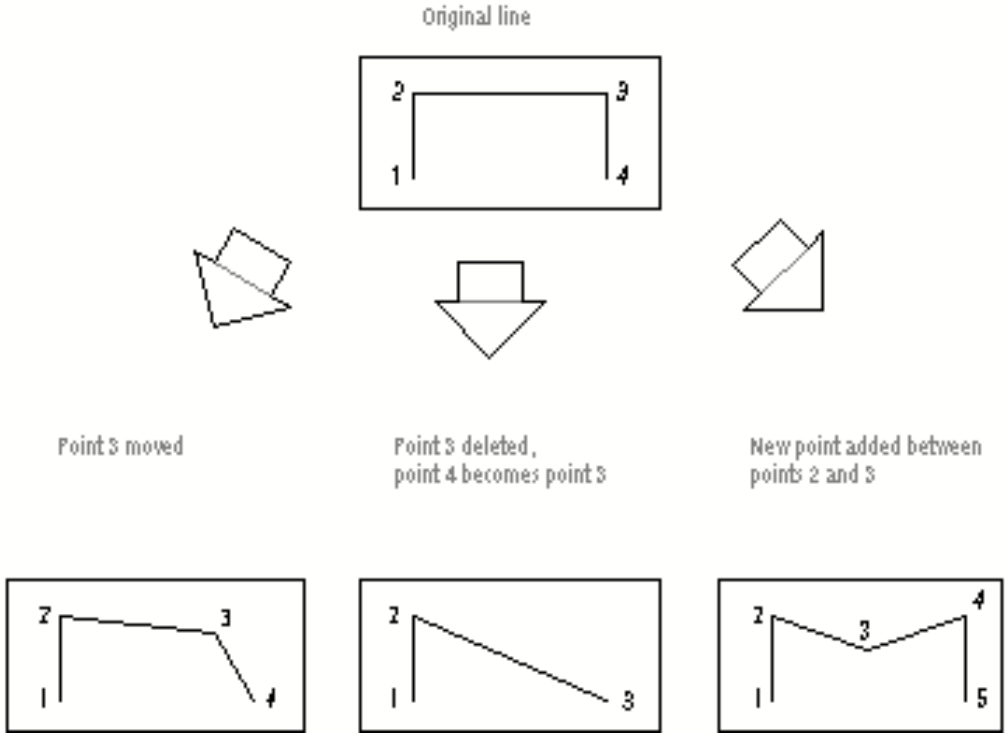
You can specify whether centerlines are displayed either by:

- Choosing Switch Centerlines **ON/OFF** from the popup menu. Centerlines will be displayed, or not displayed, for the next closed geometry element you create with the current active tool only.
-  Choosing the Closed geometry properties tool to display the Closed Geometry Properties box, clicking the Add Centerlines at time of creation button, and then clicking OK. Centerlines will be displayed, or not displayed, for all subsequent closed geometry elements you create.

The Centerlines overlap determines the amount by which the Centerlines extend beyond the boundary of the element. You can query or change the overlap in the Closed Geometry Properties box.

About Editing a Line

You can make changes to a single point, to one or more line segments, or to the whole line. For example, you can edit a line by adding a point to the line, deleting a point from the line, or moving a point as shown below.



Changing Line Shape by Editing a Line

 You can edit a line at any time; during or after its creation, when the icon, shown opposite, is displayed in the status area.
You can use the items on the popup menu for basic line editing or the tools in the Edit Line tool tray for more complex editing tasks

Procedure for editing a line

Select the line for editing using one of the following methods:

- Choose the Select tool and then with the cursor positioned in the presentation area:
- Move the cursor to the line you want to edit.
- Click the SELECT mouse button.
- Choose Edit from the popup menu. The current point marker appears against one of the points in the line. If there is a coordinate displayed in the X and Y Coord fields, the current point marker appears against the point nearest to this. If there is no coordinate, then it appears at the first point.
- Move the cursor to the point in the line that you want to edit, then double Click the SELECT mouse button.
- Routed Systems Designer displays the icon, shown opposite, in the status area to show that you are now editing the Selected line.

Choose one of the following menu items from the popup menu or any of the edit line tools.

- **New Line** ends the line you are currently creating and leaves you ready to start a new line.
- **Close Line Close**s the first and last points in the line with a line segment. The last point in the line remains current.
- **Close and New Line** creates a line segment between the first and last points in the line, then **Close**s the line, and ends it leaving you ready to start a new line.
- **Move Point** enables you to move the current point to a probed position.
- **Delete Point** deletes the current point in a line.
- **Flip** reverses the line function. For example, changes a clockwise arc to counterclockwise arc.
- **Reverse Points in Line** reverses the direction of the points in a Selected line, that is the first point in the line becomes the last point.
- **Undo** undoes any changes that you have made and returns the line to its original state.
- **Redo** undoes the result of choosing Undo.
- **Line Point Properties** calls up the Line Point Property box for the Selected line.
- **Exit Tool** enables you to exit from the active tool and automatically choose the Select tool you used last**.**

Complex line editing

Other changes that you can make to a line include:

- Adding an arc to a line segment
- Changing a line segment to a specified angle
- Changing the length of the line segment

You carry out these types of tasks using the edit line tools

About the Edit Line Tool Tray

 Extended segment tools

 Fit curve through points

 Divide segment

 Create fillet

 Rotate segment tools

 Create tangent point arcs

 Reverse points in line

 Move point

 Modify tangent arcs

 Create tangent segments

 Extend segment tools

 Orthogonalize segments or line

 Split or join segments

 Create chamfer

 Rotate segment tools

 Circular arcs

 Show or hide segment

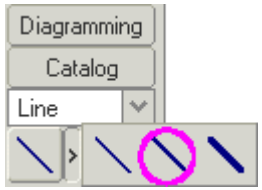
 Delete point

 Convert circular arcs

 Trace lines

Defining a circuit shape

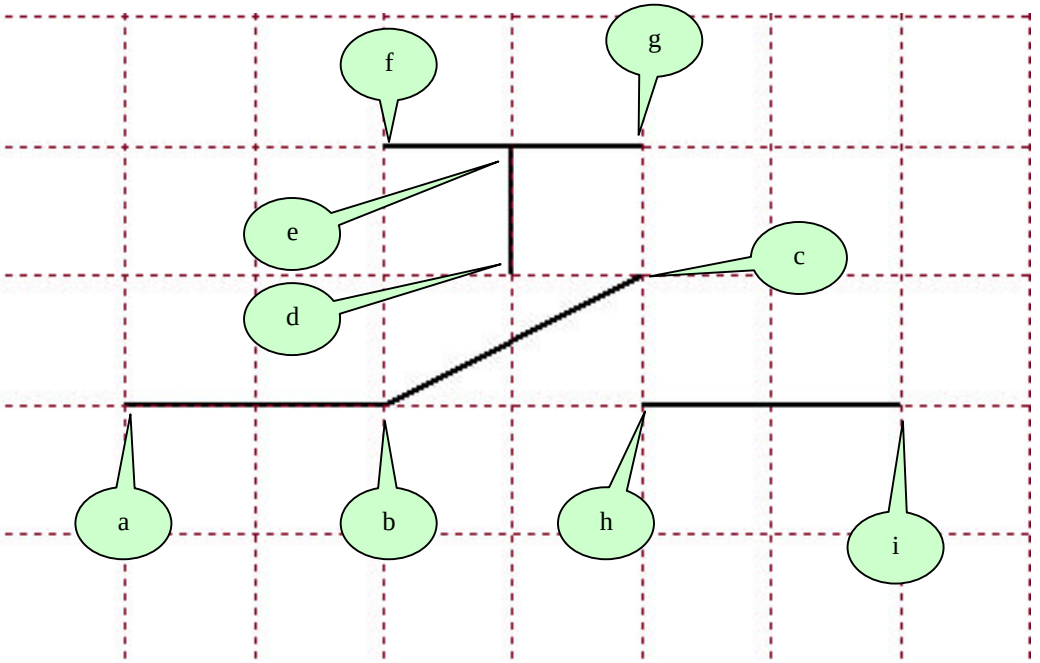
10. **Select** the **Line** tool tray.



11. **Select** Medium solid line 

12. Draw the four lines

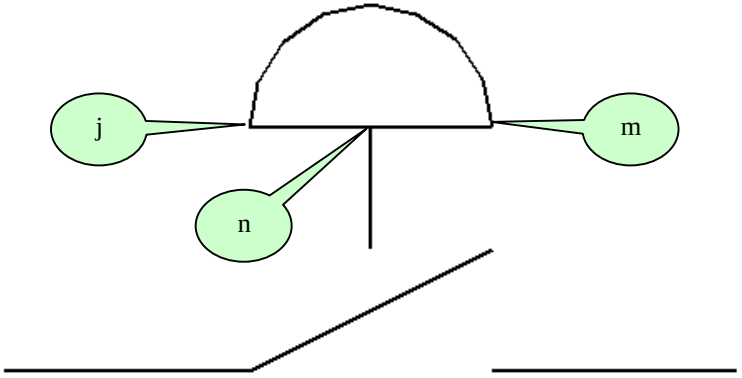
    a) **Click** LMB
    b) **Click** LMB
    c) **Click** LMB > **Click** RMB > **Select** New Line
    d) **Click** LMB
    e) **Click** LMB > **Click** RMB > **Select** New Line
    f) **Click** LMB
    g) **Click** LMB > **Click** RMB > **Select** New Line
    h) **Click** LMB
    i) **Click** LMB > **Click** RMB > **Select** Exit Tool



13. **Select** Medium solid line

14. **Draw** the curve

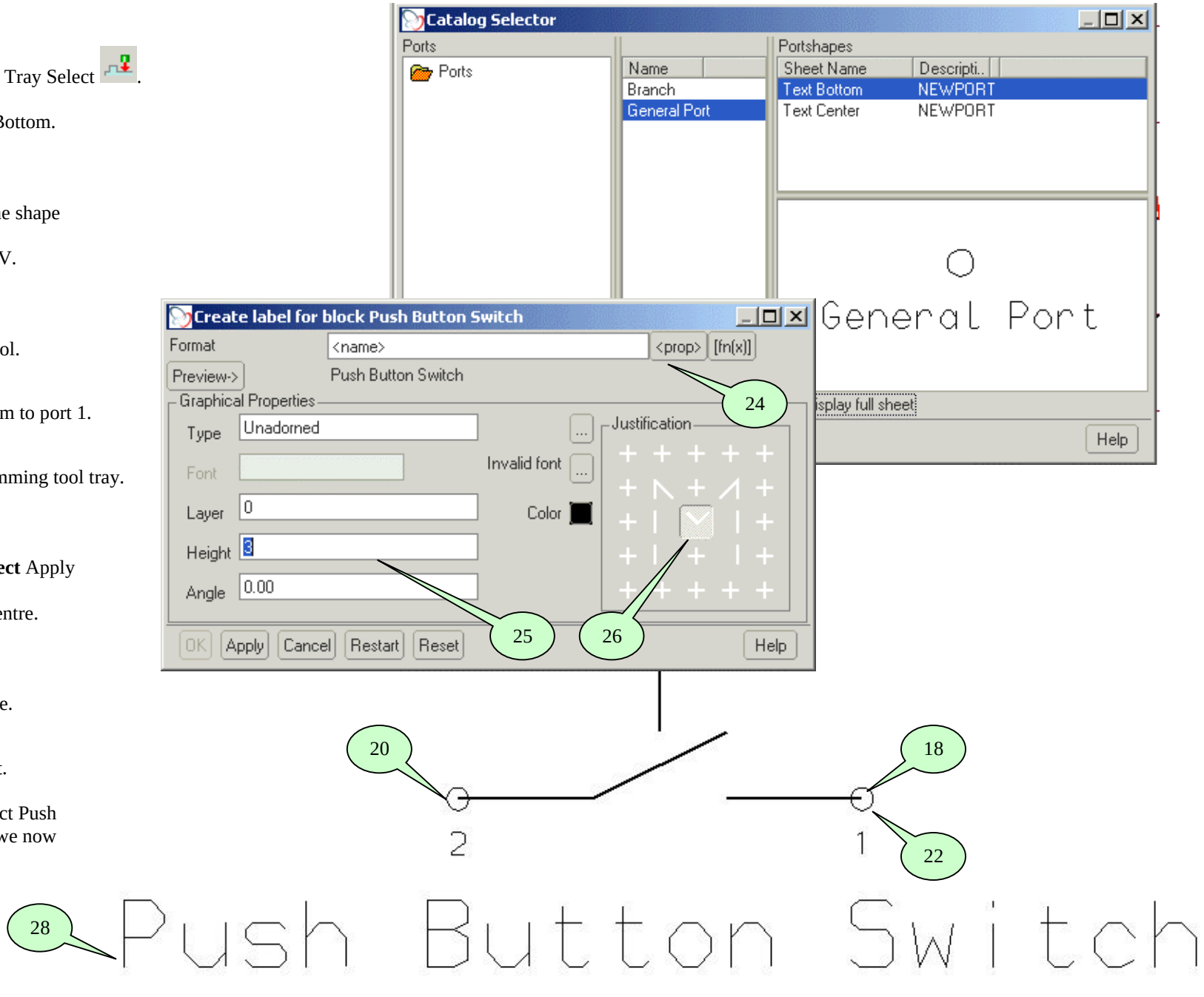
    j) **Click** LMB
    k) **Select Edit Line Tool Tray**
    l) **Select**  to create an Arc
    m) **Click** LMB
    n) **Click** LMB > **Click** RMB > **Select** Exit Tool



Placing Ports & Labels

15. From the Diagramming Tool Tray Select .
16. **Select** General_Port > Text Bottom.
17. Select OK.
18. **Place** Port 1 to the right of the shape
19. **Click** RMB > **Select** Mirror V.
20. **Place** Port 2.
21. **Click** RMB > **Select** Exit Tool.
22. **Select**  and **Set** the Datum to port 1.
23. **Select**  from the Diagramming tool tray.
24. Select Property <name>
25. **Set** the text height to 3 > **Select** Apply
26. **Set** the text justification to centre.
27. Select Appy
28. **Place** the text under the shape.
29. **Select**  to close the sheet.

Now that we have created an artifact Push Button Switch and a circuit shape we now need to create a wiring shape.

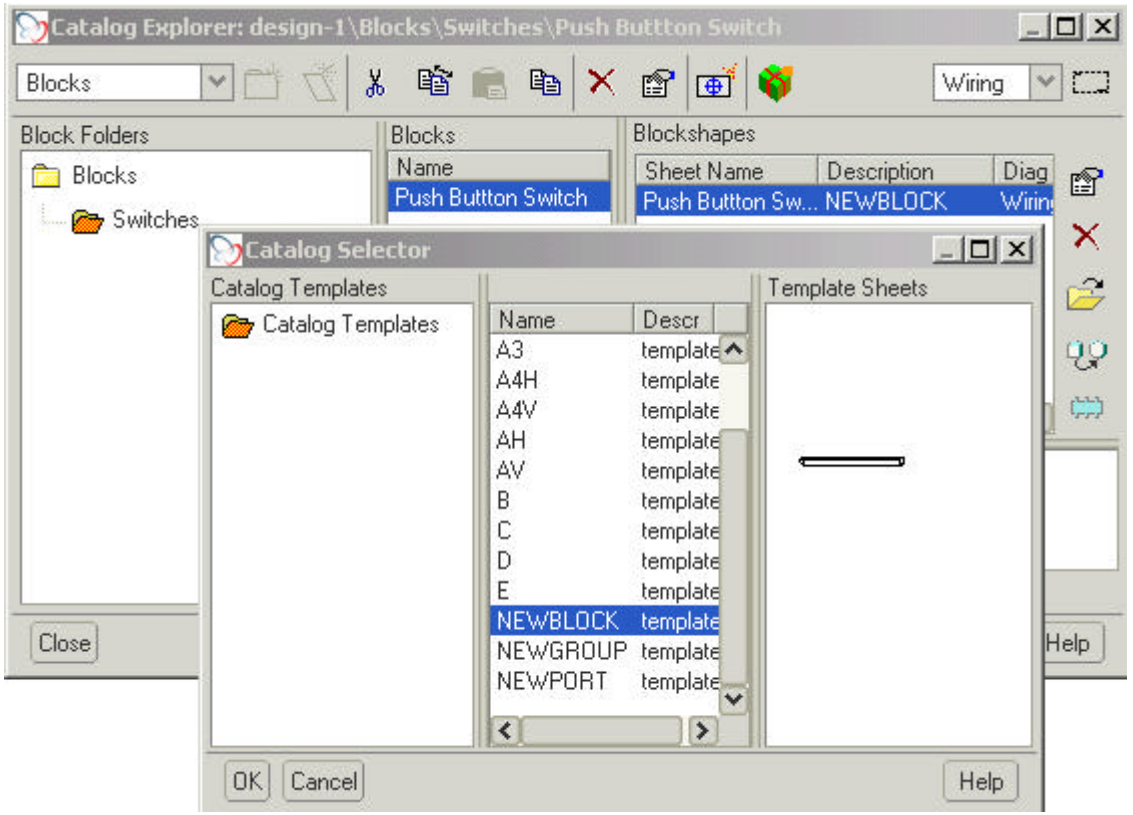


Creating a Wiring Shape

1. **Open** the Catalog Explorer.
2. **Select** Blocks > Push Button Switch.
3. **Set** the Diagramming filter to Wiring

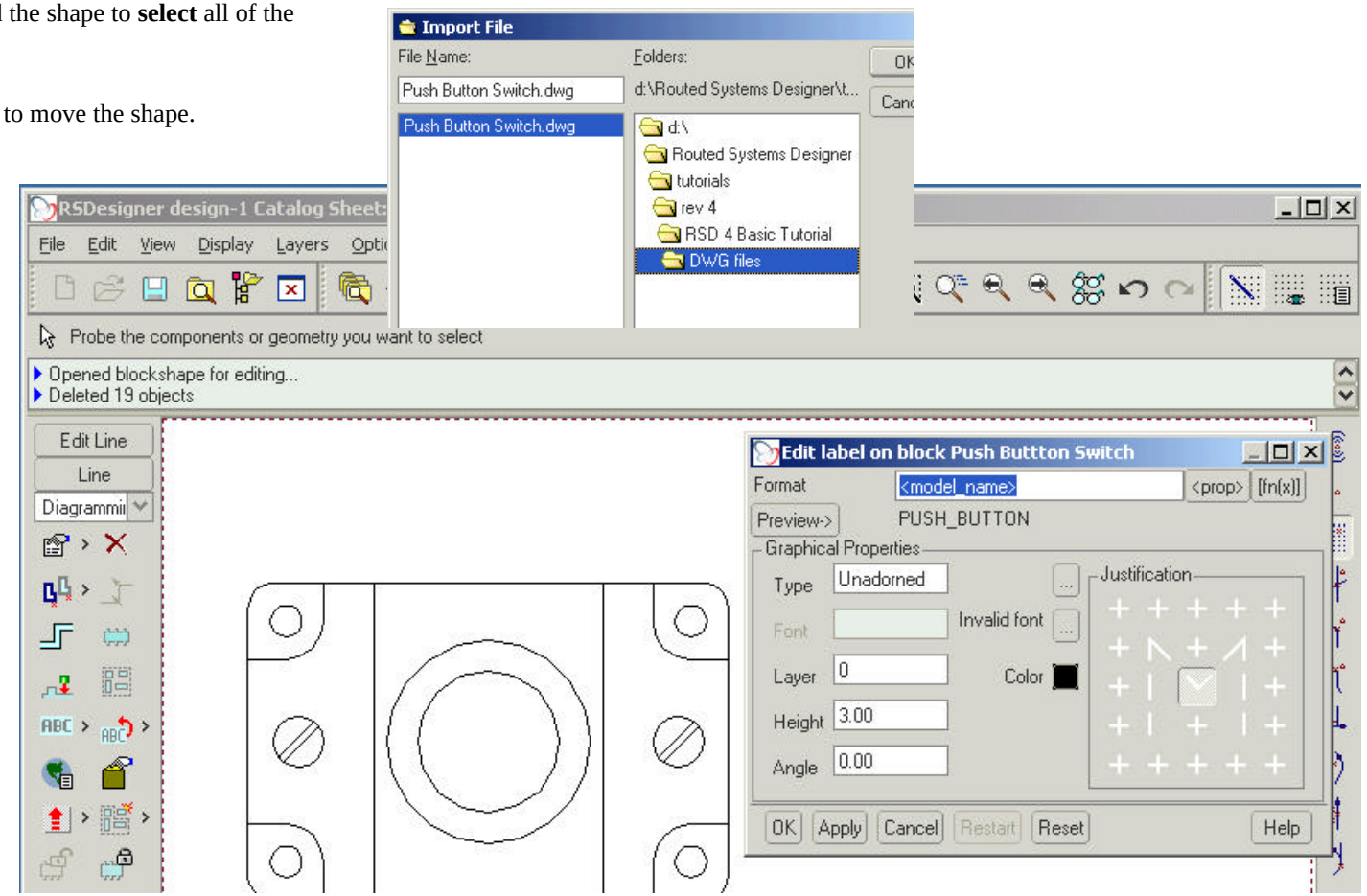
Notice that when the filter is set to wiring the circuit shape is hidden.

4. **Select**  to create new shape.
5. From the Catalog selector dialog **Select** template NEWBLOCK**.**
6. From the Catalog selector dialog **Select** OK.
7. **Select**  to open the new shape for editing.
8. **Close** the Catalog Explorer dialog.
9. **Select**  to edit the grid Properties.
10. **Set** the drawing interval to 1**.**



Utilizing DWG files

11. **Select** File > Import > DWG
12. From the DWG files folder **Select** Push Button Switch.dwg > **Select** OK
13. **Select**  to view the whole sheet > Zoom into Shape
14. **Click** and hold the LMB and make a box around the shape to **select** all of the imported geometry.
15. From the **Diagramming Tool Tray Select**  to move the shape.
16. **Select**  for Centre Probe
17. **Select** the circle in the centre of the shape
18. **Select** the nearest grid intersection (ensure that grid snap is on)
19. **Click** RMB > **Select** Exit Tool
20. **Click** RMB > **Select** De-Select all
21. **Select**  from the diagramming tool tray.
22. For the name format **Select** property name
23. **Set** the text height to 3
24. **Set** the text justification to centre
25. **Place** the label as shown
26. **Select**  from the diagramming tool tray.
27. For the name format **Select** property model_name.
28. **Place** the label as shown
29. **Click** RMB >**Select** Exit Tool



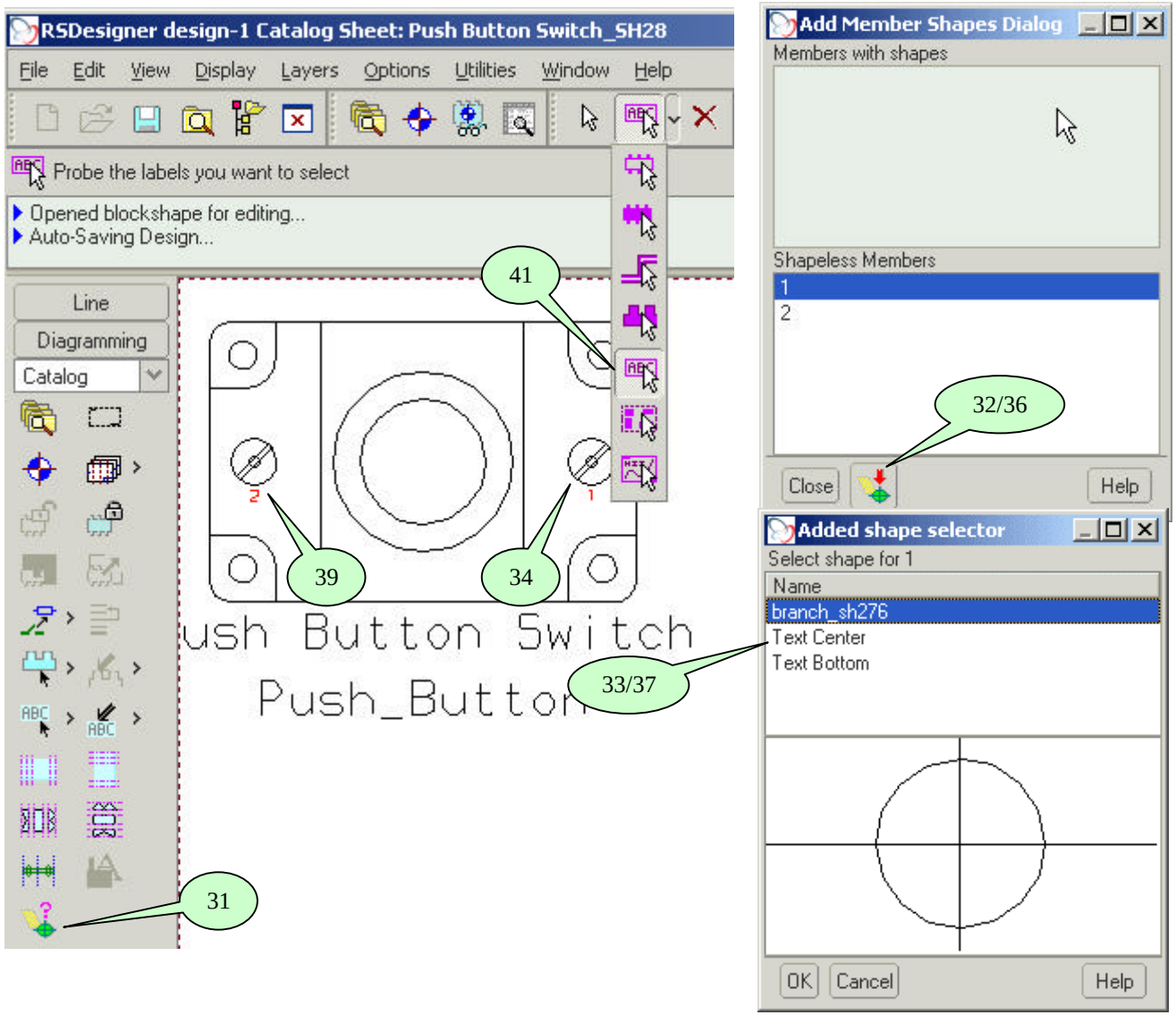
About Place Shape for Member

The Add Member Shapes dialog box displays in the top list the members that already have a shape and in the bottom list the members that do not have a shape.

Selecting a shapeless member from the bottom list and clicking  or double-clicking a shapeless member in the list displays a shape Selection viewer. Selecting a shape from the viewer allows the shapeless member to be instanced. Once instanced the member will move from the bottom to the top list as it is no longer shapeless. For ports, a shape can be selected from all the ports in the catalog. For groups and block members, you can select a shape of the Selected prototype only.

Placing Multi-Ports

30. **Select** the catalog tool tray
31. **Select** Place Shape for member 
32. **Select** Add shape 
33. Select Text Centre > Select OK
34. **Place** in the centre of the right hand circle
35. **Select** 2 from the shapeless members list
36. **Select Add** shape 
37. **Select** Text Centre > OK
38. **Click** RMB > **Select** MirrorV
39. **Place** in the centre of the right hand circle
40. **Close** the Add member Shapes Dialog
41. **Select**  for the Label Selection tool
42. Using MMB **Select** text 1 & 2
43. **Click** RMB > **Select** Move > **Move** the text below the geometry
44. **Select**  and **Set** the datum to be on port 1
45. **Select**  to close the sheet.



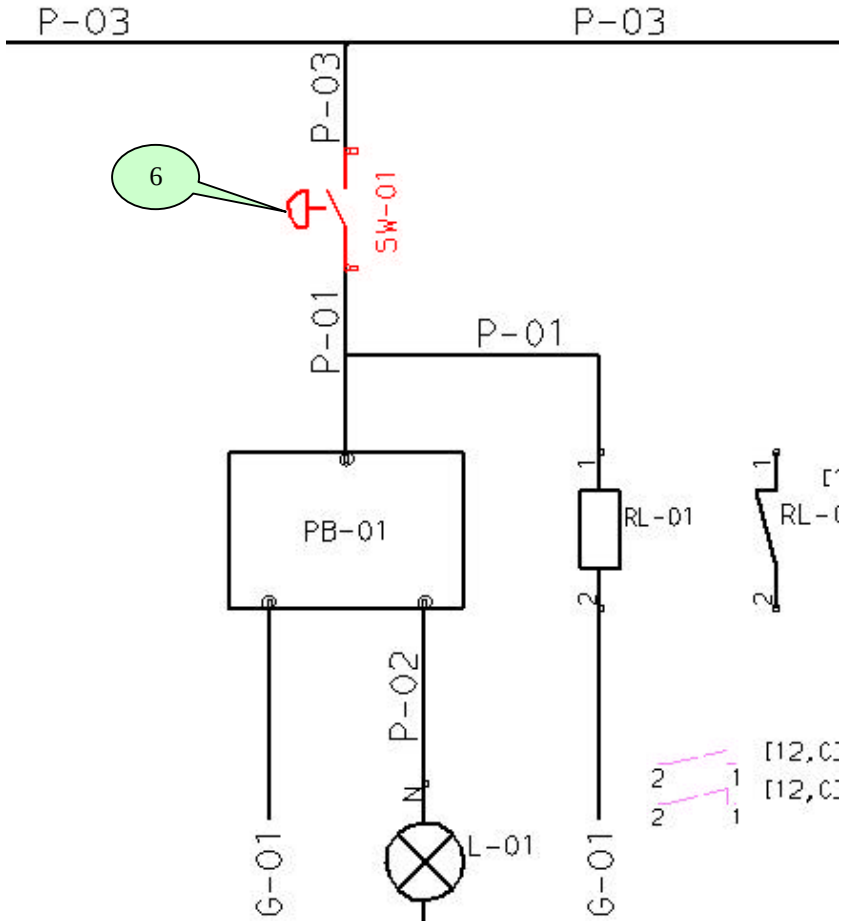
Placing the new circuit shape

Now that we have now completed our circuit and wiring shape lets use them

About Block Inlining

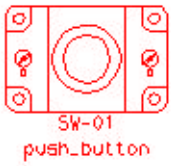
Instancing a block automatically inserts the block into a fiber if a pair of opposing ports on the block coincides with a fiber segment. The fiber itself is split into two separate fibers with identical properties. This operation is called inlining. If several sets of opposing ports on a block also coincide with fiber segments, then these sets of opposing ports are also inserted into the fiber.
During instancing, while the block is floating on the pointer, the tool shows you whether a pair of ports will be inlined by drawing an X on the relevant ports. This also works with groups, for which all the ports in the group and its members are taken into account. To make the component assume the orientation of an inclined fiber, use the middle mouse button for the initial orientation followed by the right or left relative turns, as necessary.

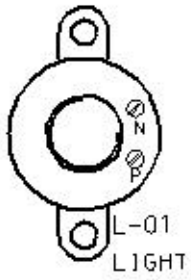
1. **Open** Circuit Diagram
2. From the Diagramming menu **Select** 
3. **Select** Switches > **Select** the circuit shape of Push Button Switch
4. Select OK
5. **Click** RMB > **Select** Turn Left
6. **Inline the switch** into the vertical section of P-01
7. **Select**  to update the labels

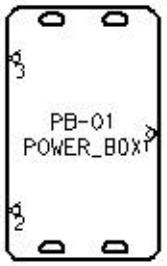


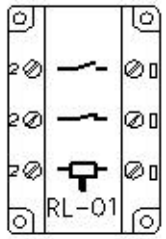
Placing the new wiring shape

1. **Open** the wiring Diagram
2. **Select**  from the diagramming tool tray
3. **Select** Open 
4. **Select** Unplaced-blocks.rep
5. **Select** the Criteria tab
6. **Select** Sheets
7. **Select** Not On
8. **Select** Wiring Diagram
9. **Select**  to generate the report
10. **Close** the Format Report Dialog
11. From the report results dialog **Select** SW-01
12. From the report results dialog **Select** 
13. **Select** Push button > **Select** OK
14. **Place** to the left of the sheet









Datatable Explorer

We are now going to use csv files to apply properties to our shapes.

About Managing Datatables Using the Datatable Explorer

The Datatable Explorer allows you to manage datatables. To Open the Datatable Explorer, Click  on the toolbar. The Datatable Explorer dialog box opens.
You can perform various operations using the icons provided in the Datatable Explorer. You first click an item in the tree in the left pane. Depending on the type of node Selected in the tree, appropriate information is displayed in the right pane.
A summary of the actions you can perform follows. These operations can also be initiated from a menu opened by clicking the right mouse button while the cursor is over an item in the tree.

Folder actions

 Navigates one level up in the tree hierarchy. This icon is enabled only when the currently Selected node is not a root node.

 Creates a new folder in the currently Selected folder. This icon is enabled only when a folder node is selected in the tree.

Load actions

 Loads a datatable. This icon is enabled only when a folder node is selected.

A file **Select**or is displayed after you **click** one of the load icons. **Select** a file and **Click OK**. The **Reload** dialog box opens. This dialog box displays:

- The files that will be loaded.
- The files that already exist in the design and will be reused.

Click OK and the Selected file is reloaded. To reload multi-level files, either delete the original file first, or reload each file explicitly.

Miscellaneous actions

 Reloads the Selected table from the file. This icon is enabled only when a datatable, membermap, or portmap is selected in the tree.

 Saves the Selected table to a CSV file. This icon is enabled only when a membermap or portmap is selected in the tree.

 Deletes the Selected table. This icon is enabled only when a datatable, membermap, or portmap is selected in the tree and the Selected table does not have any references.

About CSV files

For this task we shall create a generic fiber and use a datatable to have many different instance properties.

1. **Open** wires.csv from the CSV Files folder & review the format of the csv files

Header row Type:Datatable must be present for Routed Systems Designer to identify the type of CSV file

Name format is used by Routed Systems Designer to auto assign wire names. This can be included in the CSV file if naming convention is the same for all wires, or it can be added to each wire spool individually in the fiber catalog (if you want to add it manually DO NOT include it in the CSV file – values that are read in from a CSV file can NOT be modified in Routed Systems Designer.

The column $$datasettype$$ MUST be present in order for Pro/CABLING to be able to create the spools from the logical reference

This line is for the generic fiber

Microsoft wire_csv.csv

File Edit Insert Format Tools Data Window Help

A1 = Type

	A	B	C	D	E	F	G	H
1	Type	Datatable						
2	spool nam	color	thickness	min_bend_	units	wire_gauge	name form	$$datasettype$$
3	unset	unset	unset	unset	unset	unset	W-##	spool!wire_spool
4	4_blu_sxl	blu	5.2	6.5	MM	4	W-##	spool!wire_spool
5	4_red_sxl	red	5.2	6.5	MM	4	W-##	spool!wire_spool
6	6_red_sxl	red	4.1	5.2	MM	6	W-##	spool!wire_spool
7	6_grn_sxl	grn	4.1	5.2	MM	6	W-##	spool!wire_spool
8	8_blk_sxl	blk	3.3	4.1	MM	8	W-##	spool!wire_spool
9	8_red_sxl	red	3.3	4.1	MM	8	W-##	spool!wire_spool
10	10_blk_sxl	blk	2.6	3.3	MM	10	W-##	spool!wire_spool
11	10_red_sx	red	2.6	3.3	MM	10	W-##	spool!wire_spool
12	12_blk_sxl	blk	2.1	2.6	MM	12	W-##	spool!wire_spool

wire_csv

Min_bend_radius, thickness & units are required properties for Pro/CABLING

About Datatables

In Routed Systems Designer, properties provide objects with a way of storing information that is specific to an instance of the object. Some object properties are related. The relationship between such properties is captured in a dataset. A collection of datasets for the same object makes up a datatable.
You use datatables to provide a selection of datasets for a specific type of component and to provide datasets that can be applied to descendents. For example, if you want to indicate that a resistor in a circuit diagram has an electrical resistance, you can define the property Resistance that holds the value for that resistor (for example,150 ohms). Because resistors are commonly supplied with set values for resistance, you may need to constrain the resistance to a list of preferred values. Additionally, you may also want to provide the name of the manufacturers and material. Because different manufacturers can make their resistors from different materials, it may be useful to specify manufacturer and material at the same time

A datatable is created in a Spreadsheet .csv file containing columns of properties and rows representing different combinations of those properties called "Datasets".

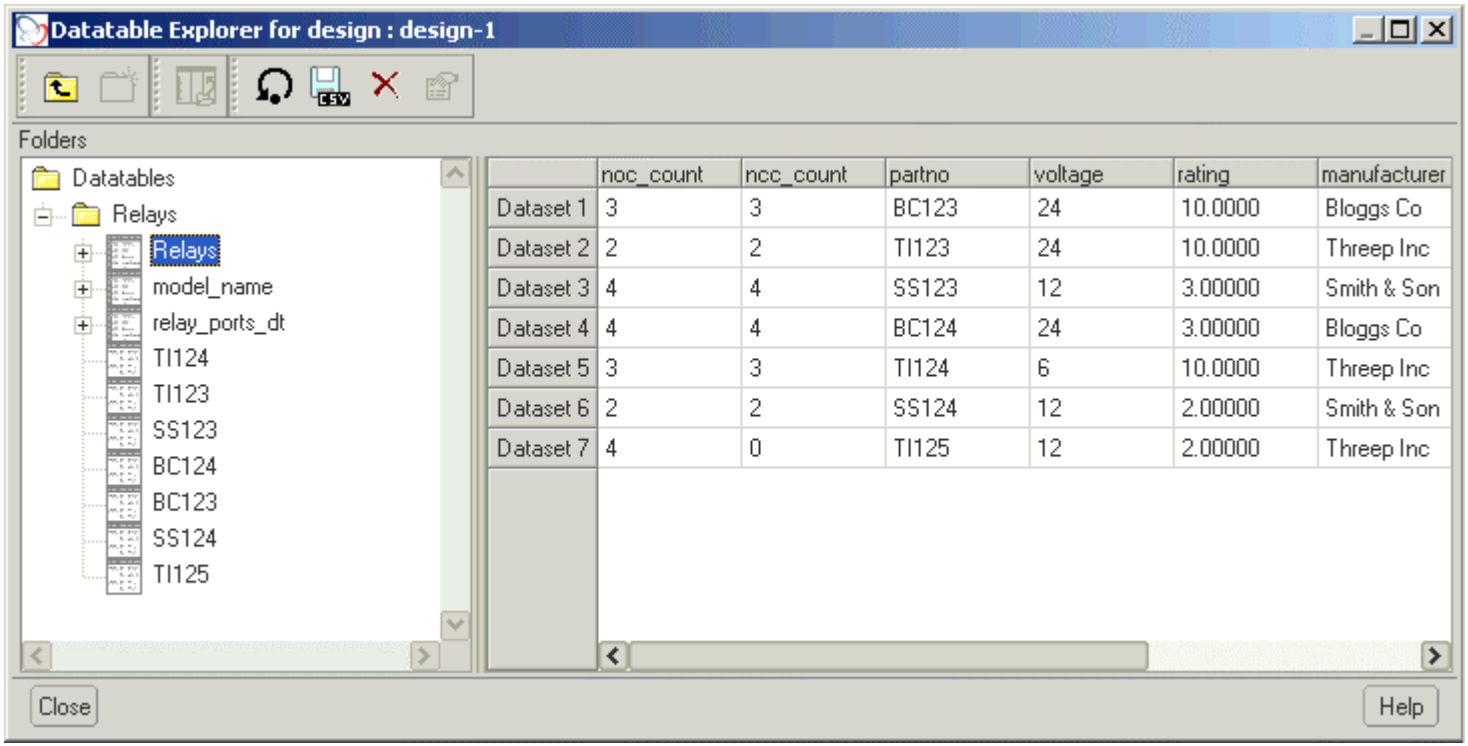
var_connectors.csv

	A	B	C	D	E	F	G	H	I	J
1	Type	Datatable								
2	manufacturer	description	num_of_pins	def_grouping	def_internal_len	model_name	obj_type	color	gender	$$membermapfile$$
3	Fargo	relay block	5	round	25	59022	connector	gry	FEMALE	var_59022_mmap.csv
4	Fargo	relay block	10	round	25	59222	connector	gry	FEMALE	var_59222_mmap.csv
5	Fargo	fuse-cb block	2	round	30	59216	connector	gry	FEMALE	var_conn_num_mmap.csv
6	Fargo	fuse-cb block 2-way	4	round	30	59217	connector	gry	FEMALE	var_59217_mmap.csv
7	PE	keyswitch conn	6	round	25	2984017	connector	blk	FEMALE	var_2984017_mmap.csv
8	Packard	150s sealed M	10	round	0	12045808	connector	blk	MALE	var_conn_alpha_mmap.csv
9	Packard	150s sealed M	8	round	0	12047931	connector	blk	MALE	var_conn_alpha_mmap.csv
10	Packard	150s sealed F	8	round	0	12047937	connector	blk	FEMALE	var_conn_alpha_mmap.csv
11	Packard	150s sealed F	2	round	0	12052641	connector	blk	FEMALE	var_conn_alpha_mmap.csv
12	Packard	150s sealed F	6	round	0	12052848	connector	blk	FEMALE	var_conn_alpha_mmap.csv
13	Packard	150s sealed F	10	round	0	12065425	connector	blk	FEMALE	var_conn_alpha_mmap.csv
14	Packard	150s sealed M	5	round	0	12103974	connector	nat	MALE	var_conn_alpha_mmap.csv
15	Packard	150s sealed F	3	round	0	12110293	connector	blk	FEMALE	var_conn_alpha_mmap.csv
16	Packard	150s sealed M	6	round	0	12124107	connector	blk	MALE	var_conn_alpha_mmap.csv
17	Packard	150s sealed M	3	round	0	12129615	connector	blk	MALE	var_conn_alpha_mmap.csv
18	Packard	150s sealed F	5	round	0	12146045	connector	nat	FEMALE	var_conn_alpha_mmap.csv
19	Packard	150s sealed M	2	round	0	12162000	connector	blk	MALE	var_conn_alpha_mmap.csv
20	Packard	150s sealed M	4	round	0	12162102	connector	blk	MALE	var_conn_alpha_mmap.csv
21	Packard	150s sealed F	4	round	0	12162144	connector	blk	FEMALE	var_conn_alpha_mmap.csv
22	Amp	2 way M conn	2	round	20	1-480683-0	connector	blk	MALE	var_conn_num_mmap.csv
23	PTC	1-way_conn	1	round	10	1W-F	connector	blk	FEMALE	var_conn_num_mmap.csv
24	Eaton	12 way switch conn	12	round	25	28_5940	connector	blk	FEMALE	var_28_5940_mmap.csv

var_connectors

In Routed Systems Designer we can create a Catalog of generic graphical component symbols and link them to datatables such that we can have a single graphical representation of a component with many possible sets of parameters.

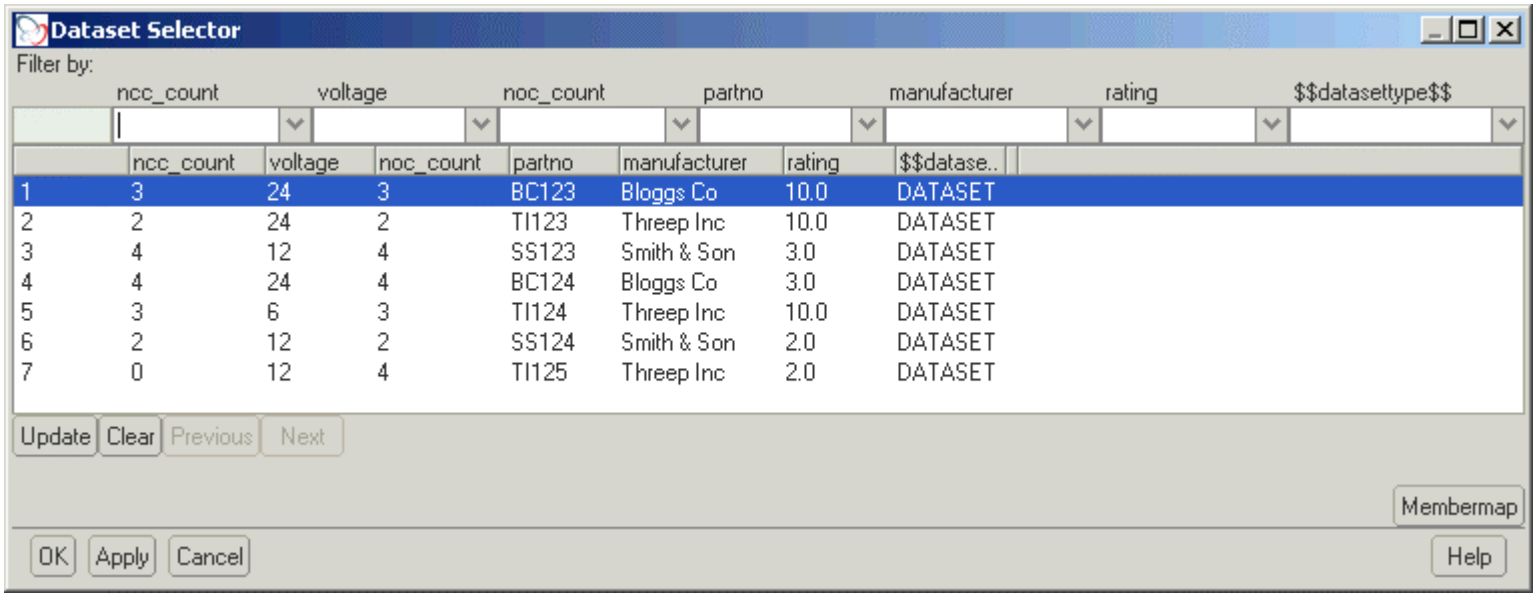
After creating the spreadsheet, datatables can be loaded into Routed Systems Designer using the Datatable Explorer:



	noc_count	ncc_count	partno	voltage	rating	manufacturer
Dataset 1	3	3	BC123	24	10.0000	Bloggs Co
Dataset 2	2	2	TI123	24	10.0000	Threep Inc
Dataset 3	4	4	SS123	12	3.00000	Smith & Son
Dataset 4	4	4	BC124	24	3.00000	Bloggs Co
Dataset 5	3	3	TI124	6	10.0000	Threep Inc
Dataset 6	2	2	SS124	12	2.00000	Smith & Son
Dataset 7	4	0	TI125	12	2.00000	Threep Inc

Once loaded, datatables can be linked to Catalog components by using the **Select** datatable command under object properties in the Catalog Explorer:

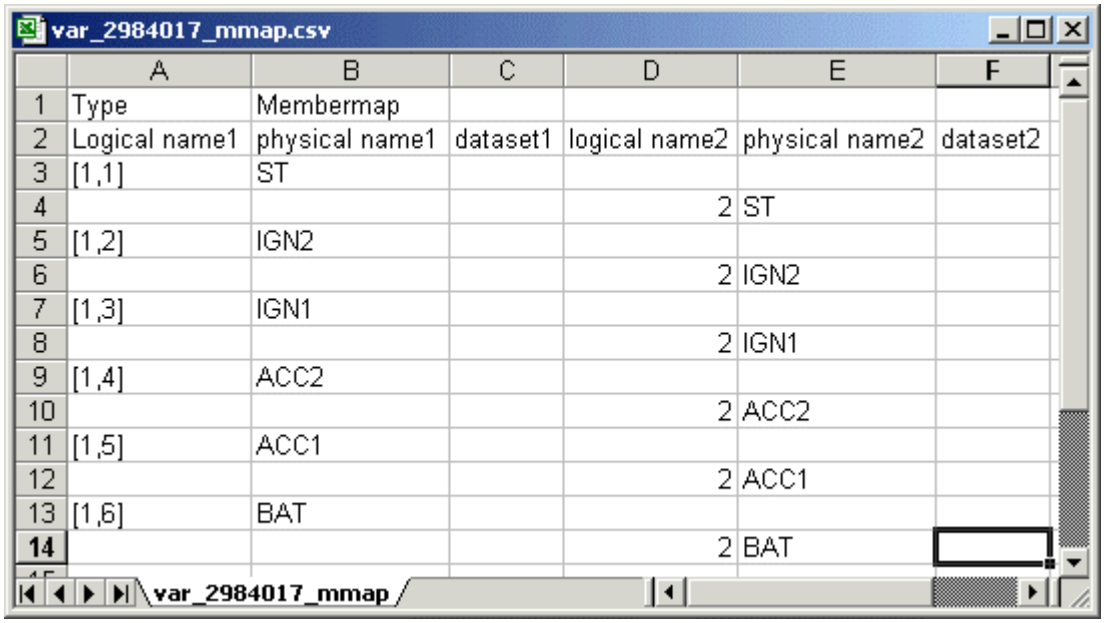
During placement of instances of the Catalog component on a diagram sheet a dataset (row) from the datatable can be applied to each component using the Dataset **Select**or:



About Membermaps

Most components and connectors in Routed Systems Designer have at least two levels (the component level and the ports level that lives inside the component) and so we can have nested datatables that can be used to apply unique datasets throughout the component structure.

To do this we need to have a "Membermap" file to define the structure of the component (which is also another spreadsheet .csv file)



	A	B	C	D	E	F
1	Type	Membermap				
2	Logical name1	physical name1	dataset1	logical name2	physical name2	dataset2
3	[1,1]	ST				
4				2	ST	
5	[1,2]	IGN2				
6				2	IGN2	
7	[1,3]	IGN1				
8				2	IGN1	
9	[1,4]	ACC2				
10				2	ACC2	
11	[1,5]	ACC1				
12				2	ACC1	
13	[1,6]	BAT				
14				2	BAT	

Note the membermap file doesn't define any properties, just the names of the members, the level they live on, and optionally reference pointers to datasets for the members within other external datatables.

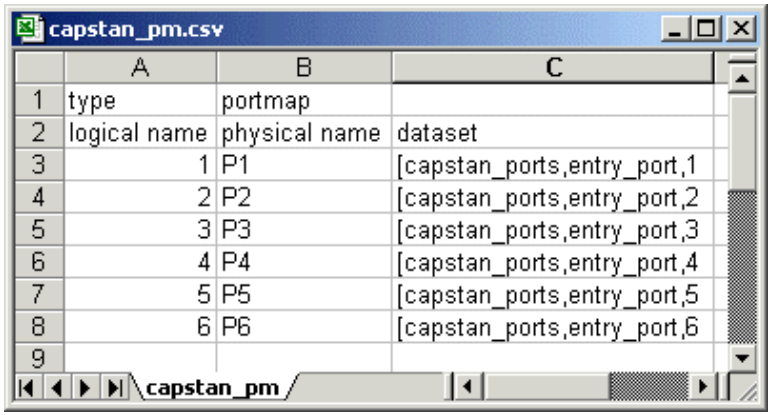
The above membermap is for a variable connector having a maximum of 6 internal blocks, each of which has a single port.

Logical Name is the “index position” of the object in the Catalog definition sheet (see the Datatables tutorial doc for more information on index position)

Physical Name is the desired name to be applied to the object instance on the diagram sheet

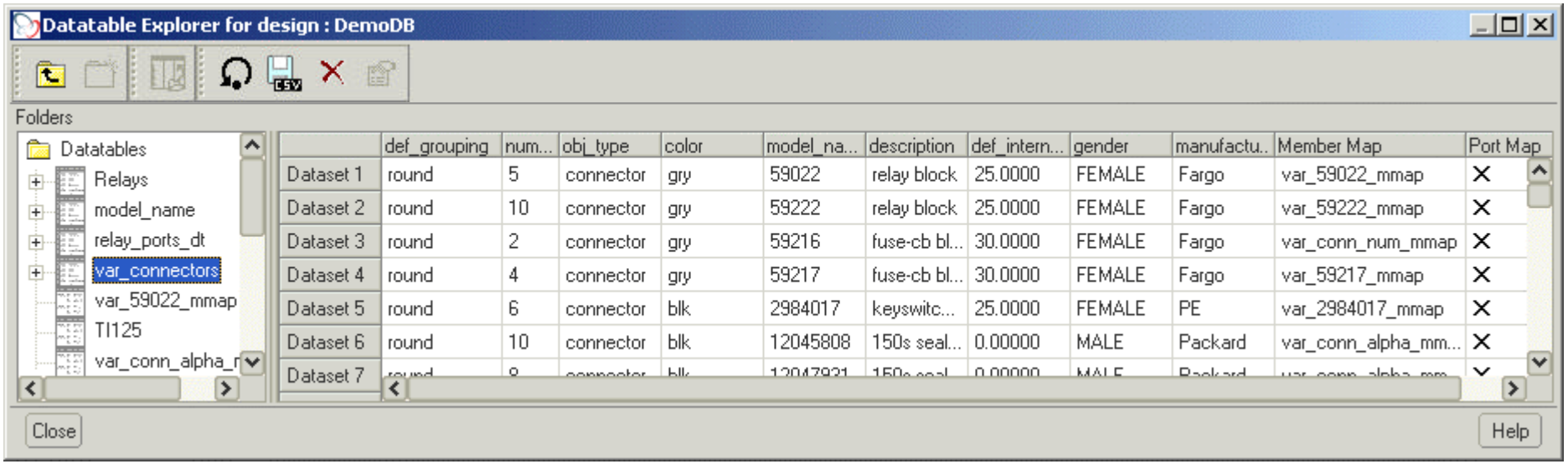
About Portmaps

A Portmap is identical to a Membermap, but can only be used with Block components, having a two level structure (i.e. Block-->Port).
Historically, Portmaps preceded Membermaps in RSDesigner. A Membermap can do all that a Portmap can do and more, essentially making Portmaps now obsolete. Portmap functionality has been maintained for those that have created them and are accustomed to using them.



	A	B	C
1	type	portmap	
2	logical name	physical name	dataset
3	1	P1	[capstan_ports,entry_port,1
4	2	P2	[capstan_ports,entry_port,2
5	3	P3	[capstan_ports,entry_port,3
6	4	P4	[capstan_ports,entry_port,4
7	5	P5	[capstan_ports,entry_port,5
8	6	P6	[capstan_ports,entry_port,6
9			

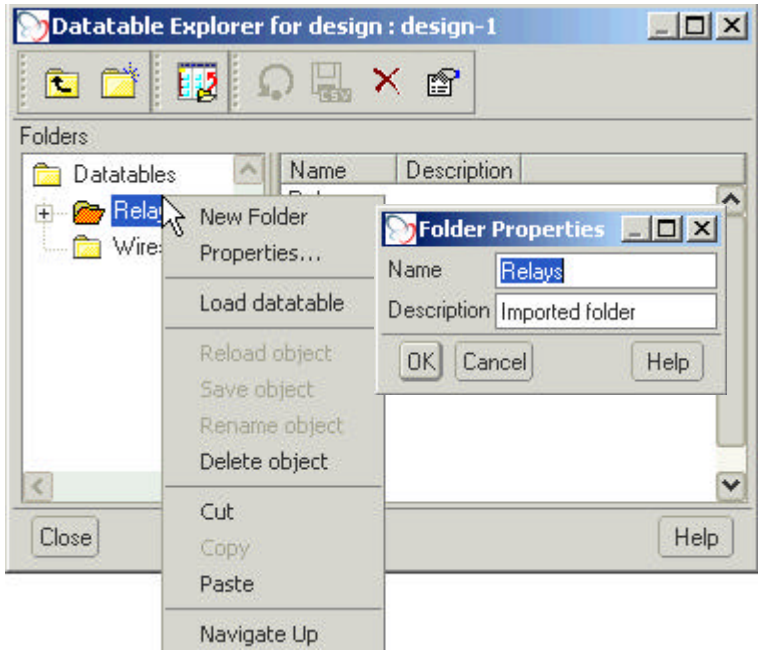
Note the use of a Portmap/Membermap is an either/or situation. In Datatable Explorer, the last two columns displayed for a Datatable are "Member Map" and "Port Map". For any given Dataset (row), there should never be references to both a Membermap and a Portmap in these columns. Although loadable in Datatable Explorer, **Select**ion of a datatable in Catalog Explorer containing any rows defined this way will result in "cannot apply dataset" errors.



	def_grouping	num...	obj_type	color	model_na...	description	def_intern...	gender	manufactu..	Member Map	Port Map
Dataset 1	round	5	connector	gry	59022	relay block	25.0000	FEMALE	Fargo	var_59022_mmap	✕
Dataset 2	round	10	connector	gry	59222	relay block	25.0000	FEMALE	Fargo	var_59222_mmap	✕
Dataset 3	round	2	connector	gry	59216	fuse-cb bl...	30.0000	FEMALE	Fargo	var_conn_num_mmap	✕
Dataset 4	round	4	connector	gry	59217	fuse-cb bl...	30.0000	FEMALE	Fargo	var_59217_mmap	✕
Dataset 5	round	6	connector	blk	2984017	keyswitc...	25.0000	FEMALE	PE	var_2984017_mmap	✕
Dataset 6	round	10	connector	blk	12045808	150s seal...	0.00000	MALE	Packard	var_conn_alpha_mm...	✕
Dataset 7											

Creating Folders

2. **Select** the data table Explorer icon  .
3. **Select** folder Datatables.
4. **Select**  to create a new folder.
5. **Select** the new folder.
6. **Select**  to edit the folder properties.
7. **Set** the name of the folder to Wires.



Loading CSV Files

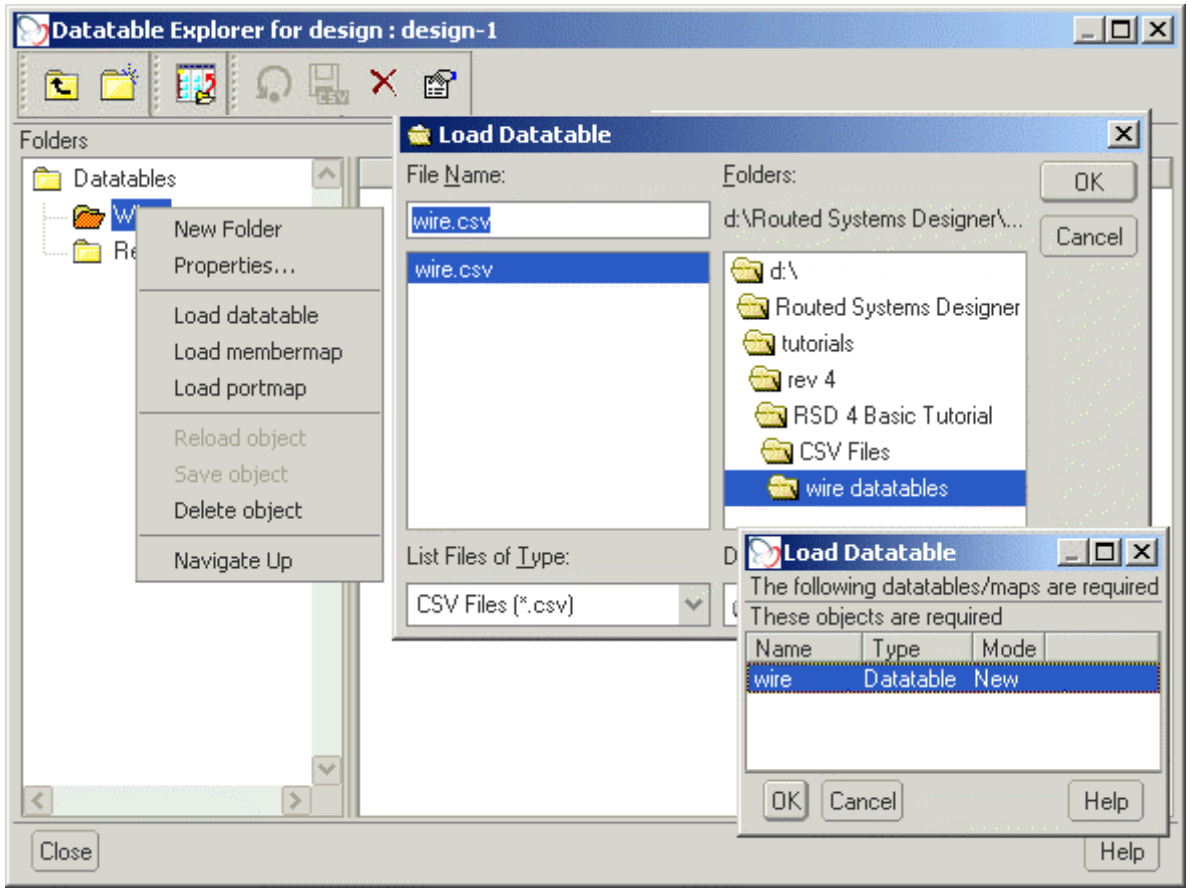
Now lets import a CSV files into the wires folder.

8. **Select** folder Wires.
9. **Select**  to load a Datatable.
10. **Select** Wire.csv from the CSV files \ wire datatables directory
11. **Select** OK from the Load Datatable Dialog
12. **Select** OK from the second Load Datatable dialog.

You will now see that the Wires folder has been populated with data from the CSV file.

13. **Close** the Datatable Explorer.

We have now loaded data from CSV files into RSD to store in a central location. We can now utilise these datatable's at any time.

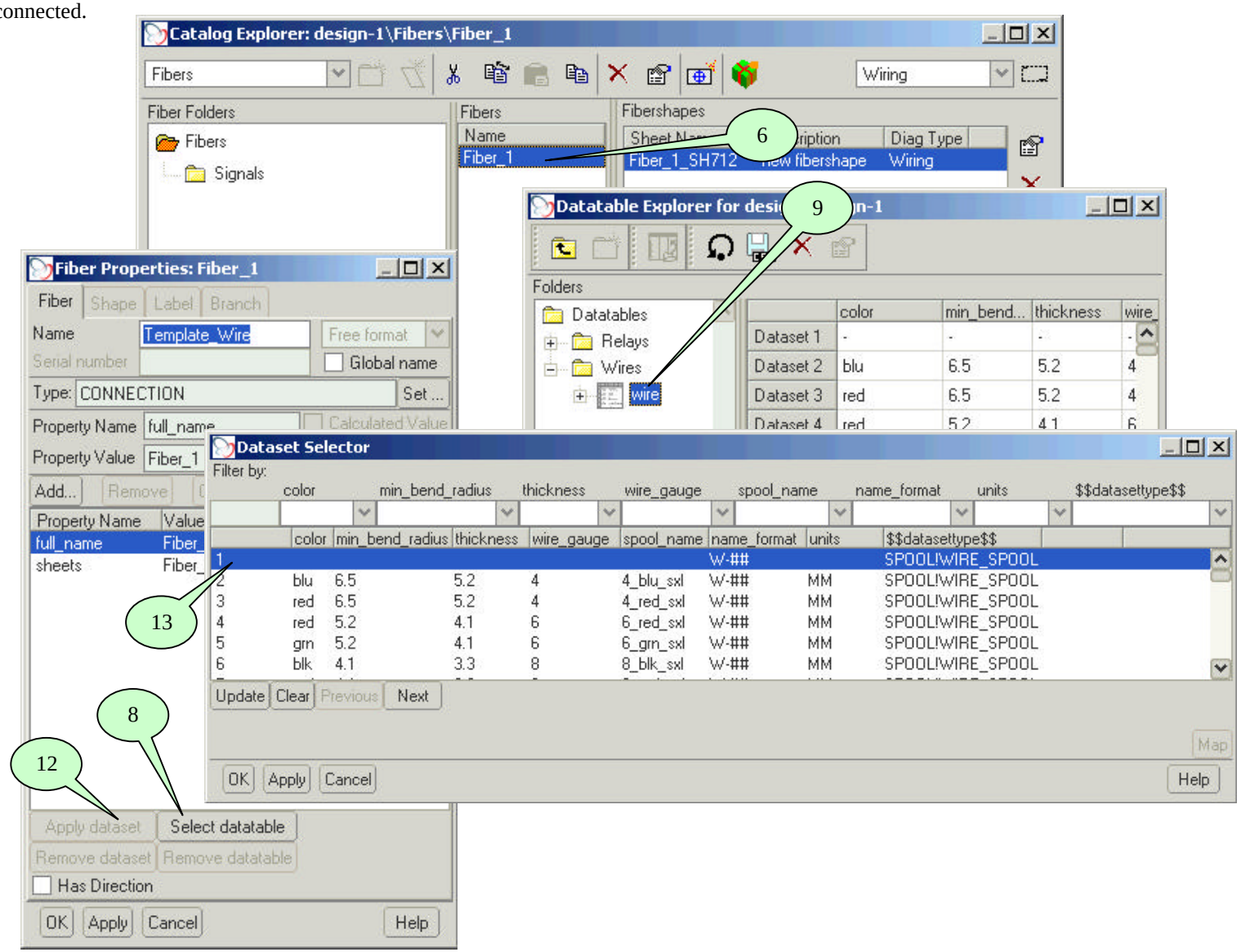


Fibers

A fiber is a component such as a wire, a signal network or a pipeline, that can be routed. A fiber can have two or more ports joined by a network of segments. The network does not contain loops.
A fiber is specific to a diagram type, unlike blocks and groups, which can be represented by multiple shapes with different diagram types.
A fiber may exist on its own, without being connected.

Creating a Template Fiber

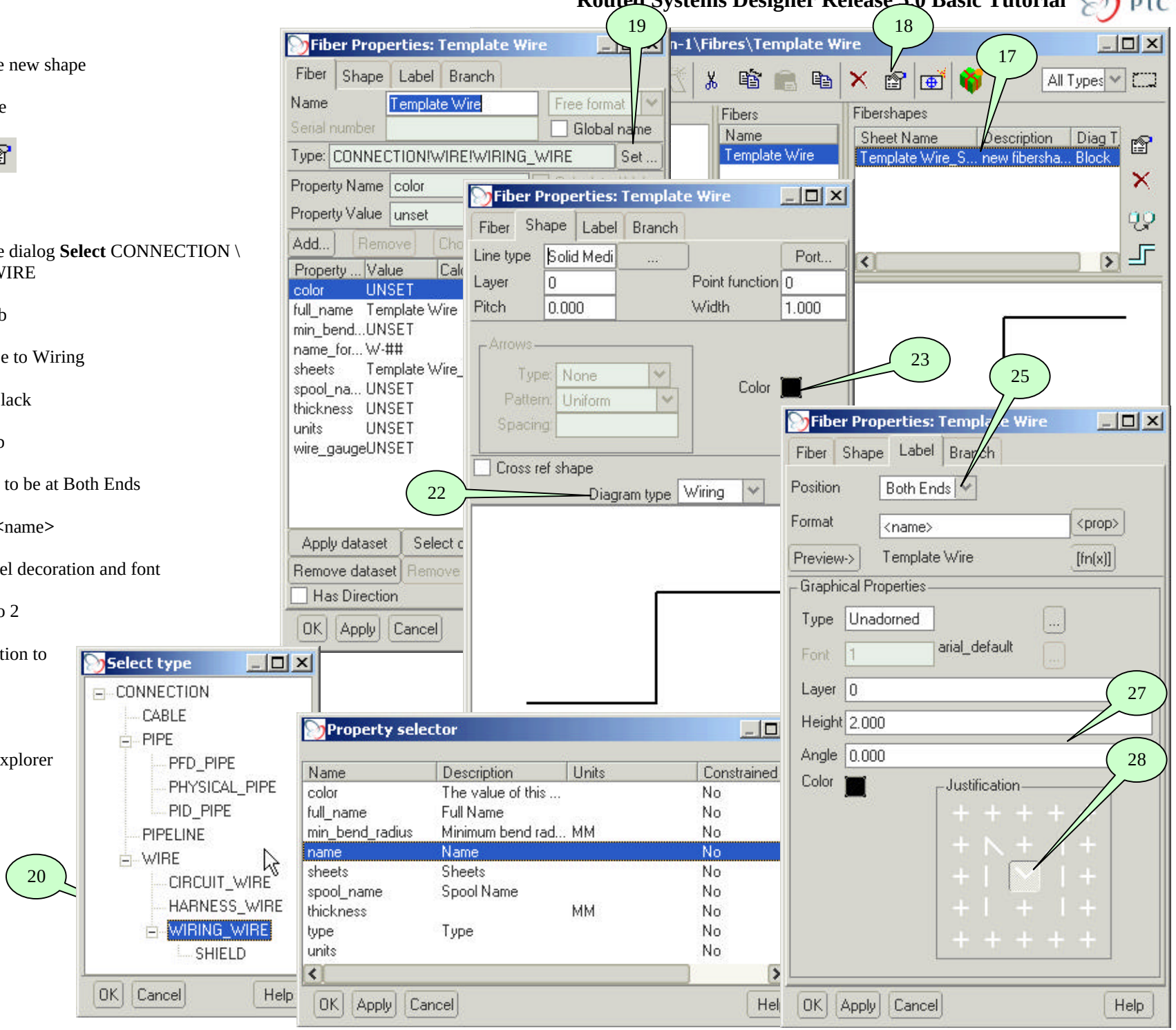
1. **Select**  to Open the Catalog Explorer
2. **Select** Fibres from the pull down menu
3. **Select**  to create a new fibre
4. **Select** the new fibre
5. **Click** RMB > **Select** Properties
6. **Set** the fibre name to Template Wire
7. **Select** Apply
8. **Select**, <Select Datatable> button
9. From the Datatable explorer **Select** wire from the wires folder
10. **Select OK** from the datatable explorer dialog.
11. From the properties dialog **Select** Apply
12. **Select** Apply Dataset
13. From the Dataset Selector dialog **Select** Line 1
14. **Select** OK from the Dataset Selector dialog
15. **Select** OK from the properties dialog



16. **Select**  to create new shape
17. **Select** the new shape
18. **Select** Properties 
19. Select Set
20. From the Select type dialog **Select** CONNECTION \ WIRE \ WIRING_WIRE
21. Select the Shape Tab
22. **Set** the Diagram type to Wiring
23. **Set** the color to be black
24. **Select** the Label Tab
25. **Set** the text position to be at Both Ends
26. **Set** label format to <name>

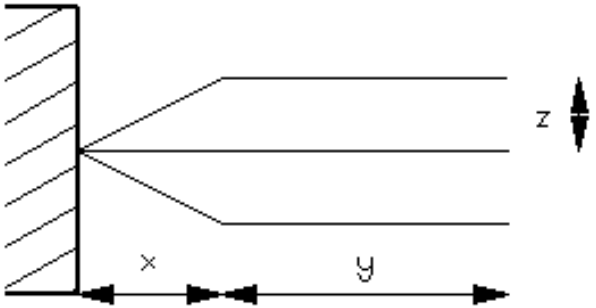
It is optional to set the label decoration and font

27. **Set** the text height to 2
28. **Set** the text justification to centre
29. Select OK
30. **Close** the Catalog Explorer



About Fiber End Geometry

When a fiber port is connected to another port that does not belong to a fiber, a section of fiber leading from that port is fixed regardless of where the fiber is routed. This section of the fiber, called the end geometry, is oriented in the direction of the port to which the fiber's port is connected and cannot be edited.
The following figure shows the three basic end-geometries currently defined in Routed Systems Designer. You can also define the number of fans, that is, the number of pairs of non-straight end-geometries (the figure below shows one fan). The total number of separate routes into a port is 1+2*<number of fans>.



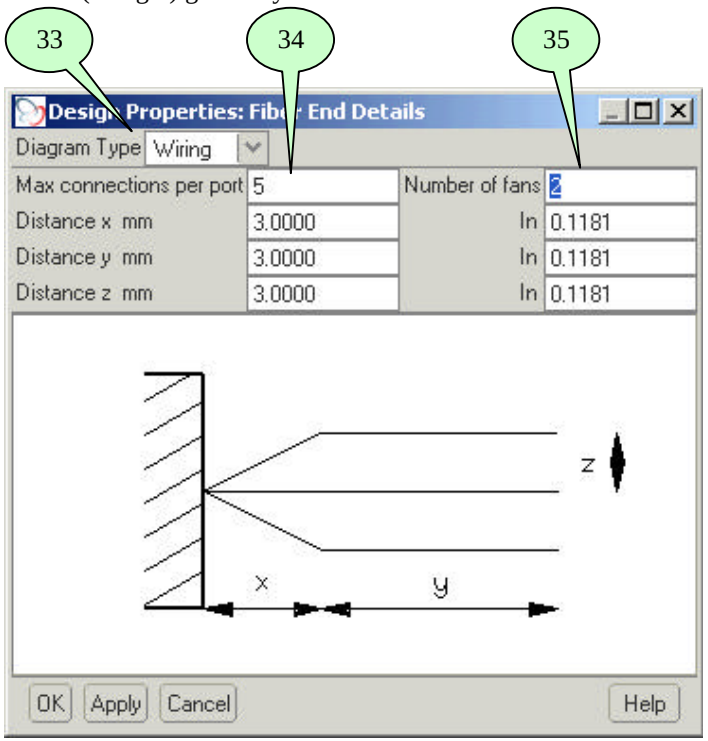
The first fiber to be connected uses the middle (straight) geometry. The second and subsequent fibers to be connected use the remaining routes such that each created fiber has minimal length. If there are more connections allowed than available separate routes then all the extra routes use the middle (straight) geometry.

Setting Fiber End Details & Auto Wiring Wire

31. **Select** File > Design Properties

.

32. **Select** Fibre End Details
33. **Select** Wiring from the pull down list
34. **Set** the max connections per port to 5
35. **Set** the number of fans to 2
36. Select OK
37. **Select** Auto Wiring Fibre
38. **Select** Template Wire
39. Select OK
40. Select OK



To Wire a Signal Network

The Wire and Core Assignment Editor allows you to create and connect wiring diagram wires that correspond to the connections of a circuit diagram fiber. The wires created by this tool are instances of the designated auto wire fiber prototype.
To wire a signal network, you can either select the ports or edit the links between the ports in the network. You cannot select ports from multiple networks or edit links across multiple networks. You can only select ports and edit links of a single signal network. The network may be simple or complex.
To designate a wiring diagram catalog fiber as the auto wire fiber prototype, select File > Design Properties > Auto Wiring Fiber. Select the required fiber using the Catalog Selector. If you have not designated a wiring diagram catalog fiber, one is created if required.

Wiring the signal network

- Click  in the Diagramming tool tray. The Signal Network Selector dialog box opens.

Note: Any fiber shape with a diagram type of `Circuit` is regarded as a signal network.

- Click Previous or next on the Signal Network Selector dialog box to navigate through multiple networks. The shapes of the current network on the current sheet are highlighted to aid Selection.
- To choose a network to edit from the multiple networks that exist, select a component or a port that is part of the signal network.
- Click Show Simple or Show Complex on the Signal Network Selector dialog box to select either a simple network or a complex network.

The status of the network wiring is one of the following:

- Under One or more ports that are connected in the circuit diagram are not connected in the wiring diagram.
- Over All the additional ports that are connected in the circuit diagram are connected in the wiring diagram. But some pins are connected by two or more paths.
- Exactly All ports that are connected in the circuit diagram are connected in the wiring diagram by exactly one path. However, some ports may have more than two wires connected to it. For example, port 1 can be wired to port 2, port 3, and port 4. All ports are connected by exactly one path. But port 1 has three wires connected to it.

- Click Display the wire assignment editor. The Wire and Core Assignment Editor dialog box opens.
- In the Wire and Core Assignment Editor dialog box, Click Edit Ports to select ports in the network or Click Edit links to edit the links between the ports in the signal network.

A link is the gap between ports. Links are of the following type:

- **Proposed link** When a network that is not wired is selected, the wiring proposal list appears and the gap between the pins is a proposed link. A proposed link is represented by dotted lines.
- **Approved link** The link changes to approved when you accept the wiring proposal.
- **Drawn link** The drawn link exists as fiber graphics on a wiring diagram and cannot be modified using the Wire and Core Assignment Editor.

Port Manipulation Mode

In the Port Manipulation mode, you can

- Select ports to change the order of the ports in the network or rework the network.
- Select ports not in the current block to work on a new network.
- Select one or more ports that are connected by approved or drawn links, or both.

Note: You cannot select ports across signal networks.
You can move, flip, cut, copy, or paste the Selected ports.

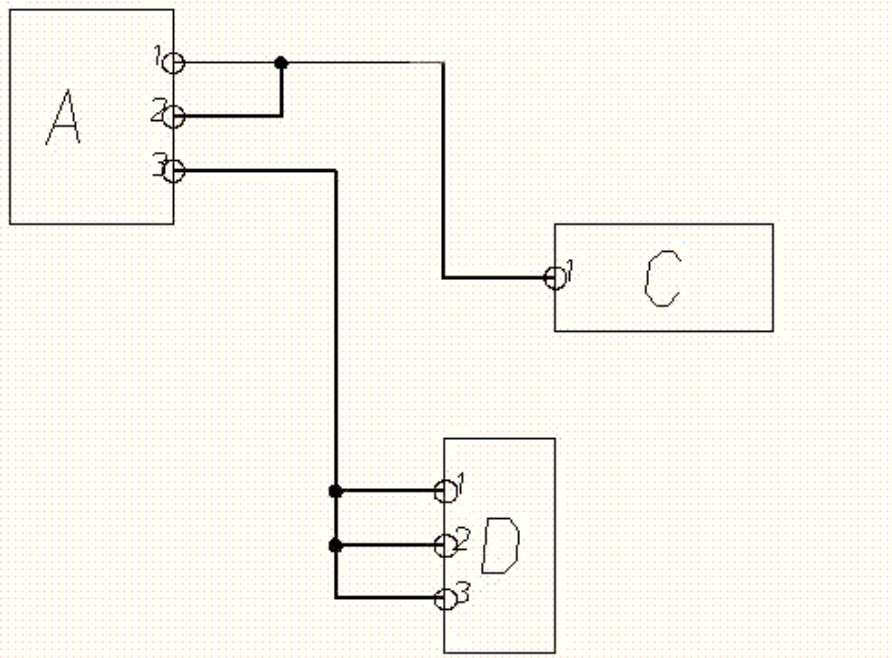
Link Manipulation Mode

In the Link Manipulation mode, you can:

- Apply Proposal Change the Selected proposed links to approved links.
- Approval Remove Change the Selected approved links to proposed links.
- Add Proposal Change Selected "no links" to proposals.
- Remove Link Change proposed or approved links to "no links".
- Edit Properties
- Edit Cable
- Click:
  - OK or Apply to apply the changes to the signal network and exit the wire edit session.
  - Abort to discard the changes that you made so that the network reverts to the state when the edit session was initiated.

Example: Use of the Wire and Core Assignment Editor

Consider the following example of how to display the wiring proposal list:



If you Select component A, the initial list proposal is as follows:

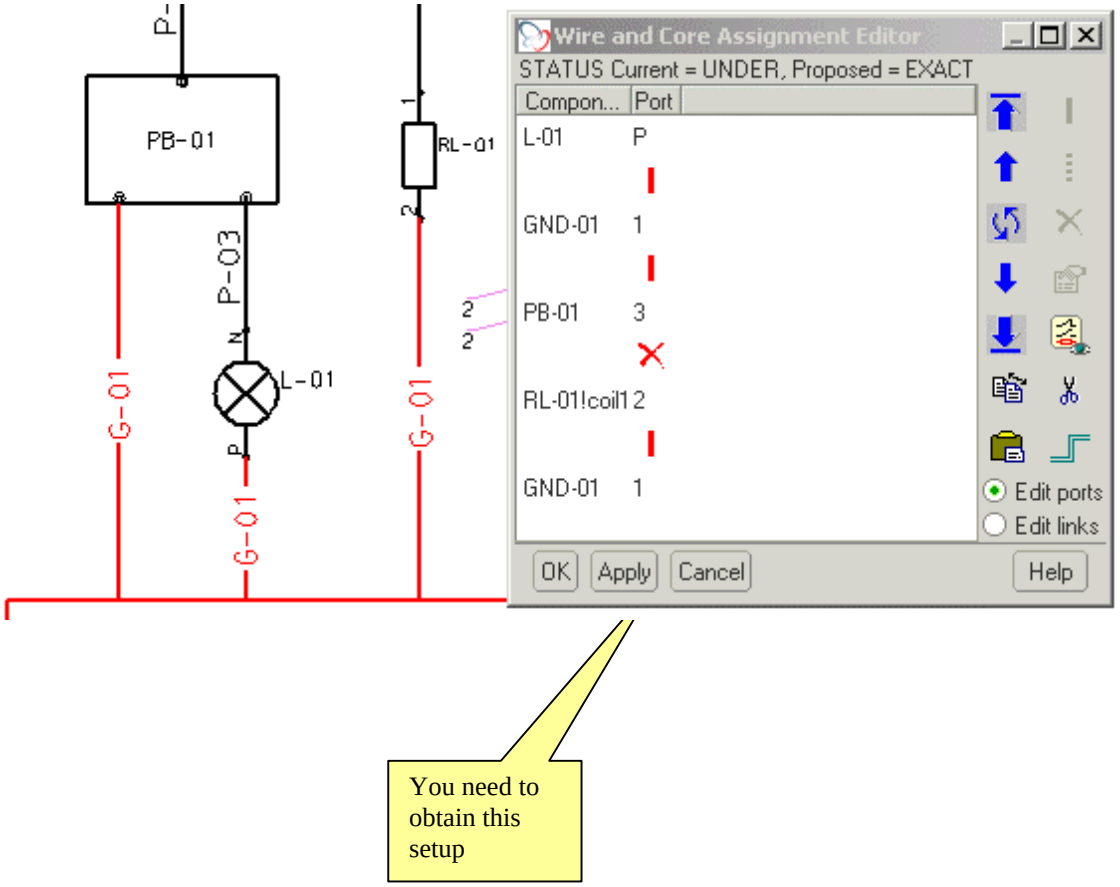
STATUS Current=UNDER,Proposed=EXACT		
Sheet1;A;1		
	x	x
Sheet1;A;2		
	x	x
Sheet1;C;1		
STATUS Current=UNDER,Proposed=EXACT		
Sheet1;A;3		
	x	x
Sheet1;D;1		
	x	x
Sheet1;D;2		
	x	x
Sheet1;D;3		

After edits, accept the proposal on the first network and edit the properties to add a wire datatable/dataset.
Change the proposal so that A;3 has a core running from it to each of D;1 D;2 and D;3. Assign core and cable data tables/data sets.

STATUS Current=UNDER,Proposed=EXACT		
Sheet1;A;1		
	x	x
Sheet1;A;2		
	x	x
Sheet1;C;1		
STATUS Current=UNDER,Proposed=EXACT		
Sheet1;A;3		
	x	x
Sheet1;D;1		
	x	x
Sheet1;D;2		
	x	x
Sheet1;D;3		

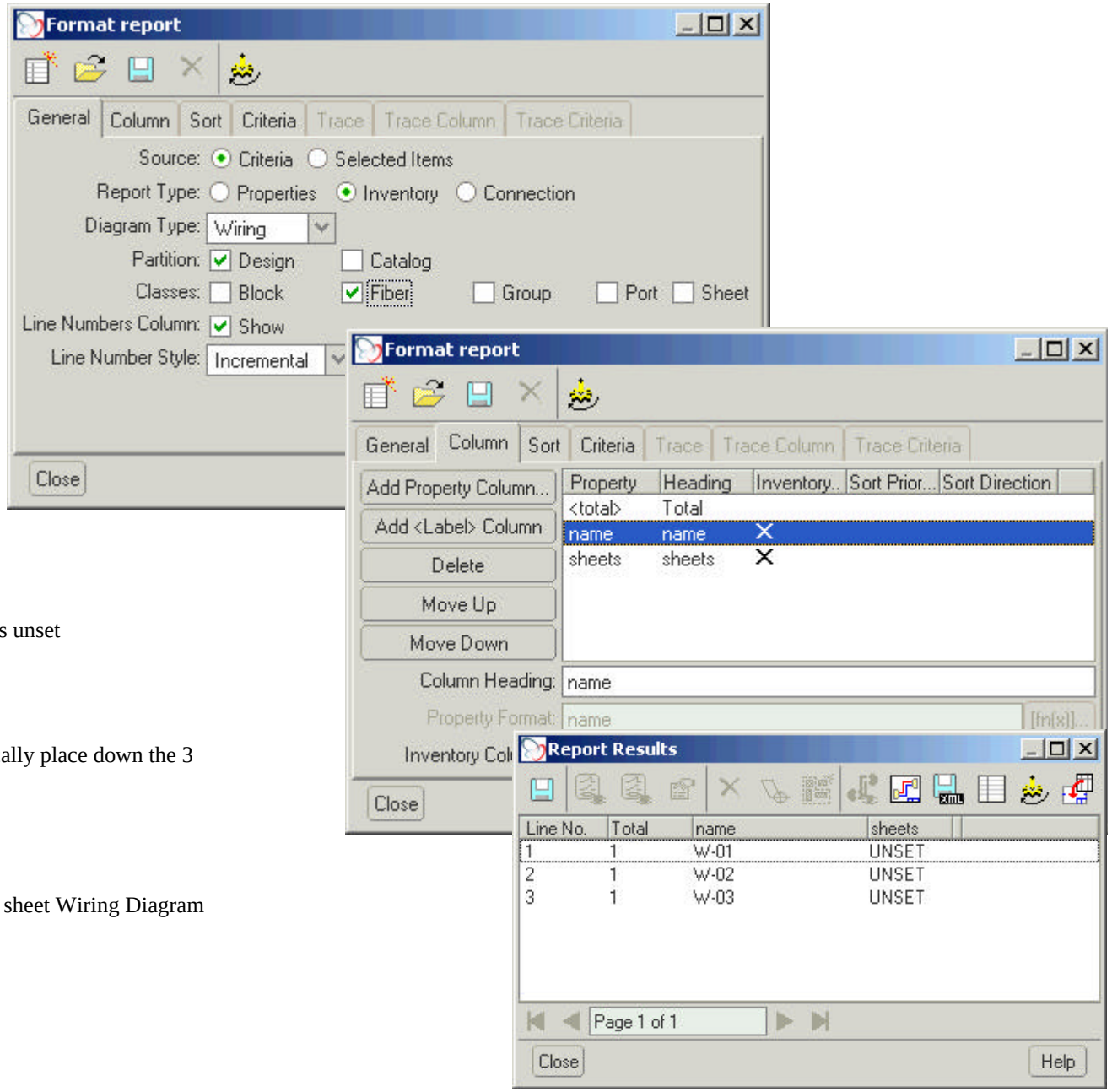
Wire & Core Assignment Editor – Circuit Diagrams

1. **Open** Sheet Circuit Diagram
2. **Select** signal G-01
3. Select 
4. **Select** GND-01
5. Select 
6. Select 
7. **Select** the copied GND-01
8. Select 
9. **Select** GND-01 at top of list
10. Select 
11. **Select** Edit links
12. **Select** the link between RL-01!coil1 2 and PB-01
13. Select 
14. **De-Select** the link between RL-01!coil1 2 and PB-01
15. **Select** all the remaining links
16. Select
17. Select OK



We have now created three wires.

Auto-generating Wires



1. **Open** Sheet Wiring Diagram
2. Select
3. **Select** new report
4. **Select** Inventory
5. **Select** Wiring
6. **De-Select** Block & Group & **Select** Fibre
7. **Select** the column tab
8. **Add** properties name and sheets
9. **Select** to generate the report.

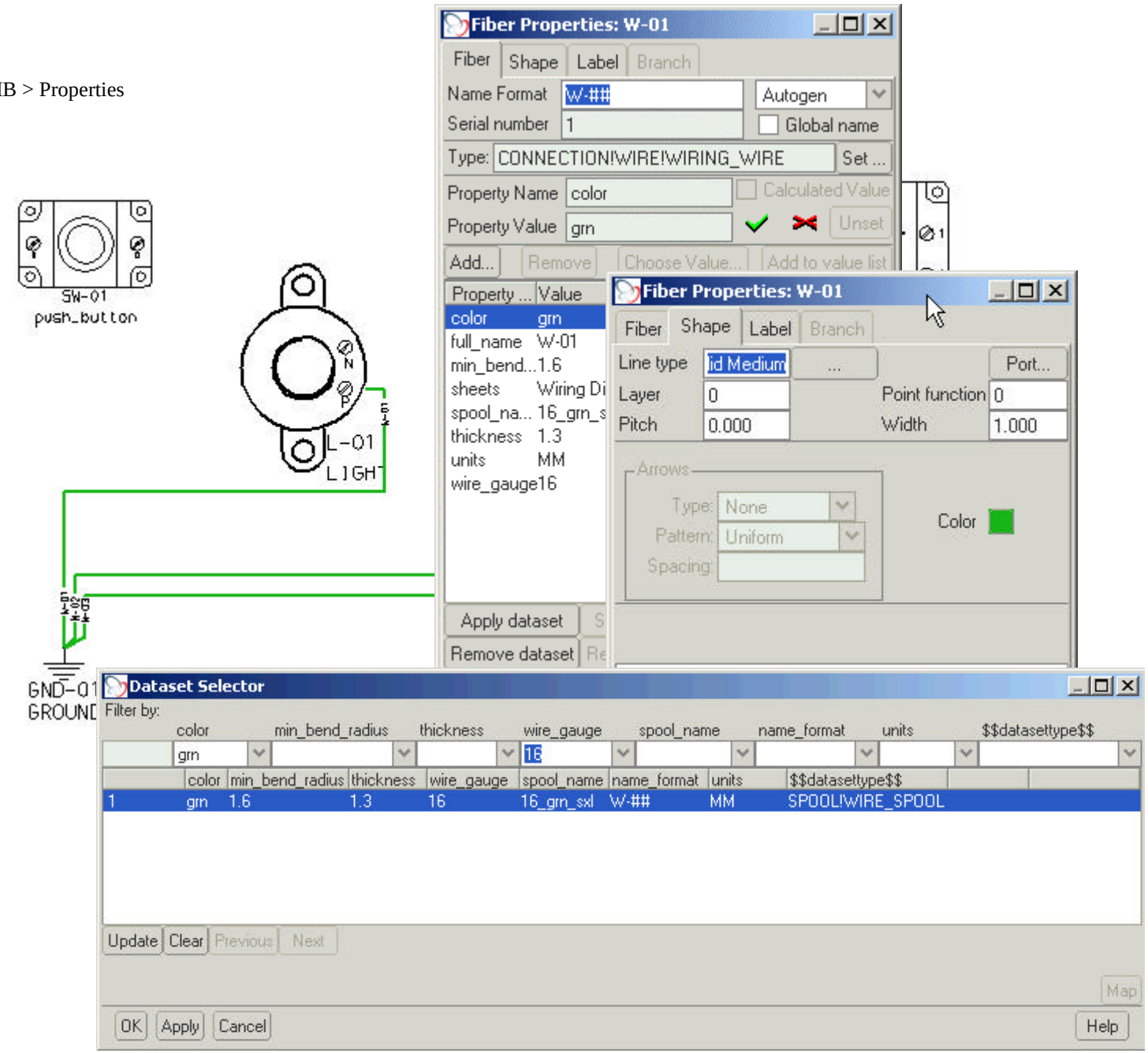
You will see that three wires are reporting the sheet name as unset

10. **Select** the three wires
11. **Select** from the report results menu to automatically place down the 3 Selected fibres.
12. Select

You will now see that the wires and now reporting to be on sheet Wiring Diagram

Applying properties to a fibre

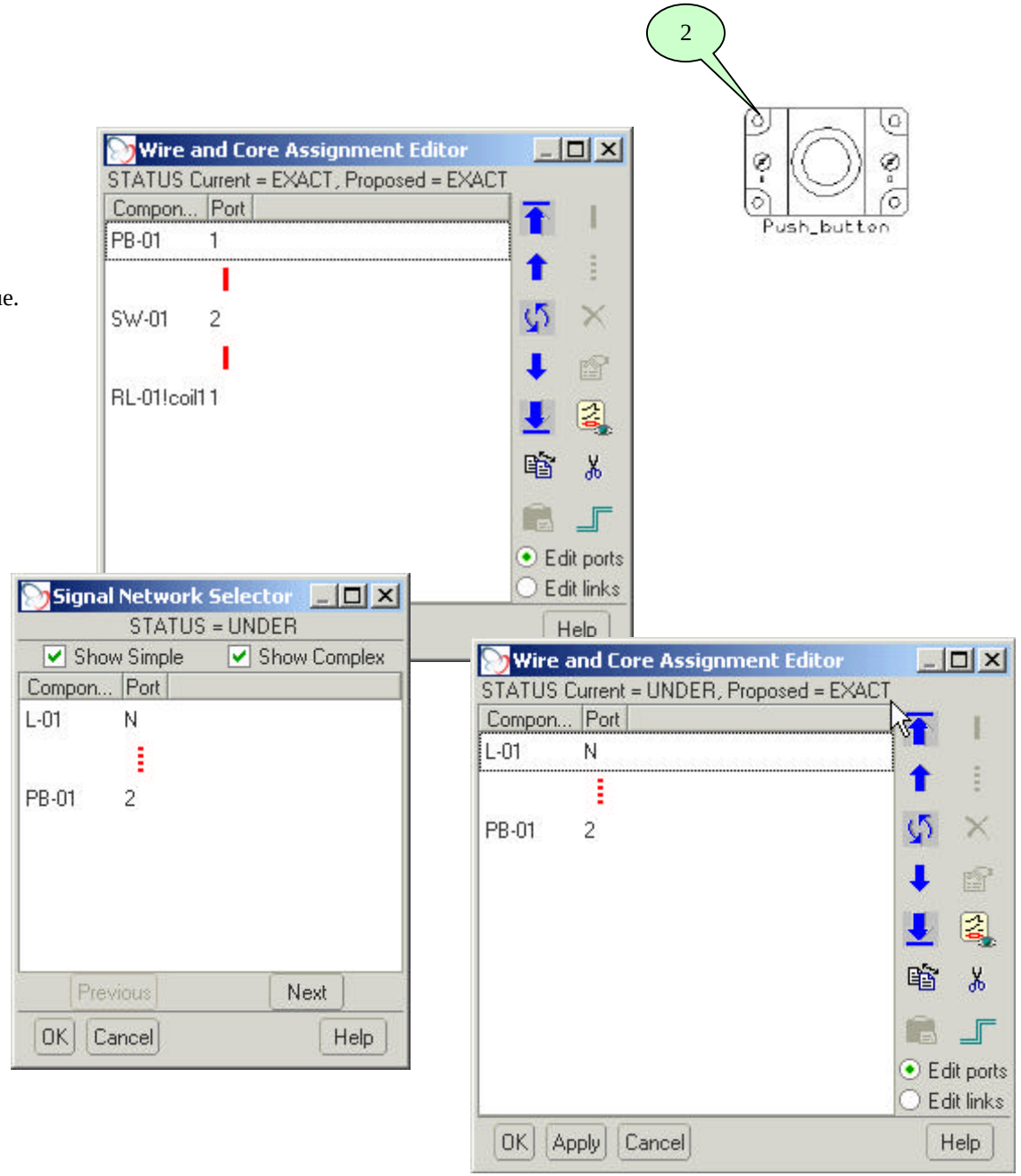
1. **Select** the fibre filter tool 
2. **Select** all three wires coming from G01 > RMB > Properties
3. **Select** Apply Dataset
4. **Enter** grn in color and 16 in wire gauge
5. **Select** Update
6. From the data Selector dialog **Select** OK
7. **Select** Apply
8. **Select** Shape Tab
9. **Set** the colour to green
10. **Select** Apply
11. Select OK



Wire and Core Assignment Editor - Wiring Diagrams

1. **Select** the general Selection tool 
2. **Select** SW-01
3. **Select** 
4. **Select** SW-01 and move into middle of list
5. **Select** Apply to create the links (wires) and **OK** to **close** the dialogue.
6. **Select** L-01
7. **Select** 
8. **Select** Next > OK
9. **Select** Apply > Select OK

We have now created a further three wires



Auto wiring

10. Select 

11. **Select** generate  (assuming that the report **Set** up has not been changed since the last wire report)

You will see that the three new wires are reporting the sheet name as unset

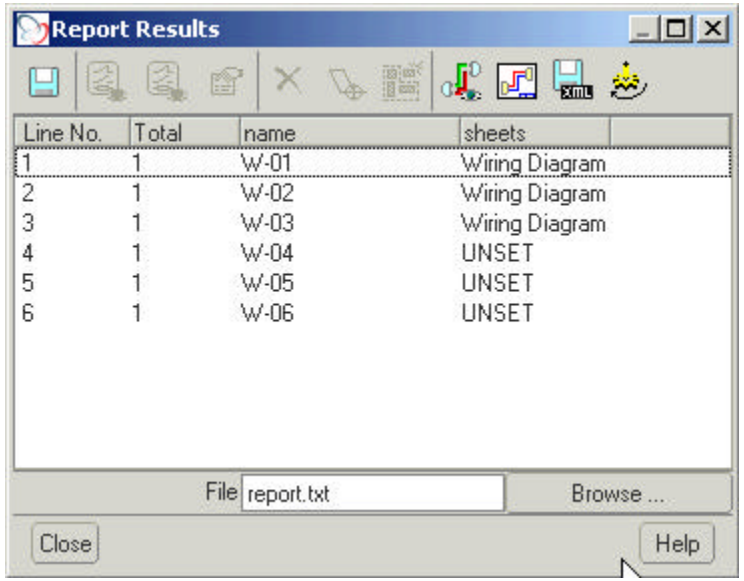
12. **Select** the three new wires

13. **Select**  from the report results menu to automatically place down the Selected fibre.

14. Select 

You will now see that the wire is now reporting to be on sheet Wiring Diagram

15. Quickly tidy the fibres up



Trail files

To **set** the properties and the color of the first three fibres involved a number of mouse clicks. Using a of trail file we can automate common tasks

About Trail Files

A trail file is a record of all menu choices, dialog-box choices, selections, and keyboard entries for a particular working session. Trail files allow you to view the record of activity so that you can reconstruct a previous working session or to recover from an abrupt termination of a session. Trail files are editable text (.txt) files.
When you run a trail file, the system replays the selections in the exact order in which they were originally made

To Record a Trail File

- Click Utilities > Trail File > Record Trail File. The Record Trail File dialog box opens.
- Type a name for the trail file in the Filename box. The file is placed in the current working directory. To place the file in a different location, Click File Selector and Browse to the new location.
- Note: The file must have a `.bac` extension.
- To add a header to the top of the trail file, type the value in the Macro Name box.
- To control the accuracy of the probe points, type a numeric value in the Decimal places box.
- To record the elapsed time of the trail file, Select Log Elapsed Time.
- Click OK.
- To stop recording the trail file, Click Utilities > Trail File > Close Trail File.

To Play a Trail File

- Click Utilities > Trail File > Play Trail File. The Play Trail File dialog box opens.
- Type the file name in the Filename box.
- Click OK.

To Automatically Run a Trail File When You Start RSD

- Click Options > Set startup trail. The Startup trail dialog box opens.
- Select the trail file that you want to run automatically when you start Routed Systems Designer.
- Click OK.

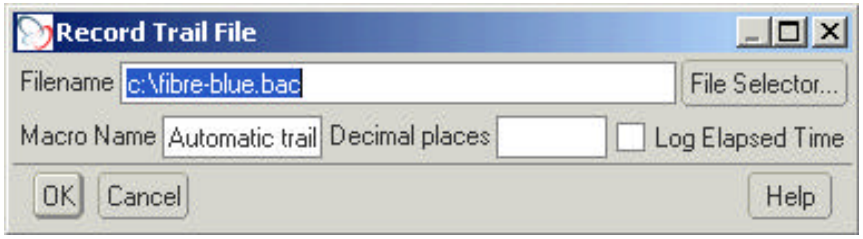
To Control the Display During Trail File Playback

- Minimize Routed Systems Designer during trail file playback so that dialog boxes do not appear. Minimizing the application also facilitates the quicker playback of the trail files.
- To stop all dialog boxes from appearing in the graphics area during trail file playback, Click Options > Switches, Select Suppress dialog when playing trails, and Click OK.
- The main window, however, remains Open. The dialog boxes appear when the trail file completes the replay.
- To restore the visibility of the application, if minimized, when the trail file completes the replay, Click Options > Switches, Select Restore Main Window after playing trails and Click OK.

To Compare Trail Files

- To compare trail files and record the comparison in another trail file:
- Click Utilities > Trail File > Compare Files. The Compare Files dialog box opens.
- Specify the file names:
- File 1—Specifies the file to be compared.
- File 2—Specifies the file to be compared with.
- Result File—Specifies the output file for the comparison.
- Specify the comparison type:
  - Exact—Compares files exactly.
  - Numeric—Compares numeric values in a line as numeric values.
- Specify the type of action to be taken:
  - Compare and Record—Compares and records the comparison in a trail file.
  - Record Only—Records the comparison in a trail file.
  - Click OK.

1. **Select** fibre W-04
2. **Select** Utilities > Trail Files> Record Trail File
3. **Enter** c:\fibre-blue.bac for file name
4. Select OK
5. **Click** RMB > **Select** Properties
6. Under the shape tab **Set** the color to Blue
7. Under the fibre tab **Select** Apply Dataset
8. Filter for color blu and a wire gauge of 16
9. **Select** Apply > **OK**
10. **Select** Utilities > Trail Files> **Close** Trail File
11. Repeat steps 1 to 10 for fibre W-05. Create a trail file that sets the color to be red with a wire gauge of 16.



Customizing the User Interface

Now we have created our trail files we create a button for them on the diagramming tool tray. We are going to use some user-defined icons. To do this we can use gif files and place them in resource directory (you may need your system administrator to do this for you)

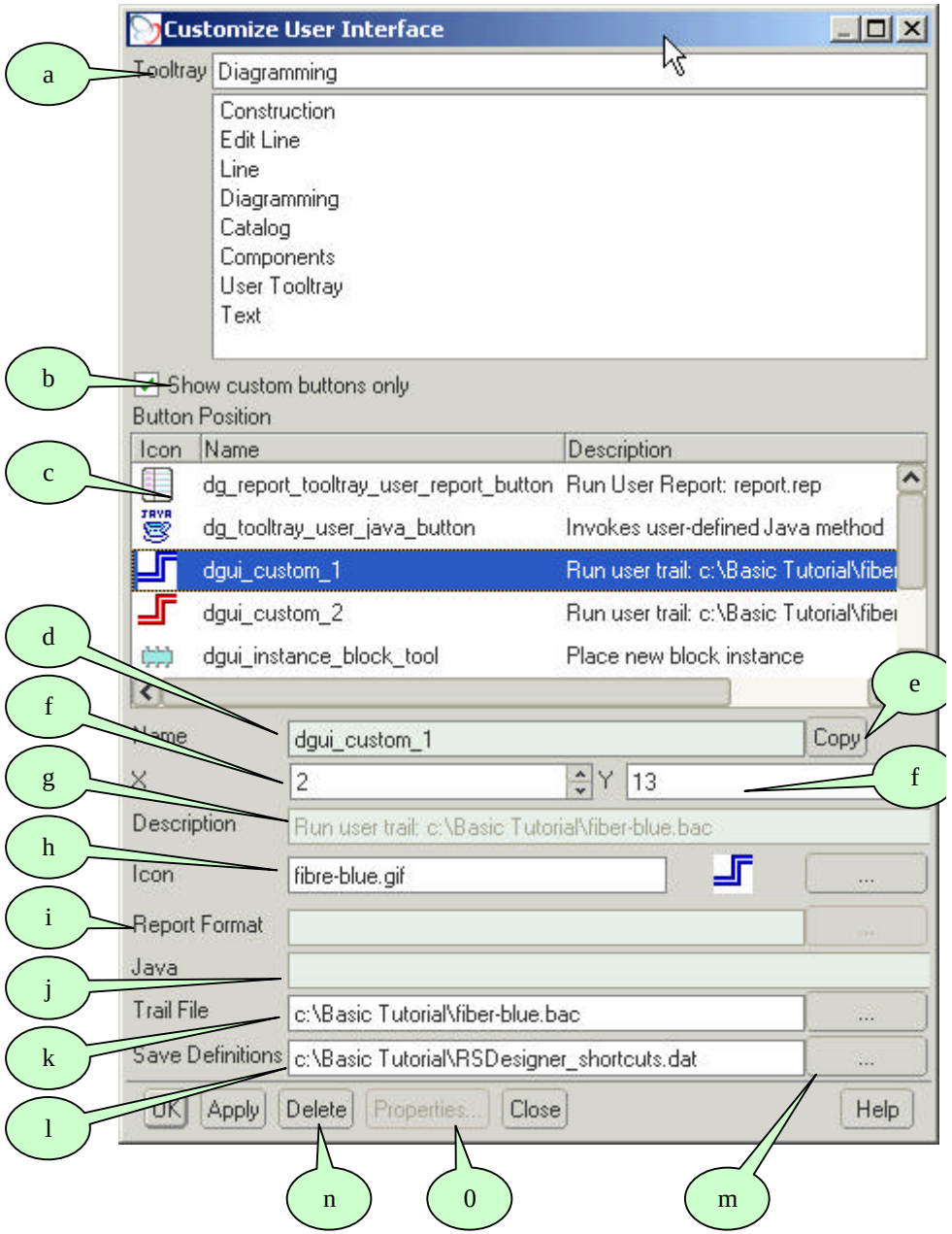
1. Within windows explorer **copy** the 5 gif files (fibre-black.gif, fibre-blue.gif, fibre-red.gif, fibre-yellow.gif, fibre-green.gif) in the icons directory **to** <loadpoint>\i486_nt\rsdesigner\ui\cursors\text\resource

About Customizing the User Interface

The User interface can be customized by clicking Utilities > Customize UI

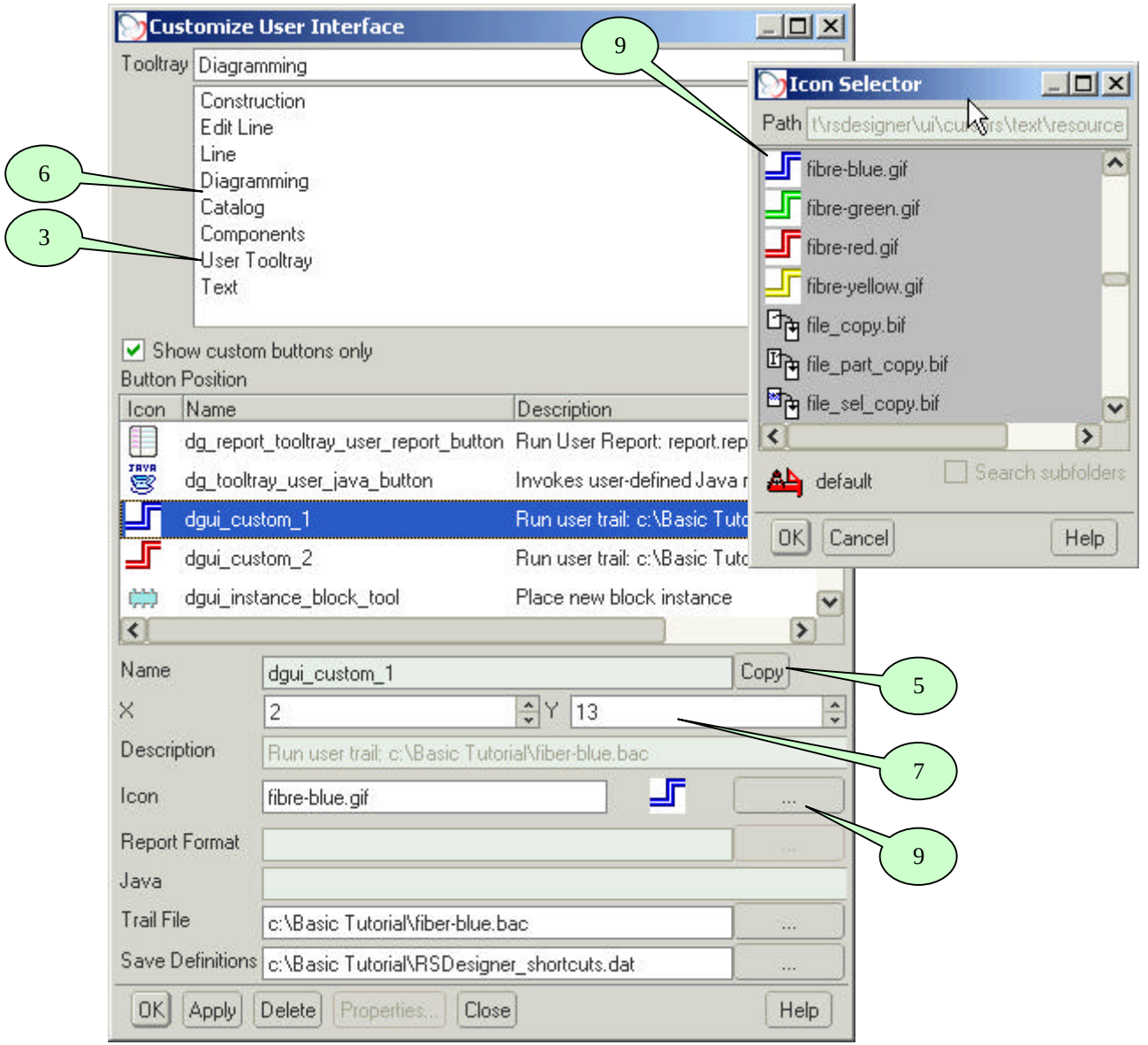
The dialogue displays the following

a) Tooltray : Displays the currently selected tool tray. To change the current tool tray, select the required tool tray from the list.
b) Show custom button only : Restricts the display in the list below this check box to only those buttons that can be copied.
c) Button Position : Displays the buttons in the current tool tray. When you select a button in this list, the relevant information is displayed in the boxes below the list.
d) Name : Displays the name of the button
e) Copy : Copies the button that is currently displayed. To make a copy of the button in another tool tray, use Tooltray to change the current tool tray and click Copy
f) X, Y : Specifies the X and Y position in the tool tray measured from the top left side.
Note : Gaps are not supported in the X or Y sequence of button positions for a tool tray. Any gap in the sequence is automatically removed.
g) Description : Displays the description of the button
h) Icon : Displays the icon associated with the button.
i) Report Format : Displays the report format associated with the button. This option is enabled for report buttons only.
j) Java : Specifies the name of the Java procedure that the button calls. This option is enabled for Java buttons only.
k) Trail File : Specifies the name of the trail file to be run by the button. This option is enabled for trail file buttons only.
l) Save Definitions : Specifies the location where the customization information is stored.
m) The … buttons on the right of the fields can be used to Set values
n) To delete a button, click Delete
o) The Properties button is used to set the item that will be instanced for an instance shortcut button.



Creating Button

2. **Select** Utilities > Customise UI
3. **Select** User Tool Tray
4. **Select** dgui_default_trail_button
5. **Select** Copy
6. **Select** Diagramming
7. **Enter** 13in Y
8. **Select** Apply
9. **Select** icon fibre-blue.gif
10. **Select** trail file c:\fibre-blue.bac
11. **Select** Apply
12. **Repeat** steps for the red fibre (place in X1 Y13
13. **Select** OK to overwrite shortcuts.dat



Mapkeys

About Mapkeys

Mapkeys are unique, and their stem must not consist of other mapkey sequences. For example, the following are valid sets of mapkeys:

Valid mapkeys	`nd, no, hell, help`
Invalid mapkeys	`h, hi, hello, no, nod`
Valid mapkey characters for use with the calculator	`+, -, *, /, =, .`

Note: The calculator, by default, can be controlled by the keyboard.

You can Set mapkeys for the following buttons through the `mapkey.dat` configuration file.

- Choice
- Toggle
- Action
- Tool

The `mapkey_sequence` is either a string of alphanumeric characters or of the form `$f1` to `$f12` for function keys. You cannot combine function keys with other keys. You can only have a single function key for an action.

Mapkey functions may not always be available. There are times when keyboard input must go to the text object, rather than be targeted to a mapkey sequence, for example, a tool prompt for parameter input in the main window status area.

A part of an application cannot assume what mapkeys are used elsewhere in the application. Two parts may use the same mapkey. Therefore, mapkeys must be taken as a whole by the application. Mapkey functions are an application-wide utility

To Create a Mapkey

- Click Edit > Mapkeys. The Mapkey Editor dialog box opens.
- Drag  and place it over the required menu button. The menu button name appears in the Button.
- In Mapkey, Enter a mapkey sequence by typing it or selecting it from the keypad in the Mapkey Editor dialog box.
- On Entering further mapkeys, keys on the keypad may turn blue indicating that they have already been used and that the key sequence may not end with that particular key. For example, if a mapkey sequence `we` already exists, on Entering `w` as the first key, `e` disappears from the list. On entering another key, say `c`, `e` can be selected again.
- A  indicates that the currently shown mapkey is valid. A  indicates that the currently shown mapkey is invalid.
- To delete the last character from the mapkey sequence, Click  .
- Click OK.

To Delete a Mapkey

- Click Edit > Mapkeys. The Mapkey Editor dialog box opens.
- Select the mapkey to be deleted.
- Right-Click and Select Delete Mapkey.
- Click OK or Click Close to close the dialog box.

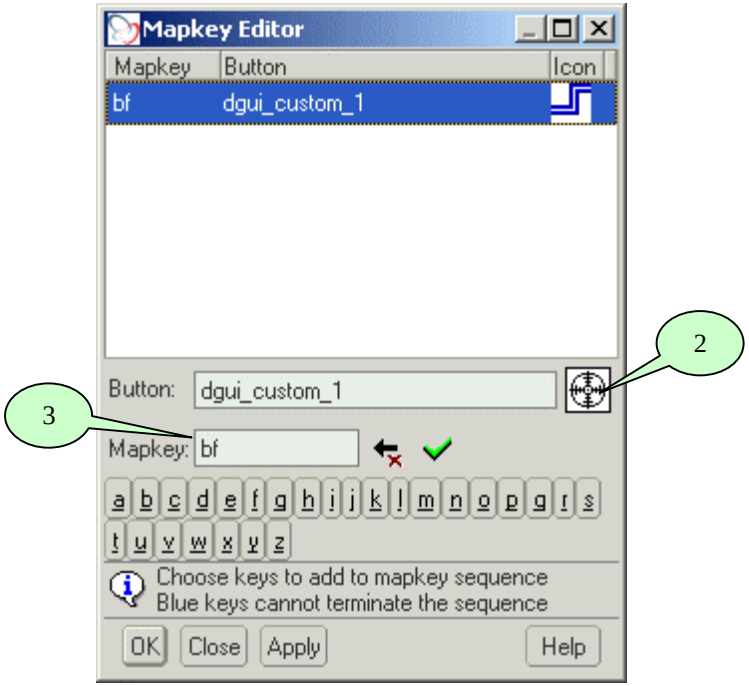
Creating a Mapkey

We can now automate the trail file even further by creating a mapkey

1. **Select** Edit > Mapkeys
2. **Select**  with the LMB, > Keeping the LMB pressed drag the cursor over the blue fibre icon and release.

You will now see that the button entry has populated with dgui_custom_1

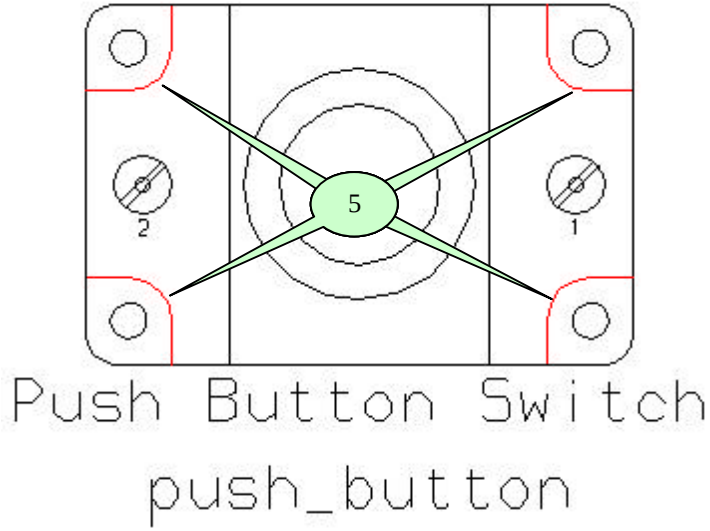
3. **Enter** BF for the mapkey sequence
4. **Select** Apply
5. **Repeat** steps 2 to 4 for the red fibre
6. **Use** one of the new mapkeys to Set W-06



Updating Prototype Instances

To update instances of the Selected prototype shape with the current state of the prototype, **Select**  from the Catalog Explorer. The Update Instances dialog opens. You can update all instances on the current sheet, on all sheets, or Selected sheets.

1. **Select** SW-01 > **Select** CTRL S to Zoom in
2. **Open** the Catalog Explorer
3. **Select** Blocks from the pull down menu
4. **Select** Switches > Push Button Switch > 
5. Using the MMB **Select** the four highlighted lines > **Click** RMB > **Select** Delete
6. **Select**  to Close the shape
7. **Open** the Catalog Explorer 
8. **Select** the block shape we have just edited
9. Select 
10. **Select** Update all Instances on All Sheets > **OK**
11. **Close** the Catalog Explorer



Notice that the switch shape has updated

About Updating Variable Groupshapes

You can reorder variable shapes, even when other shapes of the same group exist on other sheets. However, the other shapes are not automatically updated.

Click  to update the variable groupshapes on the current sheet.

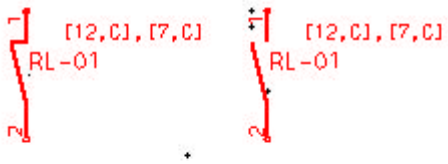
Click  to update the variable groupshapes on all the sheets in the current design.

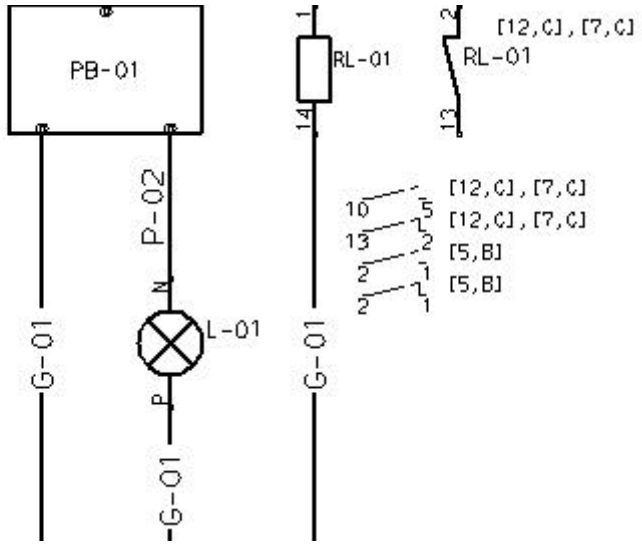
Click  to update the Selected variable groupshape.

Note: A shape is automatically updated when it is reordered.

Changing the Circuit Diagram

1. **Open** the circuit diagram
2. **Select** RL-01
3. From the Diagramming Tool Tray Select 
4. **Select** Variable Relay!ncc1_1
5. **Place** onto the sheet
6. **Select** Variable Relay!noc1_1
7. **Place** onto the sheet
8. From the Select member shape to place dialog **Select** cancel
9. From the **Diagramming Tool Tray Select** update cross refere
10. From the **Diagramming Tool Tray Select** update multiple sh 
11. **Open** the wiring diagram

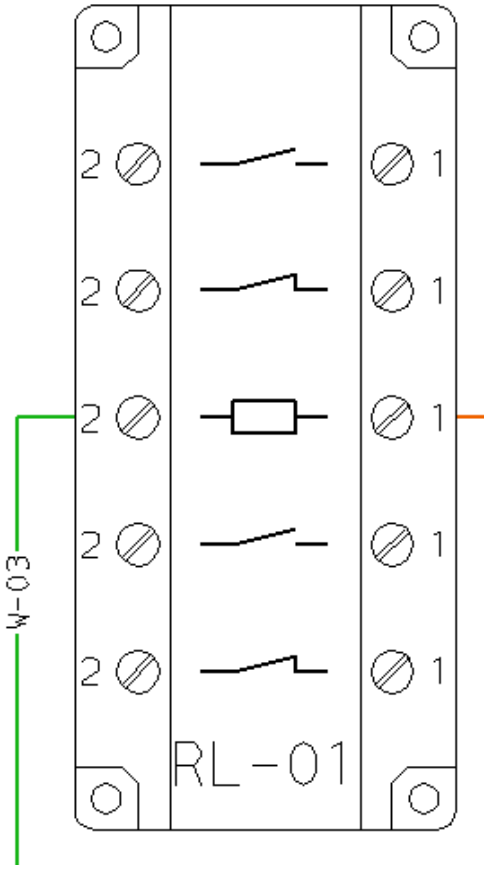




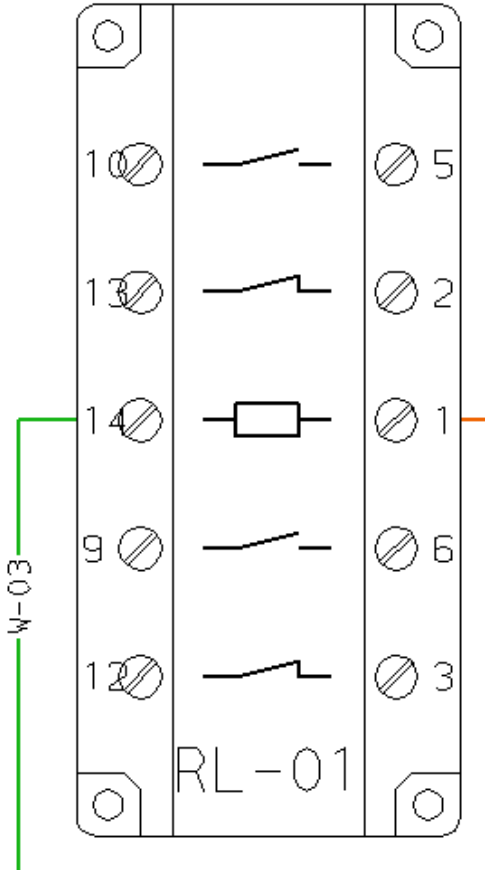
Notice that the wiring shape of the Relay has updated

Applying data sets

1. **Select** RL-01> **Select** CTRL S
2. **Click** RMB > **Select** Properties
3. **Select** Apply Dataset
4. **Enter** 2 for ncc_count and 2 for noc_count
5. **Select** update
6. Select OK
7. **Select** Apply
8. Select OK
9. **Notice** that the port name have updated



Before



After

About Adding, Removing, and Reordering Members in Variable Groups

After you have created an instance of a variable group, you can add, remove, or reorder members of that instance. You can also reorder variable groupshapes that have multiple instances of the shape on multiple sheets.

- Select the group using the Selection tool.
- From the right mouse button menu, Click Reorder Members. The New Reorder dialog box opens.

To Add a member

Choose a prototype member from the Prototype list. Specify the number of members to be added in the number field and Click Add. The new member appears in the Members list. The number of members of a particular prototype that have been added to a group are displayed in the Added column.
To add one member of all the prototypes displayed in the top list, Click Add All.

To remove a member

Select the member you want to delete and Click  .

To reorder a member

Select the members to be reordered. The following Selection tools help in easier and quicker Selection.

 Clear the Selection.

 Invert the Selection.

 Select all members.

 Cutaway members.

Any members located between the first and last Selected member are set to be visible while the rest of the members are Set to be cutaway. Members that are cutaway are marked with an asterisk (*) in the Cut column.

 Make all members visible.

 Specify a cutaway range.

Specify the range of members that are to be made visible in the Cutaway visible cells dialog box.

 Move to the top of the list.

 Move up one level.

 Invert the order of Selection.

For example, if you have Selected members A, B, C, and D in the list box in that order, this option changes the order to D, C, B, and A.

 Move down one level.

 Move to the bottom of the list.

 Modify width of instanced cells.

In the Enter Cell Width dialog box, you can enter the cell width or enter a factor by which the current width is to be modified. When a cell width is changed, the cell decoration graphics are multiplied or truncated along the group definition direction vector so that the new cell has a continuous decoration pattern throughout. The width of the instanced cells is measured in instance sheet units while the width of the prototype cell is measured in prototype sheet units.

 Cut Selected member.

 Copy Selected member.

 Paste copied or cut member.

The new member is inserted at the position after the last Selected member. If no cells are selected, new members are inserted at the top of the list.

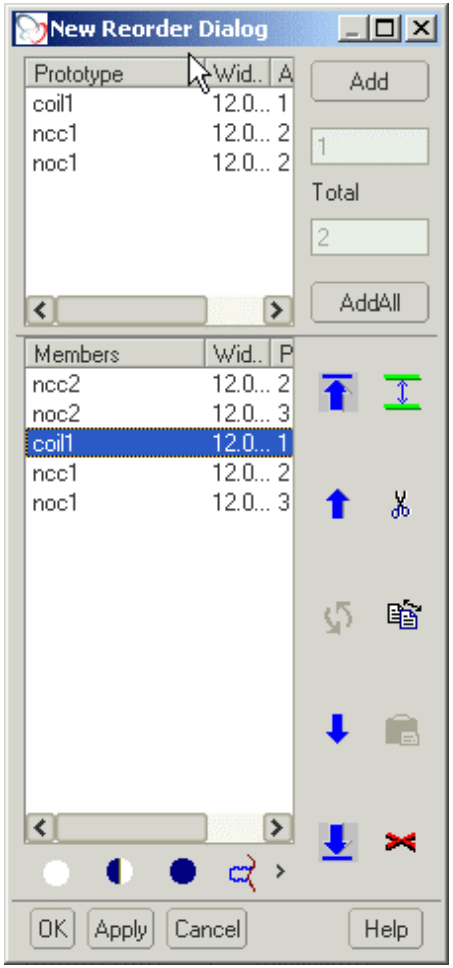
When satisfied with your changes, Click Apply. Note the operation of Apply in the following modes:

- Instancing in the Place mode—Clicking Apply updates only the image displayed on the cursor. No database changes occur until you instance the group.
- If the Reorder Update option in the Options > Switches > Switches Control Panel is selected, the image displayed on the cursor is updated automatically. If the Reorder Update option is not selected, the image on the cursor is not updated automatically.
- Reordering instanced groups in the Reorder mode—Clicking Apply applies the changes to the instance.
-

If you have Selected the Reorder Update option in the Options > Switches > Switches Control Panel, the graphics of the group are updated as you make changes in the dialog box. If you have not Selected the Reorder Update option, the graphics are updated only when you Click Apply.
Note: Clicking OK in both the Place and Reorder mode is equivalent to clicking Apply followed by Cancel.

Re-ordering RL-01

1. **Select** Rl-01 > **Click** RMB > **Select** Reorder members
2. **Select** Coil 1
3. **Select**  to move the member to the bottom of the group
4. Select OK



Creating a cable definition

We now want turn the 3 of the wires into a cable.

About Cables

In the physical world, cables are conductors comprising individually insulated cores. In Routed Systems Designer, cables are represented by groups of fibers. The cable group carries physical property information for the cable itself and for each fiber in the group. The fibers in the cable group represent the cores and carry the properties of the cores.
To add information or parameters associated with individual cores in the group, use a datatable imported into the group, or Set properties on the cable's cores.
You can use the cable tools in the Diagramming Tool Tray to perform the following tasks:
Instance cables on a sheet

- Add, edit, and remove decorations
- Add and remove cores
- Add shields
- Disband that is, remove all cores (or child cables) from a cable and then remove the cable.

Methods of instancing cables

You can create a cable on a sheet by:

- Bundling cores
- Creating a predefined cable on the sheet

Bundling cores

In this method, cores are placed individually on the sheet. They are then Selected and bundled together into a cable. Cable decorations are then applied manually. This method allows for creation of cables on an ad-hoc basis and is suitable for creation of harnesses.

Creating a predefined cable

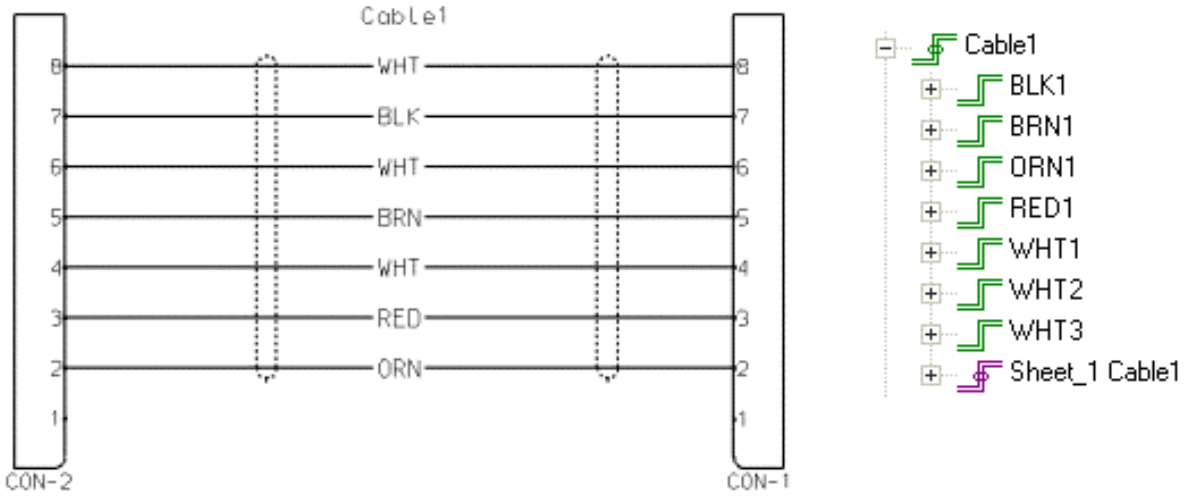
In this method, the number of cores and their decorations are defined in the catalog first. You can then choose where the cable is to be routed and Routed Systems Designer automatically draws the cores and the cable decorations. This method is most useful for routing between connectors.
You can instance a predefined cable as follows:

- Routing one core of the cable and the remaining cores are created automatically
- Selecting two connectors to connect between
- Selecting the ports on connectors to connect between

The predefined cable to be instanced can be a multi-level cable.

Example: Simple Cable

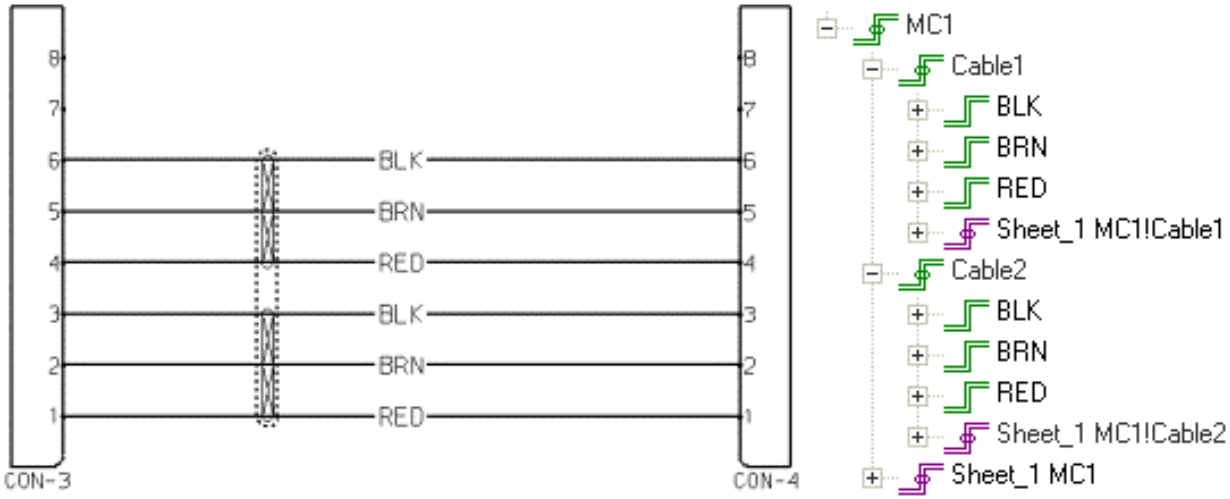
The following is an example of a simple cable with two decorations. The structure of the cable, as seen in the Model Explorer, is shown on the right.



Cable1 has seven cores and is displayed with two cable decorations surrounding all the cores. The Model Explorer shows the entry for the shape on the cable.

Example: Multi-level Cable

The following is an example of a multi-level cable. The structure of the cable, as seen in the Model Explorer, is shown on the right.



Cable MC1 bundles two 3-core cables, Cable1 and Cable2. The Model Explorer shows the shapes for MC1, Cable1, and Cable2.

To Instance a Cable By Bundling Cores

- Select the cores to be bundled into a cable.
- Click  in the Diagramming tool tray. The Select Cable dialog box opens. A list of cables that can be created is displayed.
- Select a cable and Click OK. The cable is created and the wires are added to it.

Note: If the cable is a predefined one with decorations, the decorations are drawn on the cable if the prototypes of the cores match those used in the predefined cable. If the prototypes do not match, the Add Cable Decoration mode is started and you can add a decoration to the cable.
Also, if the cable is a predefined one and has more cores than were Selected, the remaining cores are created as spares. You can view them in the Model Explorer or in a report.

To Instance a Cable By Creating a Predefined Cable

Routing a single core

- Click  .
- Right-Click and Select Deselect All.
- From the Catalog Explorer, choose a cable to instance. You can also use the group shortcut tool in the Diagramming tool tray.

The cable instancing works like the fiber-instancing tool to route between ports on connectors or in free space. Once the core has been routed, the other cores are created automatically and cable decorations are applied.
Note: As many cores will be created as there are on the connectors that the first core is routed between. Any remaining cores will be created as spares with no shapes, which can be viewed in the Model Explorer or in a report.

Routing between two connectors

- Select the two connectors that the cable is to be created between.
- Choose the cable place tool from the Catalog Explorer. You can also use the group shortcut tool in the Diagramming tool tray.

The cable is automatically created and cable decorations are applied.
Note: As many cores will be created as there are on the Selected connectors. Any remaining cores will be created as spares with no shapes, which can be viewed in the Model Explorer or in a report.

Routing between Selected ports

- Select the ports that the cable is to be routed between. The ports may be on more than two connectors.
- Note: One end of the cable must be routed from a single connector, so the number of cores that will be drawn will be the highest number of ports Selected on one connector.
- Choose the cable place tool from the Catalog Explorer. You can also use the group shortcut tool in the Diagramming tool tray.

The cable is automatically created and cable decorations are applied.

Note: As many cores will be created as the number of ports that are Selected on one connector. Any remaining cores will be created as spares with no shapes, which can be viewed in the Model Explorer or in a report.

About Spare Cores

When a dataset is applied to a cable, any core specified by the dataset that does not exist in the cable, is created as a spare core. This spare core is an instance of the spare core catalog fiber specified in the Design Properties dialog box.
The Spare Core field in the Design Properties dialog box displays the name of the catalog fiber that has been designated as the spare core. To designate a different catalog fiber as the spare core, Click … and Select the required catalog fiber. If a spare core has not been designated, Routed Systems Designer creates one, if required.
Note: Spare cores can also be created when a predefined cable is instanced but not all the cores have shapes. This can occur when the connectors that are routed between do not have as many ports as there are cores in the cable.

About Shields

A shield is an extra core that has a different graphical representation and type from the other cores in a cable. Shields are usually drawn from cable decorations and can be connected to components in different ways.
A shield can be identified by its type, that is, `connection!wire!wiring_wire!shield`.
Note: Only one shield can be added to a cable at each level in the group hierarchy. Use the Model Explorer to view the group hierarchy and to see which cables have shields.

Ways in which a Shield can be connected

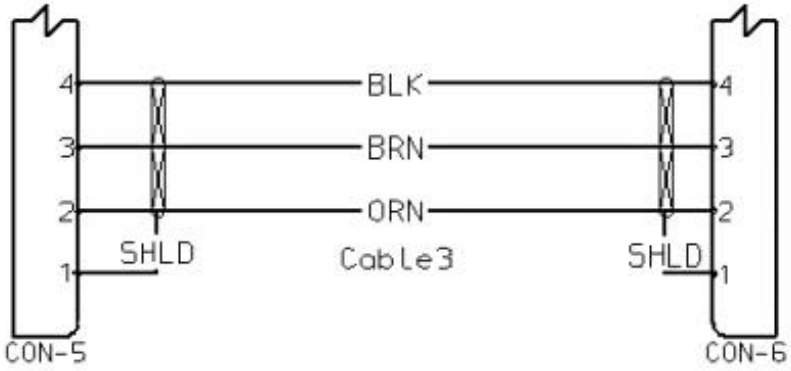
Shields can be connected in the following ways:
- Shield not connected
- Shield connected to back shell
- Shield connected through connector
- Shield connected to the ground
- Shield connected to the daisy chain

Reporting Shields

Shields can be reported in a From-To list in the following ways:

Direct Link

The following figure shows an example where the shield wire is reported as a direct link between two connectors.

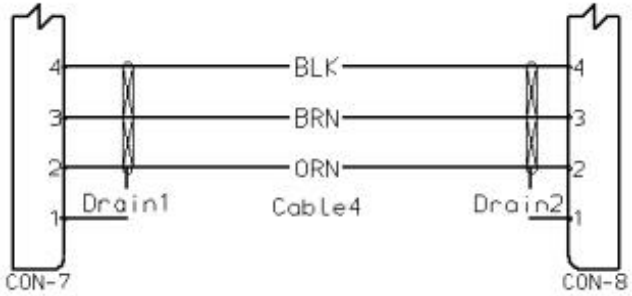


The shield can be reported in the following way:

	From		To	
Wire	Component	Pin	Component	Pin
Cable3!SHLD	CON-5	1	CON-6	1

Through drain wires

The following figure shows an example where the shield wire is reported through drain wires.



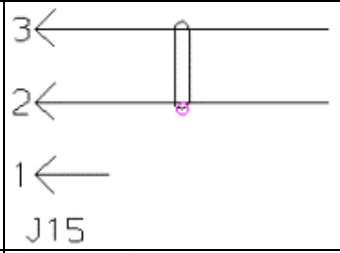
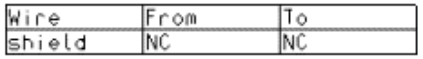
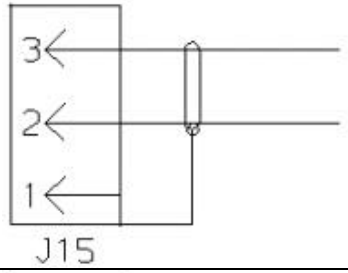
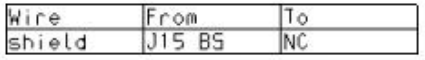
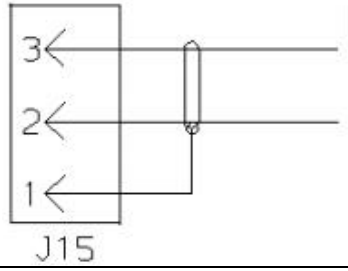
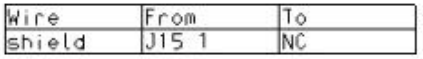
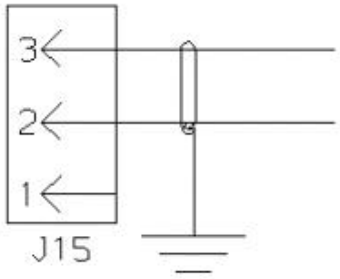
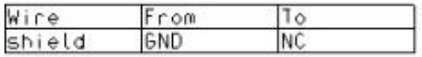
The shield connections can be reported in the following way:

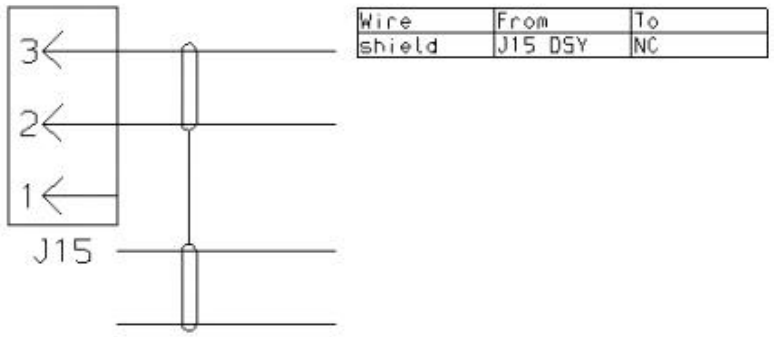
	From		To	
Wire	Component	Pin	Component	Pin
Drain1	CON-7	1	Cable4	SHLD
Drain2	CON-8	1	Cable4	SHLD

From this report it is clear that extra wires are attached to the connectors and subsequently attached to the shield of Cable4.

Example: Ways in Which a Shield Can be Connected

Examples of the different ways in which a shield can be connected on a diagram are shown below.

Connection Type	Diagram and Run List
Not connected	 
Connected to back shell	 
Connected through connector	 
Connected to the ground	 

Connected to the daisy chain	

To Add a Shield to a Cable

To draw the shield port shapes

To draw shields on the cable shape, the shield must exist at the artifact level of the cable. This is achieved by drawing a shield core in the catalog cable before it is instanced or by applying a dataset to the cable with a member map that specifies a shield.

- Select the cable to which you want to add the shield.
- Click  in the Diagramming tool tray.
- Place the shield on a decoration in the Selected cable.
- Note: The probing mechanism automatically chooses a suitable position on the cable as you move the cursor near it. If you would like to use the probe tools instead, right-Click and choose Use Probe Tools.
- Right-Click and choose Exit to stop adding shield port shapes.

After the shield port shapes are placed on cable decorations, you can route shield wires as necessary. You can also add multiple shield port shapes to a decoration thereby allowing for daisy chaining of cable decorations.

To draw the shield wires

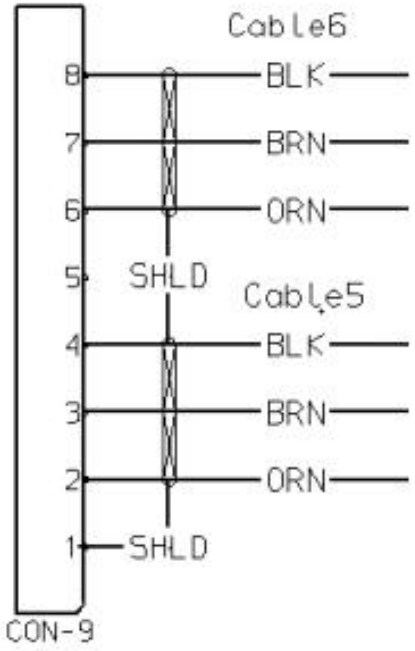
Shield wires can be drawn as follows:
Direct link

- Select the cable from which the shield wire is to be drawn.
- Click  in the Diagramming tool tray.
- Click on the shield port shape of a decoration to start the fiber.
- Create a normal fiber.
- Click on the port of a component to terminate the fiber (or) Click in free space, right-Click, and Select Terminate.

Through drain wires
Drain wires are created by routing a standard wire from the shield port shape on a cable decoration to the point where the wire is to terminate.

Example: Multiple Shield Port Shapes

A sample of a cable decoration with multiple shield port shapes is shown below. Adding multiple shield port shapes allows for daisy chaining of cable decorations.



To Create a Cable Group Definition

You can instance a cable by bundling existing fibers or by creating a complete cable.

To create a cable in the catalog for bundling cores together

- Open the Catalog Explorer and Select Groups from the list.
- Click  . A new group is added to the list with a default name.
- Click  . The Properties dialog box opens.
  - Specify the name the cable and Set the type to connection!cable, or its subtype.
  - Select a datatable for the cable if necessary.
  - Click OK to close the dialog box.
  - Click  in the Catalog Explorer. A new shape is added to the Group Shapes list with a default name.
  - Click  . The shape opens in the graphics window.

- A cable prototype for bundling cables without a shield should not have any children. If the cable must have a shield (screen), draw a short piece of fiber in the group shape to represent the shield wire type. The type of the fiber should be connection!wire!wiring_wire!shield, or its subtype.
- Close the shape. The group is complete.

To create a Predefined cable in the catalog

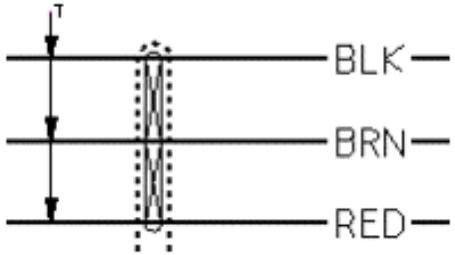
- Create a new fiber and Set the property type to CONNECTION!WIRE!WIRING_WIRE.
- Click  in the Catalog Explorer and create a shape for the fiber. A new shape is added to the Fiber Shapes list with a default name.
- Create as many fibers as are required for the different types of cable core.
- Switch to the Groups folder in the Catalog Explorer.
- Create a group in the Catalog Explorer and Set the property type to CONNECTION!CABLE.
- Click  in the Catalog Explorer to create a shape for the group. A new shape is added to the Group Shapes list with a default name.
- Click  . The shape opens in the graphic window.
- Use construction lines (3 vertical lines) to represent the edges of the connector and the middle of the cable.
- Draw primary cores, other cables (to make multi-level cables), and shield cores as follows:
  - Click  in the Catalog tool tray. The Select a core fiber dialog box opens.
  - If you want to instance a fiber, select a core to be placed.
  - If you want to place a cable or a shield, Click Cancel. In the main window, right-Click and select new cable or Select shield core. Then, Select the required object.
  - The new core or cable is automatically shown between the rightmost and leftmost construction line and follows the vertical movements of the cursor.
  - Place the new core as required.
  - When all cores have been placed, right-Click and Select Exit.
- Add cable decorations and shield ports if required.

Cable decorations placed between the left and middle constructions lines apply to the start side of the fiber route. Cable decorations placed between the right and middle constructions lines apply to the end side of the fiber route. This allows the user to create a mirrored (or non-symmetric) look for multi-decorated cables.

- Click  and place the datum for routing. Choose one of the cores as the core to route when instancing the cable and place the datum on it to signify the choice.

About Cable Decorations

When a cable is created with cores, you can **add** a cable decoration to give a graphical grouping of the cores of the cable. Cable decorations can be added and modified on cables both in the catalog and on design sheets.
Some sample cable decorations are shown below (for twisted triple, twisted overall and over braiding).



Shields are usually drawn from cable decorations.

To Add a Cable Decoration

- Select the cores of the cable to which the decoration is to be added. If a cable splits then only select the cores that split from the main cable.
- Click  in the **Diagramming Tool Tray** to add a decoration.
- Right-Click and Select cable geometry. The Select Cable Geometry dialog box opens.
- Select the type of cable decoration symbol and Click OK.
- Click in the main window to place the cable decoration symbol.

Note: All Selected fiber segments must be parallel to be decorated.

To Change the Type of a Cable Decoration

- Select the cable with the decoration to be edited.
- Click  in the Diagramming tool tray.
- With the cursor over the decoration, use the shortcut menu to **select** alternative cable decoration symbols.

Note: You cannot change cable decorations with shield port shapes**.**

To Move a Cable Decoration

- Select the cable with the decoration.
- Click  in the Diagramming tool tray.
- Select the decoration and drag it to the new position.

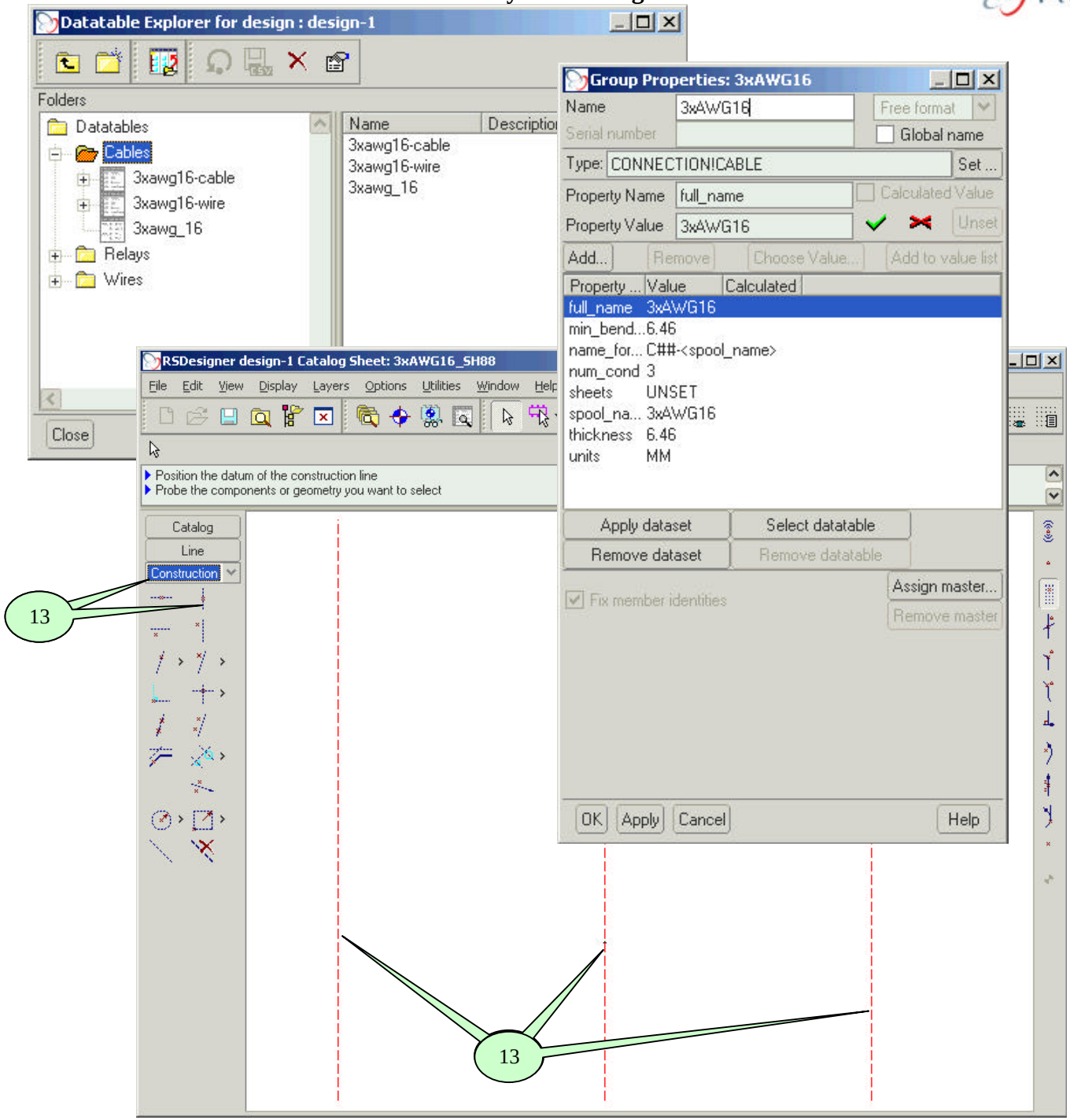
Note: All fiber segments Selected must be parallel to be decorated.

To Delete a Cable Decoration

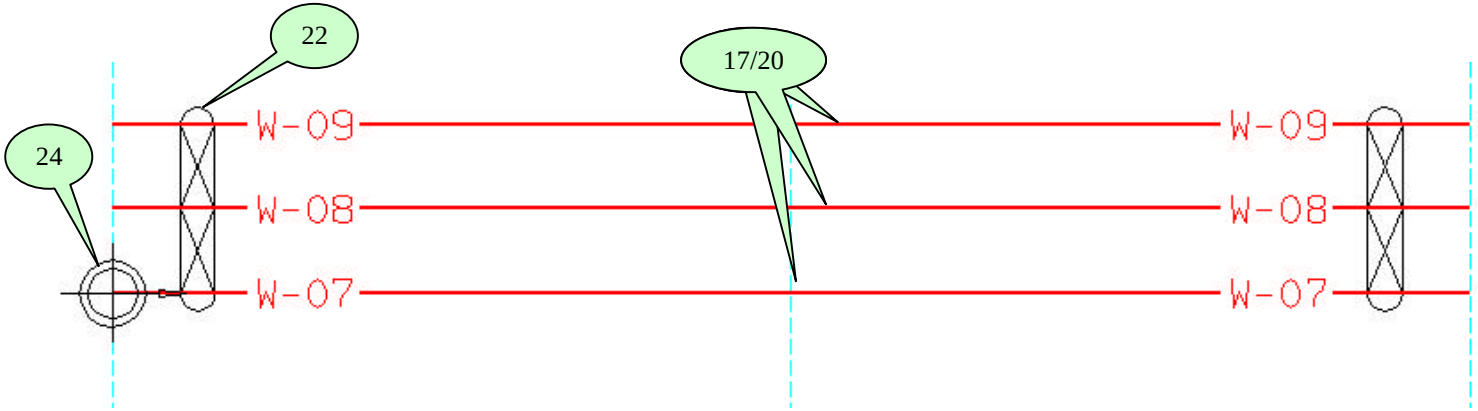
- **Select** the cable with the decoration to be deleted.
- **Click**  in the Diagramming tool tray.
- **Select** the decoration to be deleted.

Creating a 3 Conductor Cable

1. **Open** the Datatable explorer
2. **Create** a new folder and re-name to Cables
3. **Load** dataset 3xawg16-cable.csv from CSV files\cable dataset \ 3 conductor
4. **Open** the catalog explorer
5. **Select** Groups from the pull down menu
6. **Create** a new folder and rename to cables
7. **Create** a new Group
8. **Rename** the group to 3xAWG16
9. **Select** Datatable 3xAWG16
10. **Apply** the new dataset
11. **Select** OK
12. **Create** a new group shape & Open
13. **Select** the Construction tool tray and place 3 vertical construction lines

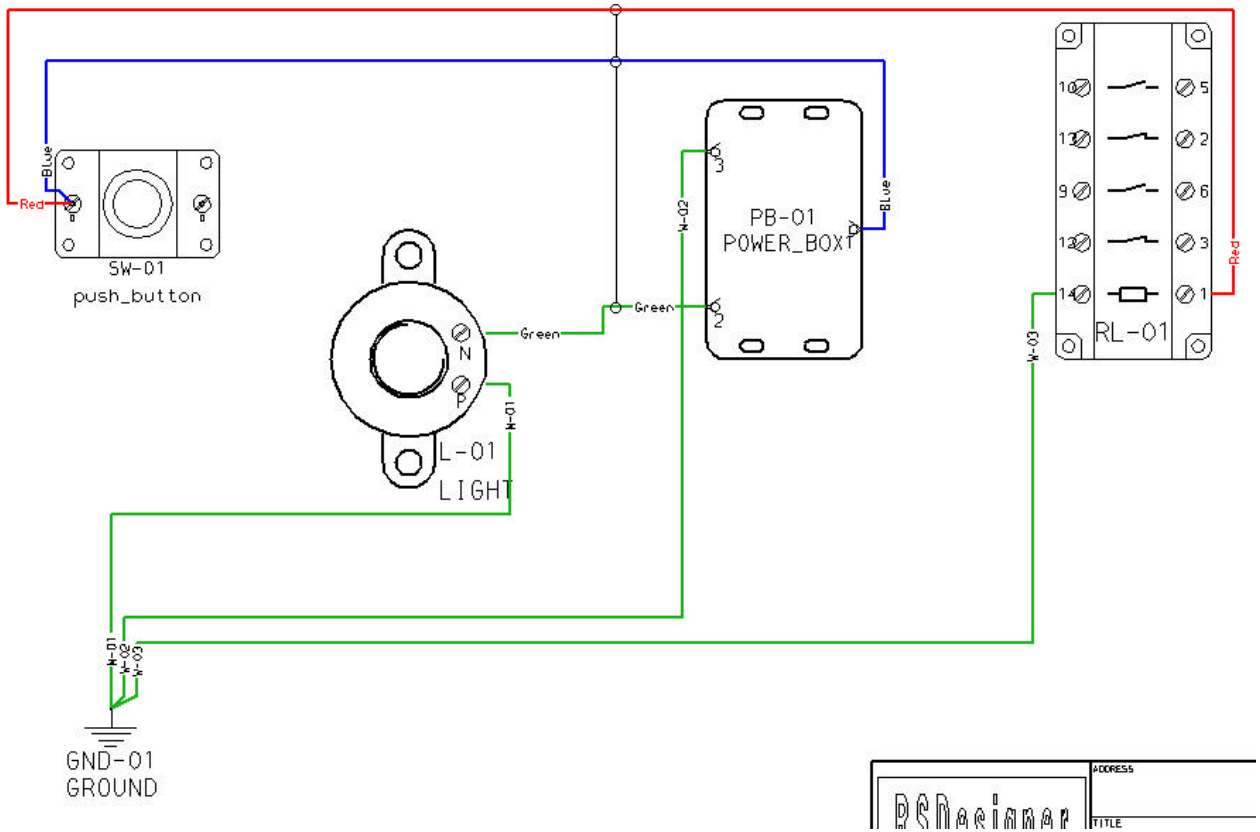


14. **Select** the Catalog tool tray
15. **Select** Define the Contents of a Cable Icon
16. From the catalog explorer **Select** Template Wire
17. **Place** the fibre 3 times
18. **Select** RMB > Exit Tool
19. **Select** the Diagramming tool tray
20. **Select** the 3 placed fibres
21. **Select** the Add Cable Decoration to Selected Cables
22. **Place** the cable decoration that is on the cursor on the left of the cable
23. **Repeat** Steps 21 –22 to place a second cable decoration on the right of the cable
24. **Set** the datum to be at the bottom cable on the left. By placing the datum point at the bottom the cable will be created upwards



Using the new cable

25. **Select** Wires W-05 & W-06
26. From the Diagramming menu **Select** create cable 
27. **Place** the decoration
28. Now we have created a2 conductor cable, we now want to **Add** the third conductor
29. Select W-06
30. **Select** Add Selected cores to cable 
31. **Select** the cable decoration
32. **Re-Select** the cable > **Select** delete cable decoration 
33. **Select** the cable decoration
34. **Select** all the cores of the cable
35. **Select** Add cable decoration  > By clicking the RMB you are able to **Select** alternative cable decoration
36. You may want to **edit** the conductor colors.



Setting up for Pro/Cabling

Now that we have finished our wiring diagram lets use the model explorer to set the port information that is required for sending connectivity information to Pro/Cabling

About the Model Explorer

The Model Explorer provides a hierarchical representation of a design and allows you to manage various aspects of the model. You can select multiple objects and perform actions on them. For example, you can Select multiple sheets and delete them.
The Model Explorer consists of the following:

- A Folders tree on the left that displays a hierarchical view of the currently Open design.
- A pane on the right that displays information about the item Selected in the Folders tree. This pane consists of the following tabs. These tabs may or may not be displayed depending on the item selected in the Folders tree.
- Contents—Displays the contents of the item selected in the Folders tree.

If multiple objects are selected in the Folders tree, the contents of all the objects are displayed. By default, the contents are grouped by class and sorted alphabetically.

Note: You can select objects either in the Folders tree or the right pane (within the contents tab). However, only one Selection is active at any point of time. If an item is selected in the right pane, the text in the Folders tree is highlighted in blue.

Preview

Displays a preview of the Selected sheet.
If multiple objects are selected in the Folders tree, the Preview tab is enabled only if all the Selected objects are on the same sheet. If all the Selected objects are on the same sheet, the window zooms such that it contains all the Selected objects.
The Preview tab shows the latest version of the sheet. If the Selected sheet is Open, changes made to the sheet are reflected in the Preview tab. If the Selected sheet is not open, the Preview tab shows the current saved state of the sheet. If an unopened sheet is subsequently opened, the Preview tab reflects the subsequent changes.

Folder Properties, Sheet Properties

Displays the properties of the folder or sheet that is selected in the Folders tree and allows you to modify them. If different types of objects are selected, only those properties that are common to all objects are displayed.

Artifact Properties, Shape Properties

Displays the properties of the artifact or shape that is selected in the Folders tree and allows you to modify them.

Property Set Properties

Displays the common properties of a Selection that includes artifacts, shapes or sheets, in the Folders tree and allows you to modify them.

About Using the Model Explorer

The Model Explorer is available only after you have opened a design. To open the explorer, click  .
You can perform various operations using the icons provided in the Model Explorer. You must first click an item in the Folders tree. Icons are enabled or disabled depending on the item Selected in the Folders tree. A summary of the operations you can perform follows.

 Create a new folder in the Selected folders

 Create a new sheet in the Selected folders

 Delete Selected items

 Move to the common parent of the Selected items

 Cut the Selected item

 Copy the Selected item

 Link the Selected item

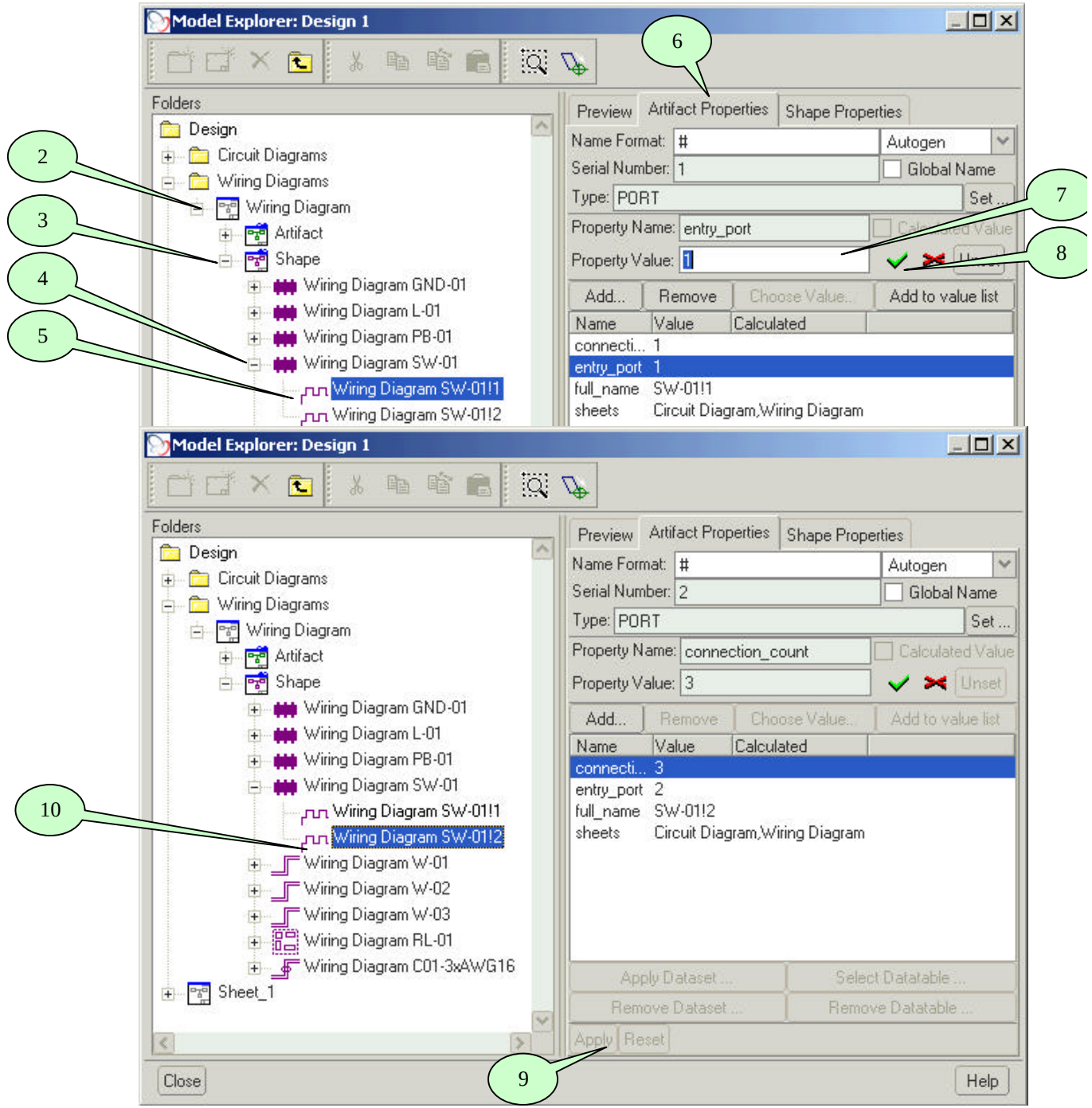
 Paste the current clipboard item

 View the Selected item in the main window

 Place another shape of the Selected item

Setting Port information

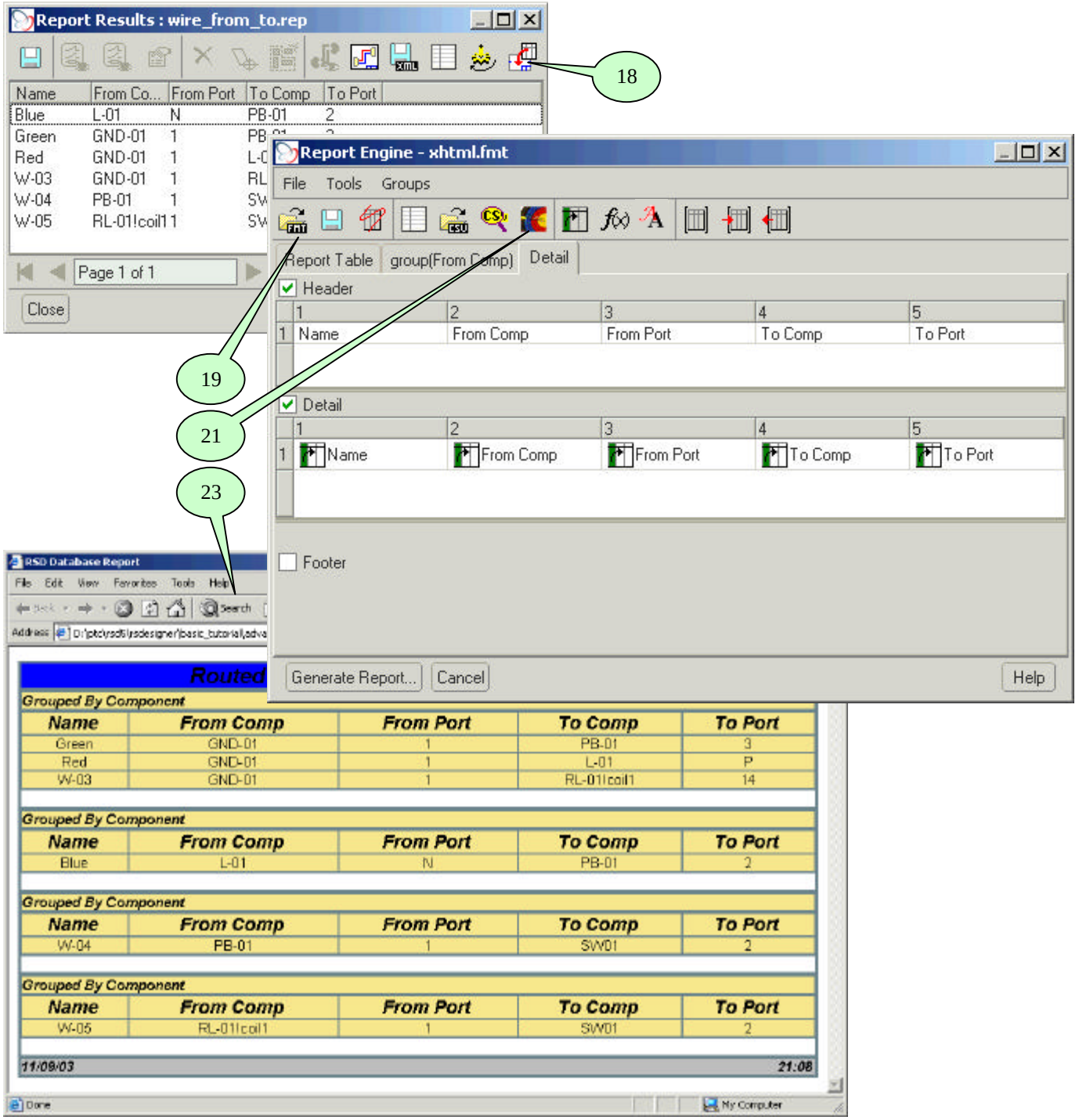
1. **Open** the Model Explorer
2. **Select** Sheet Wiring Diagram & Expand
3. **Expand** folder Shape
4. **Expand** SW-01
5. **Select** Port 1
6. **Select** the Artifact Properties Tab
7. **Set** Property entry_port to 1
8. **Select** to confirm the change
9. **Select** Apply
10. **Select** Port 2
11. **Set** Property entry_port to 2
12. **Select** to confirm the change
13. **Select** Apply



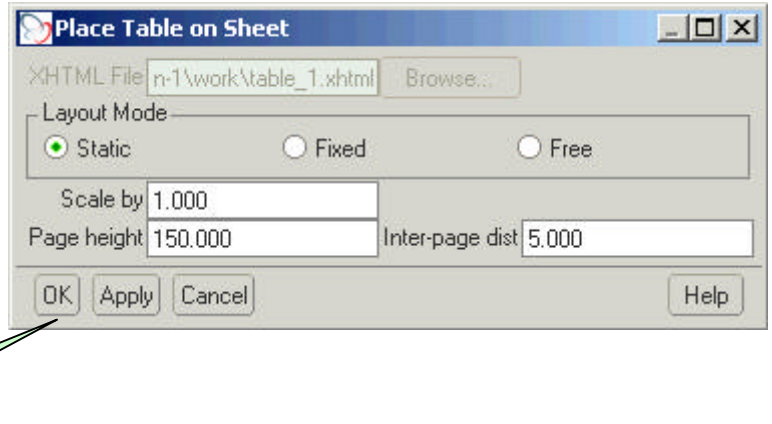
Wiring from to list

To finish our schematic lets create a *from to list*

14. **Select**  from the diagramming tool tray
15. **Select** to generate a new report 
16. **Open** wiring_from_to.rep from the reports folder
17. **Investigate** the column tab for the settings
18. **Select** Generate a report table icon
19. **Select** Load Report Layout Icon
20. **Load** XHTML.fmt from the report directory
21. **Select** Open Target Selector Icon
22. **Select** the XHTML Tab
23. **Set** the destination of the file
24. **Select** Generate Report
25. **Open** the generated report in a Web Browser



26. Load a new report format named Sheet.fmt
27. **Select** Open Target Selector Icon
28. **Select** the Sheet Tab
29. Select Generate Report
30. Select OK from the Place Table on Sheet Dialog
31. Place the table on the sheet



31

Routed Systems Designer 5.0 Basic Tutorial

Grouped By Component

Name	From Comp	From Port	To Comp	To Port
Green	GND-01	1	PB-01	3
Red	GND-01	1	L-01	P
W-03	GND-01	1	RL-01!coil1	14

Grouped By Component

Name	From Comp	From Port	To Comp	To Port
Blue	L-01	N	PB-01	2

Grouped By Component

Name	From Comp	From Port	To Comp	To Port
W-04	PB-01	1	SW01	2

Grouped By Component

Name	From Comp	From Port	To Comp	To Port
W-05	RL-01!coil1	1	SW01	2

11/09/03 21.18

Exporting XML Data to Pro/Cabling

About Exporting a Design to Pro/CABLING

To export a Routed Systems Designer design to Pro/CABLING as an XML file, you must specify certain properties for connectors, spools, and cables. Specifying these properties enables you to use the logical reference seamlessly.

Connectors

Name	Description	Type	Value_list
model_name	Pro/ENGINEER part name.	string	
def_internal_len	Default length of internal connector.	string	
def_grouping	Default connecter grouping.	string	[round, flat, wire]
num_of_pins	Number of pins in the connector.	string	
obj_type	Type of component.	string	[component, connector, wire, prefab]

Cables

Name	Description	Type	Units
spool_name	Name of the cable spool.	string	
num_cond	Number of individual conductors.	real	
min_bend_radius	Minimum bend radius.	real	
thickness	Outside diameter of the cable.	real	
units	As required, millimeters or inches.	real	MM

Ports

Name	Description	Type
entry_port	Name of the Pro/ENGINEER csys the wire will attach to.	string

Wires

Name	Description	Type	Units
spool_name	Name of the wire spool.	string	
min_bend_radius	Minimum bend radius.	real	MM
thickness	Outside diameter of the wire.	real	MM
obj_type	WIRE.	string	
units	As required, millimeters or inches.	real	MM

About Exporting a Design to Pro/PIPING

To export a Routed Systems Designer design to Pro/PIPING as an XML file, you must specify certain properties for connectors and pipes. Specifying these properties enables you to use the logical reference seamlessly.
In Pro/ENGINEER, Schematic-Driven Pipeline Modeling works based upon a relationship with Routed Systems Designer. The information from Routed Systems Designer is used in conjunction with the Pro/ENGINEER SpecDB. If a fitting, pipeline, or piping specification does not exist or is not correct in the Pro/ENGINEER SpecDB, Schematic-Driven Pipeline Modeling does not work. Therefore, ensure that the properties used in Routed Systems Designer match those in the Pro/ENGINEER SpecDB.
In all cases, the Routed Systems Designer property `Name` is used as the component's reference designator in Pro/ENGINEER.

Pipelines (Groups)

Spec	The value of this parameter must match the name of the SPEC parameter defined in the piping_spec_dir.ptd file in the SpecDB. Properties defined in the piping_spec_dir.ptd file take priority over those defined in Routed Systems Designer. Properties read from the XML file are added to the pipeline as user-defined properties.

Equipment

Category	To classify a Routed Systems Designer component as equipment, specify CATEGORY = EQUIPMENT.
Model_Name	This property is required and is used by Pro/ENGINEER to match components in the 3D assembly with those in the schematic during designation. If this property is not specified, match the components manually.

Nozzles

Category	To classify a Routed Systems Designer component as a nozzle, specify CATEGORY = NOZZLE.
Model_Name	This property is required and is used by Pro/ENGINEER to match components in the 3D assembly with those in the schematic during designation. If this property is not specified, match the components manually.

Note: Nozzles often belong to a piece of equipment.

Branches

Category	To classify a Routed Systems Designer component as a branch, specify CATEGORY = BRANCH.
Sname	The presence of this property is NOT required and it should only be used if the actual type of branch is known, i.e.: olet, bran, etc. If this property is used it MUST match the *Sname* parameter found in the piping_mcat_dir.ptd file in the SpecDB for the branch being used.
Model_Name	This property is optional and is used by Pro/ENGINEER to match components in the 3D assembly with those in the XML during designation. If this property is not specified, match the components manually.

Note: Branches can be created in several ways; tees and stub-ins are common branches. Many times, the component to be used in the assembly is not known at the time of the schematic's creation. Therefore, the Model_Name property is not often used and the branch is manually designated in the assembly. In the case of tees, designate the Pro/ENGINEER part as the component. In the case of stub-ins, designate the Pro/ENGINEER point as the component.

Fittings

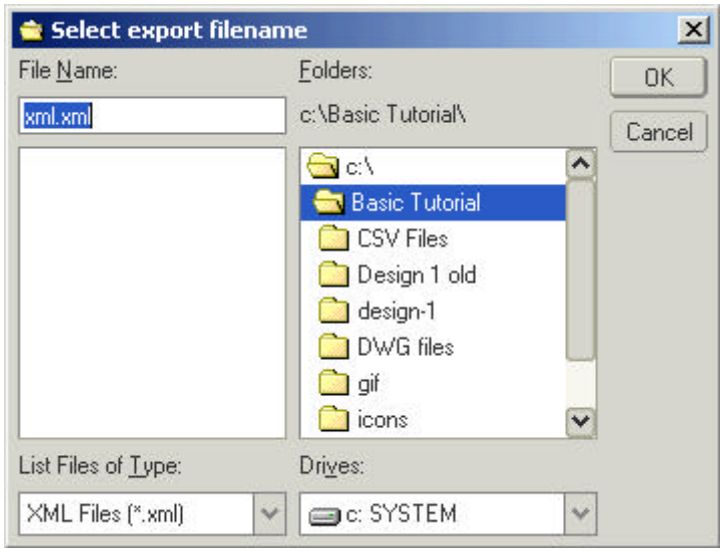
Category	The value of this parameter must match the value found in the piping_mcat_dir.ptd file for the fitting being used. For example a reducer's category property is set to REDUCER. A valve's category property is set to VALVE.
Sname	The value of this parameter must match the value found in the piping_mcat_dir.ptd file for the fitting being used. For example, a concentric reducer's sname property is set to CRED. A gate valve's sname property is set to GATE and a check valve's sname property is set to CHECK as defined in the piping_mcat_dir.ptd file.
Model_Name	This property is required and provides extra insurance that the component selected for insertion in Pro/ENGINEER is exactly the fitting defined in the schematic.
Size	This property is optional but it is recommended to use the Size property as found in each individual fitting's .ptd file in the Pro/ENGINEER SpecDB. This property is used to automatically select the correct fitting for insertion.
Nsize/Bsize	These properties (reducer and branch properties) are optional and can be added to the Routed Systems Designer component as found in each individual fitting's .ptd file in the Pro/ENGINEER SpecDB. These properties are used to automatically select the correct fitting for insertion. If these properties are not specified, you may need to select some fittings manually.

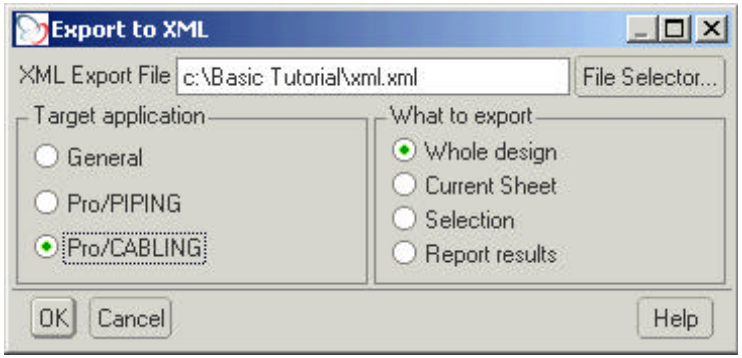
Ports

Entry_Port	This property is optional and can be added to the port of a fitting or nozzle. The value of this property must match the name of the coordinate system in the 3D component that the pipeline needs to be attached to.

Exporting to an XML file

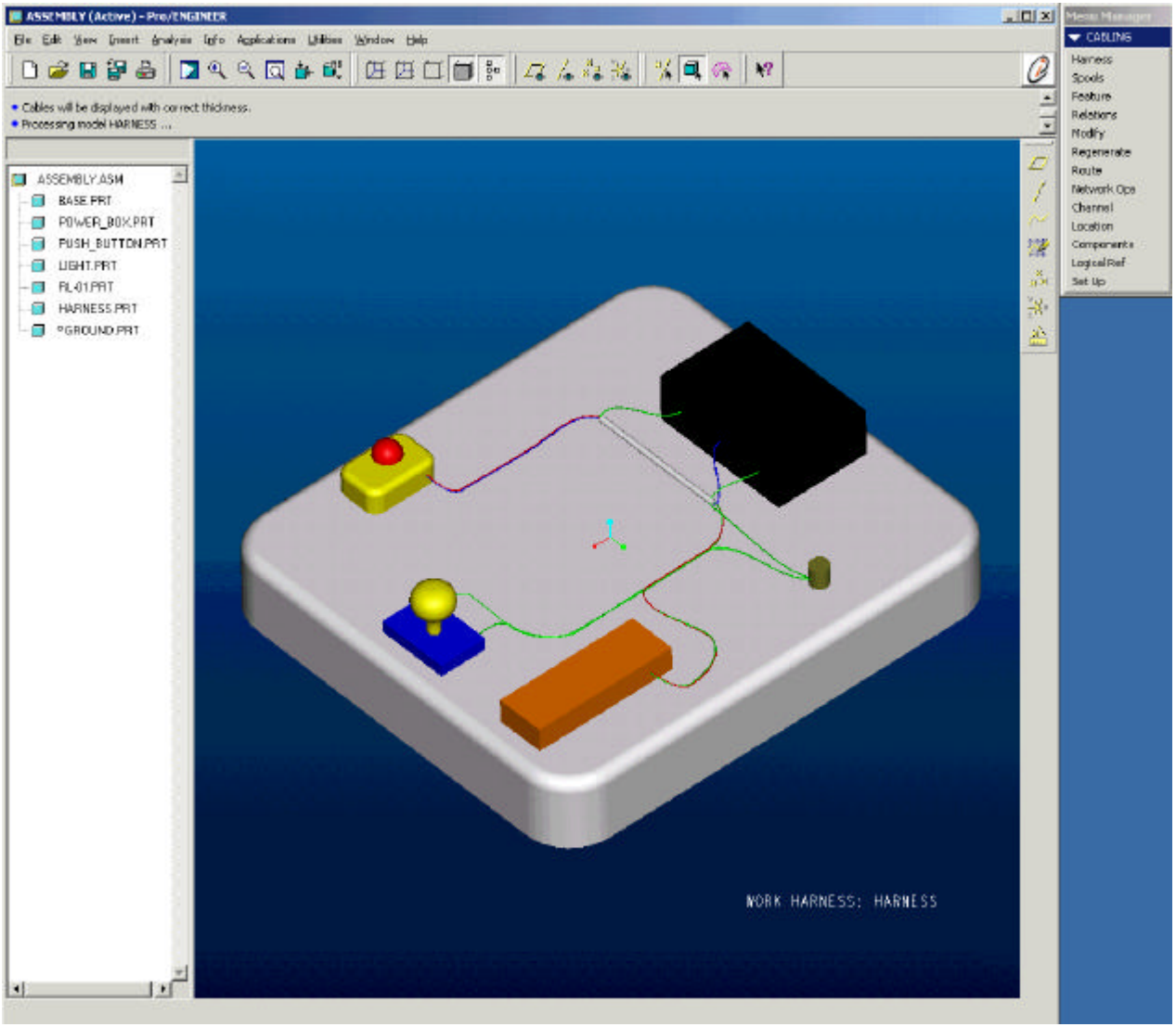
32. **Select** File > Export > XML
33. **Select** File Selector
34. **Enter** xml.xml for file name
35. Select OK
36. **Select** Pro/CABLING
37. **Select** Current Sheet
38. Select OK





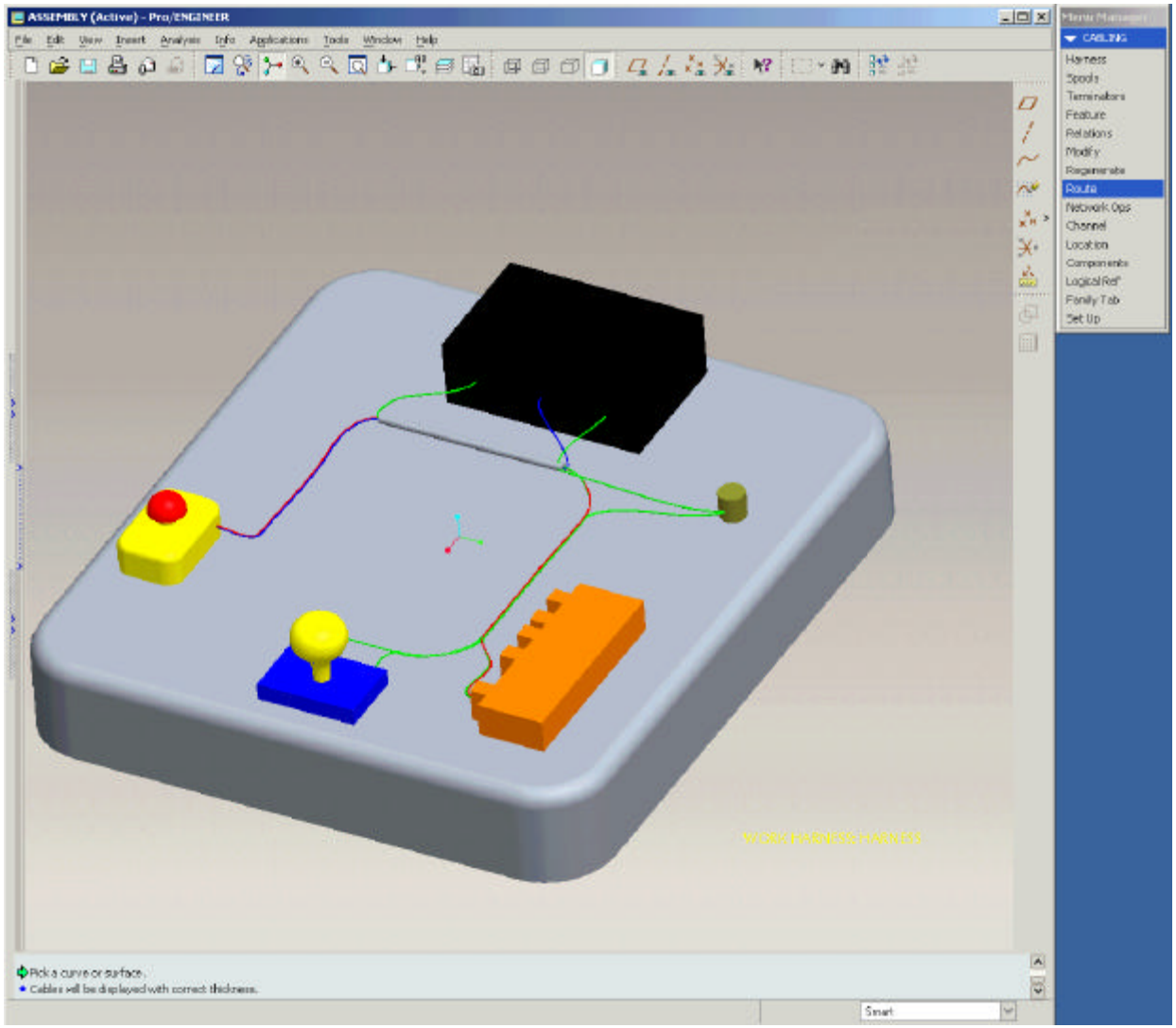
Importing XML Data into Pro/CABLING 2001

39. **Start Proe 2001** in proe 2001 models directory
40. **Open** Assembly.asm
41. **Select** Applications Cabling
42. **Select** Harness > HARNESS
43. **Select** Logical Ref > Import > PTC Neutral
44. **Select** xml.xml
45. **Select** Components > AutoDesignate > Apply >**OK**
46. **Select** Route > Create > Wire > From Logical
47. **Select** All > Done Sel > Done Sel
48. **Select** Autoroute > Accept
49. **Select** Route > Create > cable > From Logical
50. **Select** All > Done Sel > Done Sel
51. **Select** Autoroute > Accept
52. **Select** Logical Ref Compare



Importing XML Data into Pro/CABLING Wildfire

53. **Start Wildfire** in proe wildfire models directory
54. **Open** Assembly.asm
55. **Select** Applications Cabling
56. **Select** Harness > HARNESS
57. **Select** Logical Ref > Import > PTC Neutral
58. **Select** xml.xml
59. **Select** Components > AutoDesignate > Apply >**OK**
60. **Select** Route > Create > Wire > From Logical
61. **Select** All > Done Sel > Done Sel
62. **Select** Autoroute > Accept
63. **Select** Route > Create > cable > From Logical
64. **Select** All > Done Sel > **Done Sel**
65. **Select** Autoroute > Accept
66. **Select** Logical Ref Compare



Glossary of Terms

Term	Definition
Artifact	An artifact is a block, group, fiber or port that can have properties. It is graphically represented on a sheet as a shape. An artifact need not always be represented on a sheet. For example, fibers created using the auto wiring dialog box do not have shapes, and so, do not appear on any sheet. An artifact has a prototype and an instance. (See *Prototype* and *Instance*.)
Block	A block is an object that can contain ports. The ports can accept connections. (See *Port*)
Catalog	The library of prototype and shape definitions. (See *Prototype* and *Shape*)
Circuit Diagram	A diagram that consists of components, represented either by block graphics or group graphics, connected together by signal networks. It is a schematic diagram type.
Cross-reference Shapes	Visual aids that help track the relative locations of the members of a variable unconstrained group throughout the design.
CSV Files	Comma Separated Variable files. Use CSV files to specify tables of information. The CSV file contains numbers or strings (lines of text). Data elements (or fields) are separated by a comma.
Datasets	A fixed set of properties and values that can be applied to an object are termed as a dataset. These are created by the selection from a datatable.
Datatables	Datatables are tables of properties, usually derived from real-world manufactured parts, that can be applied to Routed Systems Designer components. Datatables are imported from Comma Separated Variable (CSV) files.
Design	A design consists of sheets that are organized into folders. A design also holds the catalog.
Design Sheet	Sheets that are organized (linked) into folders that comprise a design. A sheet may appear in several folders. Several folders may be linked to it.
Design Snippet	A collection of shapes from various design sheets.
Fiber	A fiber is a component such as a wire, a signal network or a pipeline, that can be routed. A fiber can have two or more ports joined by a network of segments. The network does not contain loops. A fiber is specific to a diagram type, unlike blocks and groups, which can be represented by multiple shapes with different diagram types. A fiber can exist on its own, without being connected.
Folder	A container object that contains links to sheets that exist in the design. A folder contains links to other folders and sheets in the design. When you create a folder, it is automatically linked to the currently selected folder. Folders are also used in the Catalog Explorer, where they contain blocks, groups, fibers and ports, and in the Datatable Explorer, where they contain datatables.
Groups	Groups are used to associate multiple blocks or fibers together in a single object, for example, contacts and coils in a relay. Groups can contain blocks, fibers, groups, and ports.
Healing	The automatic closing of a gap in a fiber after deletion of a shape.
Inlining	The insertion of block and group shapes into fibers.
Instance	An instance is a copy of a prototype that retains a link to the prototype from which it was created. This enables it to be updated from the original prototype. Both artifacts and shapes have instances and prototypes.
Location Set	A set of drawings that cross-reference each other.
Mapkeys	A shortcut tool used to map a sequence of keys to a tool or command. Mapkeys are generally used to activate tools in the application's main window.
Membermap	An entity, that applies to a hierarchical object, and defines the properties for each element in the hierarchy. A membermap must specify a name for every member in the group. If the member is a group, member names must be specified for the

	members of the group also. The member mappings can also specify a dataset for each member. (See *Portmap*.)
Mnemonics	A tool that allows keyboard short cuts.
Multi-level Cable	A cable that contains many cables.
P&ID	Process and Instrumentation Design, a commonly used type of Piping diagram.
Port	A port is a terminal or pin that can be connected to zero or more ports. A port always has a parent or an owner. The parent can be a block, a group, or a fiber. A port can connect to any port irrespective of the parent. For example, you can connect a port on a block to a port on another block, a port on a fiber, or a port on a group. A port from a block, a port from a fiber, and a port from a group can all be connected together in a single connection. Ports are of two types, rigid and scaleable. A rigid port does not scale when the parent is scaled. A scaleable port scales when the parent is scaled.
Portmap	Applying a dataset to an instance not only sets properties on that instance, but also (optionally) renames and sets the properties of all the ports for that instance. This is called a portmap. Portmaps can be applied to blocks or fibers. The port mappings can also specify a dataset for each port. (See *Membermap*.)
Prototype	A prototype is a definition object from which instances are created in the design. (See *Instance*.)
Scenery	Miscellaneous 2D graphics such as sheet border. These 2D graphics are not a part of the shape.
Shape	A shape is a graphical representation of an artifact on a sheet. A block shape represents a block, a group shape represents a group, a fiber shape represents a fiber, and a port shape represents a port. A shape has a prototype and an instance. (See *Prototype* and *Instance*.)
Shield	An extra core that has a different graphical representation and type than the other cores in a cable. There can be more than one shield in a cable.
Signal Network	A signal network is a fiber network with a diagram type of "circuit" that represents the connections between block and group ports. It is independent of any fibers in other diagram types, such as wiring or block.
Spare Core	When a dataset is applied to a cable, any core specified by the dataset and not existing in the cable, is created as a spare core.
Spool	Synonymous with a dataset combined with a membermap.
Template	Basic sheet format, paper size, associated graphics, and so on.