

Pro/ENGINEER[®]

Wildfire[™] 2.0

Pro/CABLING[™]

Help Topic Collection

Parametric Technology Corporation

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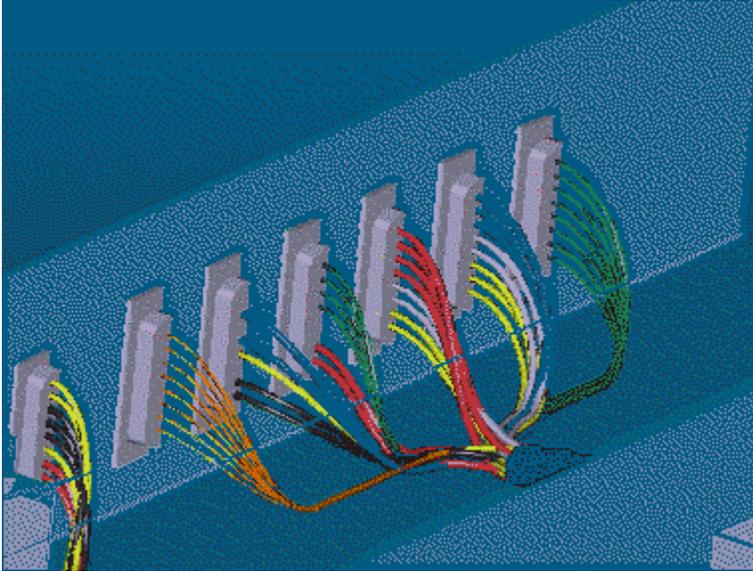
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Pro/CABLING

Using Pro/CABLING

About Pro/CABLING

Use the Pro/CABLING module to define 3D cable harnesses in Pro/ENGINEER assemblies. In Pro/CABLING, you can route cables concurrently with the design and assembly of electrical and mechanical components.



A Harness Part Within an Assembly

If you use Pro/DIAGRAM to create two-dimensional schematic representations of electrical assemblies, you can use the logical reference capability in Pro/CABLING and Pro/DIAGRAM to compare logical connections and parameters in both modules.

To enter the Cabling mode, click **Applications > Cabling** in the Pro/ENGINEER main menu.

Using the Cabling Workflow

This workflow is a sample of the basic steps used to create a cabling assembly. In the assembly design, use **Applications > Cabling**. You can perform the following operations in Pro/CABLING:

- **Create a Harness Part**

Use **CABLING > Harness > Create** to create a new harness part. The harness part is a .prt file and is a part of the assembly, but cannot be opened as a separate part. A harness part cannot be opened in the Part mode. For manufacturing output, the part is extracted from the assembly and represented as flattened.

The new harness is the active harness or work harness as shown in the lower right corner of the graphics window. You can create more than one harness in an assembly. Only one harness is active at a time. You can only select and edit cable entities of the active harness in the cabling assembly. Any item you create is saved to the active harness.

- **Read in a Logical Reference (Optional)**

Use **CABLING > Logical Ref** to read in a file if you are using a logical reference from Pro/DIAGRAM or another formatted wire list. A logical reference can pass spools, wires, and cables with preset parameters and values from the diagram to the cabling database.

- **Add Spools to the Database**

Use **CABLING > Spools > Create** to either add new spool definitions to the database or read them in from your logical reference. Use **Create > From Logical** to bring in selected spools from the logical reference that you have specified.

- **Designate Components**

Use **CABLING > Components > Designate** to designate 3D parts in the assembly as connectors. This procedure adds parameters to the definition of the components such that the components become the start and end point of the cables. If a diagram connector is referenced to a part in the assembly with a `model_name` parameter, the referenced model can be automatically designated as a connector.

- **Create Wires and Cables**

Use **CABLING > Feature > Create > Wire** or **Cable** to add wires and cables to the harness. These wires and cables are added to the database and not physically added to the design. If you are importing a logical reference, you can use the **From Logical** command to import cables, else you are prompted to type a new wire or cable name. If you are importing a cable, the conductor properties of the cable are also imported, if they are not then you must define them.

- **Route or Autoroute Cables between Locations**

If you are routing manually, you can begin adding a routed cable to the harness as soon as you have created it in the database and defined its conductors. Use the **CABLING > Route** command to define a point-by-point path of locations that the cable follows as it is added. Locations can be fixed, offset, or dependent on other locations, to capture the design intent of the cable in case of changes in the assembly. Offsetting a part of the network, offsets all the locations on the network.

Alternatively, you can use autorouting for larger wire lists. To autoroute, use the **CABLING > Network Ops** command to add a network of locations between the components that will be connected by the harness. When the network is complete, you can autoroute the entire wire list.

Configuring Pro/CABLING

About Configuring Pro/CABLING

You can set environment options by specifying `config.pro` configuration file options and their values in the **Options** dialog box (**Tools > Options**). For example, the `align_cable_bundles` option allows you to align newly-created bundles where they meet or branch out. Setting the `auto_xml_on_save` option automatically creates an XML Logical Reference when saving a cabling assembly.

In the **Options** dialog box, in **Current Session**, under the **Electromechanical** category, a list of configuration options is available in alphabetical order for CABLING, DIAGRAM, and HARNESS. Each option contains the following information:

- Configuration option name
- Default and available variables or values. All default values are in italics.
- Brief description and notes describing the configuration option

Note: After you set the configuration options, all settings take effect immediately in the current Pro/ENGINEER session.

To Set Pro/CABLING Configuration Options

1. Click **Tools > Options**. The **Options** dialog box opens.
2. Select **By Category** in the **Sort** box.
3. Select **Current Session** in the **Showing** box.
4. Clear the **Show only options loaded from file** box to see all configuration options or to see configuration options that are available for the current session.
5. In **Current Session**, select the **Electromechanical** category. A list of configuration options arranged in alphabetical order for CABLING, DIAGRAM, and HARNESS appears.
6. Select a Cabling-specific configuration option from the list or type the valid configuration option name in the **Option** box.
7. When you select a configuration option from the list, its corresponding value appears in the **Value** box. Modify this value.

or

Type a new value to be assigned to the configuration option in the **Value** box.

Note: The default value is followed by an asterisk (*).

8. Click **Add/Change**. The configuration option and its value appear in the list. The status of the configuration option changes to .

Note: The **Add/Change** option is enabled only when you change the configuration option name or the value of an existing configuration option or type a value for a new configuration option.

9. When you finish configuring Pro/CABLING, click **Apply** or **OK**.

Note: It is recommended that you set the Pro/CABLING configuration options before starting or opening a new cabling assembly.

align_cable_bundles

yes, no

Aligns or unaligns newly created bundles at a point where they meet or branch out.

By default, the bundles are aligned. If not, ensure that this option is set to *yes* before creating a network.

Note: Only the new bundles that are created after setting this configuration option will be aligned.

autoroute_path_param_name

USE_PATH

Sets the wire parameter to be used while selecting a path during autorouting.

auto_xml_on_retrieve

no, yes

Automatically loads Cabling Logical Reference from a XML file when retrieving the cabling assembly.

auto_xml_on_save

no, yes

Automatically creates an XML Logical Reference when saving a cabling assembly.

cable_int_portions_for_clr

no, yes

no—Global clearance check for internal cable portions is excluded.

yes—Global clearance check for internal cable portions is included.

display_internal_cable_portion

no, yes

If *yes*, internal cable portions are displayed for wires that have been manually set to *yes* in the **Cable Paths Setup** dialog box for each individual splice and custom component. This option can be overridden in the **Environment** dialog box.

display_thick_cables*no, yes*

Sets the default startup mode.

yes—Displays thick cables and wires.

no—Displays centerlines of wires and cables only.

You can override this option in the Cabling mode using the **Environment** dialog box, **Model Display** dialog box, or by selecting **Thick Cables** from the menu bar.

full_hlr_for_cables*full, partial, none*

If set to *partial* or *none*, some lines that should be hidden behind cables are visible in the wireframe mode. Using the **Model Display** dialog box, under the **Edge/Line** tab, you can set the **Cable HLR** options for the current session.

full—Removes hidden lines from view when cables hide other geometry. Use when the **Display Style** is **Hidden Line** in the **Environment** dialog box.

partial—Cables hide other non-interfering cables except when cables route together between same locations.

none—Display hidden lines (faster) that should be hidden behind cables or wires.

Note: Effects of using this configuration option are visible only in the hidden-line display mode. Helps speed up the display of hidden lines, but at the loss of some quality.

harn_tang_line_display*yes, no*

Display the tangent lines between the segments of a cable in the Thick Cable Display mode.

multipoint_location_count

1

Specifies the maximum number of cable locations to allow in one feature. To create locations as features, set this configuration option to 1. To limit the number of locations in one feature, set to a number between 2 and 100. The suggested number of locations in one feature must range from 10 to 20. If you set the number of locations to be greater than 1, then up to the specified number of locations are created in one feature. Features with multiple locations are automatically created during routing.

Note: If a single feature has several locations, you can only suppress or reorder them all together.

pro_spool_dir

<home directory>

Sets the default directory from which the spools are retrieved by default. Use the full path name, for example: `/home/users/spools`.

The current working directory is the default directory.

pro_cbltrm_dir

<home directory>

Sets the default directory from which terminators are retrieved by default. Use the full path name, for example: `/home/users/terminators`.

The default is the current working directory.

update_pre_16_cable_layers

no, yes

Updates the pre-release 16.0 harnesses.

If *yes*, automatically updates the harnesses containing cables routed prior to Release 16.0 when retrieved to comply with new display standards.

fan_with_network

yes, no

If *yes*, the network is visible in flat harness.

Cable-specific Display Setups

About Setting Up the Pro/CABLING Display Environment

You can set up the Pro/CABLING display environment in the following ways:

- Using the configuration options.
- Using the **Environment** dialog box.

Use the following options from the **Environment** dialog box to change the Pro/CABLING display settings:

- **Thick Cables** or **Centerline Cables**—Shows cables with true width or as centerlines. Centerline shows the cable's centerline with defining location points. Mass property calculations are performed using the correct thickness regardless of the display state.
- **Internal Cable Portions**—Determines if all cable portions inside the splice and custom components are displayed.
- **Reference Designators**—Determines if the reference designator for the component is displayed.

Note: You can also access these options directly from the menu bar.

- **Use Fast HLR** (Hidden Line Removal)—Displays hidden lines while spinning and reduces time to compute HLR.

Note: The display style must be set to **Hidden Line** in the **Environment** dialog box.

To Display Cables as Thick or Centerline

1. Click **Tools** > **Environment**. The **Environment** dialog box opens.
2. Select either **Thick Cables** or **Centerline Cables**.
3. Click **Apply** and then **OK** to close the dialog box.

Note: You can also access the **Thick Cables**, **Centerline Cables**, and **Reference Designator** options directly from the menu bar in the Cabling mode.

or

1. Click **View** > **Display Settings** > **Model Display**. The **Model Display** dialog box opens.
2. Click the **Edge/Line** tab.
3. Under **Cable Display**, select either **Thick Cables** or **Centerline Cables**.
4. Click **Apply** and then **OK** to close the dialog box.

Note: To manually route cables, select **Centerline Cables**.

Displaying Hidden Lines

Hidden lines are drawn lines that appear behind other objects in a 3D view. When you set the display style to **Hidden Line**, the hidden lines appear behind solid objects, but are displayed in grey. When you set the display style to **No Hidden**, lines behind the solid objects are removed from the display.

Cable HLR is available only if the display style is set to **Hidden Line** in the **Environment** dialog box.

The cable is hidden by geometry and can also hide other geometry, depending on the setting for the `FULL_HLR_FOR_CABLES` configuration option.

Note: Hidden line display does not always work as expected. In cases where cables physically intersect a part or another cable, the entire cable may be incorrectly displayed. This condition indicates cable routing interference.

To reduce cable interference, additional locations can be added. If you use routing commands, such as **Use Dir** or **Along Axis**, greater control is provided over cable tangency at selected locations and the possibility of interference between cables is reduced.

Other Cabling Hidden Line Display Options

Use the **View > Display Settings > Model Display** command to display the hidden lines of a cable. The following options are available under the **Edge/Line** tab in the **Cable HLR** (Hidden Line Removal) box to display the hidden lines of a cable.

- **Full**—Cables are hidden by other objects and hide other objects and wires within harnesses.
- **None**—Hidden lines are displayed and the overall display process is faster. Cables are hidden by other objects, but do not hide other objects or hidden cables within the harness. This is the most recommended method.
- **Partial**—Similar to **Full**, but does not perform calculations within the harness.

About Cable Colors

Cable or wire spools contain parameters such as name, type, color, gauge, and so on for a single insulated conductor. Cable spools contain parameters for a specific number of conductors and property descriptions for each conductor in the cable.

Using the cable's spool file, you can set a default color for a cable. The value of the **COLOR** parameter for a cable or wire specified in the spool file appears in the **Color and Appearance** dialog box. Cables can be created using the spool parameters specified in the cable's spool file. You can retrieve the cable color and appearance data from the `.dmt` or `.map` files.

Note: The `.dmt` and `.map` files must exist in the directory from where you start Pro/ENGINEER. Ensure that the color that you assign to the cable exists in the `color.map` file.

If the **COLOR** parameter for the spool is not defined or if you want to override the default value of this parameter, use the **Color and Appearance** dialog box to update the color of a selected cable. If you edit the color of a cable using the **Color and Appearance** dialog box, all cable segments created from its spool are also updated with the new color. Cables created from the spool after editing the **COLOR** parameter use the new color. However, the new color is local to the design file and the value of the **COLOR** parameter is not updated in the spool file. By default, the system color "Letter" is applied to the wires or cables that have unknown color assigned to it.

Save the new colors that you have added to the appearance palette in the **Color and Appearance** dialog box as a `.dmt` file.

To Switch Model Color Display

1. Click **View > Display Settings > Model Display**. The **Model Display** dialog box opens.
2. Click the **General** tab.
3. Under **Display**, select **Colors**. The model is displayed with colors.
4. Click **Apply** and then **OK** to close the dialog box.

or

1. Click **Tools > Environment**. The **Environment** dialog box opens.
2. Under **Display**, select **Colors**. The model is displayed with colors
3. Click **Apply** and then **OK** to close the dialog box.

To Display Cable Portions Inside Components

1. Open a harness part in the **Assembly** or **Harness-Mfg** mode.
2. Click **View > Display Settings > Model Display**. The **Model Display** dialog box opens.
3. Click the **General** tab.
4. Under **Display**, select **Internal cable portions**. The internal cable portions routed through splices and components are globally displayed for wires that have **Show** set to **Yes** in the **Cable Paths Setup** dialog box
5. Click **Apply** and then **OK** to close the dialog box.

or

1. Click **Tools > Environment**. The **Environment** dialog box opens.
2. Under **Display**, select **Internal Cable Portions**.
3. Click **Apply** and then **OK** to close the dialog box.

Cabling Drawing Setup Options

Command	Description
ref_des_display	Sets whether the reference designator is to be displayed or not: When set to <i>yes</i> , the reference designator is displayed on connectors in a cabling or flat harness assembly. When set to <i>no</i> , the reference designator is not displayed on the connectors in the assembly.
show_cbl_term_in_region	Allows use of the report symbols <code>&asm.mbr.name</code> and <code>&asm.mbr.type</code> to show terminators in Pro/REPORT tables for cable assemblies having connectors with terminator parameters. Shows terminators if set to <i>yes</i> and if the Cable Info attribute is set for the repeat region. When creating new drawings, the default value is <i>yes</i> . For existing drawings, the default value is <i>no</i> .

Logical Referencing to Pro/DIAGRAM

About Logical Referencing

Use the **CABLING > Logical Ref** command to reference Pro/DIAGRAM information directly in Pro/CABLING. This information includes the reference designator, pin-to-pin connection information, and parameter values of connectors, pins, spools, wires, and cables. You can reference the whole diagram or only items on selected layers. You can also autoroute wires and cables, and designate connectors.

You can also import the same information from an ASCII text file, either in the PTC neutral wire list format (.nwf), the Mentor Graphics wire list format, or the Routed Systems Designer format (.xml). You can use the information to automatically designate connectors and autoroute complete cables. At any point in the design process, you can compare logical information with the original wire list or diagram reference for accuracy. If you change the diagram later, you must run the **Logical Ref** command again to update the cabling assembly accordingly.

You must have unique reference designators across the Pro/CABLING assembly. This is because you perform a logical referencing operation on the entire assembly and not specifically with respect to any harness part.

If you export a logical reference, you can write physical data of the wires, connectors, and components in your assembly to a file, for analysis purposes in other systems.

On comparing the current logical information with the referenced data, the output can be displayed on the screen or written to a file.

You can update the parameter data from the referenced data. All specified spools, connectors, and wires in the assembly are updated. For example, if you change the characteristics of a spool in your referenced data, you can automatically update the information in Pro/CABLING using the **Update** command and selecting the spool from the list of mismatched objects.

To Reference a Diagram

1. Click **CABLING > Logical Ref > Ref Diagram**. The **Open** dialog box opens. The **REF DIAGRAM** menu also appears.
2. Choose one of the following **REF DIAGRAM** commands:
 - **Full Diagram**
 - **Chosen Layers**

By default, you can reference the full diagram.

3. Select a diagram from the **Open** dialog box and click **OK** to reference the full diagram.
4. If you want to logically refer to specific layers in a Pro/DIAGRAM diagram, click **Chosen Layers** and select a diagram from the **Open** dialog box. You must

provide the name of the diagram to be referenced. If no layers are present in the diagram, Pro/CABLING displays an appropriate message. Else, Pro/CABLING fetches a complete list of layers that are available in the referenced diagram and displays them in the **Layer Sel** menu.

5. Select one or more layers to be referenced from the list of layers available in the referenced diagram or select **Select All** to select all the available layers.
6. Click **Done Sel**.

Updating Logical References

Use the **Update** command in the **LOGICAL REF** menu to update not only the cable or wire parameters, but also the spool, component, and connector parameters for any linked logical references.

Note: Prior to using the **Update** command, ensure that you again logically refer the diagram in order to get to the latest changes.

About Comparing Data Output

When you compare model data and reference data, a comparison file is generated. This comparison file lists all the items in both the parameter data and the reference data. One line is generated for two corresponding items. Additional lines are added if the parameter data and the reference data do not match.

The object of the comparison such as a wire list or diagram is identified as the REFERENCE DATA while the active Pro/CABLING 3D assembly is identified as the DESIGN DATA in the **INFORMATION WINDOW <assembly_name.cmp>** window.

When you select the **Compare** command on the **LOGICAL REF** menu to compare the design data and the reference data, Pro/ENGINEER compares the following:

- **Spools**—A match occurs if the design data and the reference data have the same parameters with the same values. If the parameters common to both the reference and the design data have the same values, the comparison file indicates a subset match, denoted by the word *subset* in parentheses. The spools are identified by the spool names.
- **Connectors and Components**—Connectors and components match when they have the same parameters with the same values. Connectors and components are identified by their reference designators.
- **Wires and Cables**—The logical end connections of a wire or cable are matched and the cable and wire parameters are compared. A wire is said to be matched if it runs between the same connectors and pins, has the same name, and has the same parameters with the same values. Wires are identified by the names.
- **Cable Conductors**—If you specify cable conductors separately in the reference information, their From/To and parameter information is compared.

You can use the **Matched** and **Alphabetical** commands from the **LOGICAL REF** menu to display the details of the missing, unmatched, or matched items, in alphabetical order.

The Neutral Format Wire List

NEW CONNECTOR T2

PARAMETER MODEL_NAME CLEANCONN2

PARAMETER NUM_OF_PINS 4

PARAMETER GENDER MALE

PIN 1

PARAMETER ENTRY_PORT sys1

PARAMETER GROUPING ROUND

PARAMETER INTERNAL_LEN 0

PIN 2

PARAMETER ENTRY_PORT sys2

PARAMETER GROUPING ROUND

PARAMETER INTERNAL_LEN 0

PIN 3

PARAMETER ENTRY_PORT sys3

PARAMETER GROUPING ROUND

PARAMETER INTERNAL_LEN 0

PIN 4

PARAMETER ENTRY_PORT sys4

PARAMETER GROUPING ROUND

PARAMETER INTERNAL_LEN 0

! Rails

! Wires and cables

NEW CABLE CABLE0002 F_SPOOL

ATTACH T1 "" T2 ""

CONDUCTOR 4

ATTACH T1 4 T2 4

PARAMETER NAME WIRE4

CONDUCTOR 3

ATTACH T1 3 T2 3

PARAMETER NAME WIRE3

```

CONDUCTOR 2
ATTACH T1 2 T2 2
PARAMETER NAME WIRE2
CONDUCTOR 1
ATTACH T1 1 T2 1
PARAMETER NAME WIRE1
! comment
NEW WIRE_SPOOL <wire spool name>
PARAMETER <param> <value>
PARAMETERS <param1> <param2> <param3>
VALUES<param1 val> <param2 val> <param3 val>
NEW CABLE_SPOOL <cable spool name> <number of conductors>
PARAMETER <param> <value>
PARAMETERS <param1 name> <param2 name>
VALUES <param1 val> <param2 val>
CONDUCTOR <id> <optional cond name>
PARAMETER <param> <value>
PARAMETERS <param1 name> <param2 name> <param3 name>
VALUES <param1 val> <param2 val> <param3 val>
CONDUCTOR <id> <optional cond name>
VALUES <param1 val> <param2 val> <param3 val>
CONDUCTOR <id>
NEW WIRE <wire name> <wire spool name>
ATTACH <from conn or comp name> <from pin name> <to conn or comp name>
<to pin name>
PARAMETER <name> <value>
PARAMETERS <param1> <param2> <param3>
VALUES<param1 val> <param2 val> <param3 val>
NEW WIRE <wire name> <wire spool name>
ATTACH <from rail name> "" <to conn or comp name> <to pin name>
NEW CABLE <cable name> <cable spool name>

```

Cabling - Help Topic Collection

PARAMETER <name> <value>

PARAMETERS <param1 name> <param2 name>

VALUES <param1 val> <param2 val>

CONDUCTOR <id>

PARAMETER <param name> <value>

PARAMETERS <param1 name> <param2 name> <param3 name>

VALUES <param1 val> <param2 val> <param3 val>

CONDUCTOR <id>

PARAMETER <param name> < value>

ATTACH <from conn or comp name> <from pin name> <to conn or comp name>
<to pin name>

VALUES <param1 val> <param2 val> <param3 val>

NEW CONNECTOR <name>

PARAMETER <param> <value>

PARAMETERS <param1 name> <param2 name>

VALUES <param1 val> <param2 val>

PIN <pin name>

PARAMETERS <param1 name> <param2 name>

VALUES <param1 val> <param2 val>

PIN <pin name>

PARAMETER <name> <value>

PARAMETERS <param1 name> <param2 name>

VALUES <param1 val> <param2 val>

PIN <pin name>

PIN <pin name>

VALUES <param1 val> <param2 val>

NEW COMPONENT <name>

PARAMETER <name> <value>

PARAMETERS <param1 name> <param2 name>

VALUES <param1 val> <param2 val>

PIN <pin name>

PARAMETERS <param1 name> <param2 name>

VALUES <param1 val> <param2 val>

PIN <pin name>

PARAMETERS <param1 name> <param2 name>

PARAMETER <name> <value>

VALUES <param1 val> <param2 val>

PIN <pin name>

VALUES <param1 val> <param2 val>

NEW RAIL <name>

PARAMETER <param name> <param value>

To Export a Wire List

1. Click **CABLING > Logical Ref > Export**. The **WIRELIST EXP** menu appears.
2. Click one of the following options to write logical information of the wires, connectors, and components in your assembly to a Routed System Designer XML wire list file, PTC Neutral wire list file, or the Mentor Graphics Format wire list file, respectively, for analysis purposes in other systems:
 - **RS Designer**
 - **PTC Neutral**
 - **Mentor Graph**
3. In case of **Mentor Graph** file output, the output wire list file is written to the current directory. To write this file to another directory, at the prompt, type the full path including the file name and click .

Note: Pro/CABLING can output a wire list as a back annotation file for export to the Mentor Graphics application.

To Import a Wire List

1. Click **CABLING > Logical Ref > Import**. The **WIRELIST IMP** menu appears.
2. Click one of the following options to import logical information of the wires, connectors, and components in your assembly from a Routed System Designer XML wire list file, PTC Neutral wire list file, or the Mentor Graphics Format wire list file, respectively:
 - **RS Designer**

- **PTC Neutral**
 - **Mentor Graph**
3. If you select **PTC Neutral**, you can either select a Routed Systems Designer created `.xml` file or a Neutral Wire List `.nwf` file using the **Open** dialog box. If you select **Mentor Graph**, you can select any Mentor Graphics file.
 4. Click **OK**.

Creating the Harness Part

About Creating Harnesses

Before routing wires or cables in an assembly, you must create a harness. A harness is a part file that contains cables and wires within the assembly. When you create a harness, it is named and saved as a `.prt` file, but it can only be accessed within a Pro/CABLING assembly.

However, a harness part can only be accessed in Assembly mode and cannot be opened in Part mode. You can output a harness to a drawing view: by selecting a harness part and setting it as the active model using Set Model or by selecting a flat harness.

You can create several harnesses in the same assembly. In assemblies with multiple harnesses, you can work on only one harness part at a time. The name of the active harness is displayed below the assembly name in the graphics window, with the label `WORK HARNESS`. While this harness is active, every feature you add to the cabling is added to the active harness part. You can select cabling entities that have been added to other harnesses only when the respective harness is active.

To organize a large design, you can divide the harness into subharnesses. A subharness is a subset of a harness part and not a separate file. Subharnesses can use all the entities available to a harness, but you can also create them separately in Harness Manufacturing mode.

Use the **CABLING > Set Up** command to create and edit harness parameters, names, notes, and reference dimensions.

To Create a Harness

1. Click **CABLING > Harness > Create**.
2. At the prompt, type a new harness name and press ENTER. The harness part is created and added to the model tree. The newly created harness becomes the active harness.

To Modify a Harness

1. Ensure that a harness is active.

2. Click **CABLING > Set Up**. The **HARN SETUP** menu appears.
3. Edit units of measure, harness names, report definitions, notes, parameters, terminator tables, or strip tables. Changes affect only the active harness part.
4. Click **Done**.

Using the Harn Setup Menu

You can specify the following options to modify a harness:

- **Units**—Determines the length of the harness in units.
- **Name**—Sets up a name for the harness. The **NAME SETUP** menu appears:
 - **Component**
 - **Feature**
 - **Detail Item**
 - **Other**
- **Report Defs**—Sets default values for cable type locations as From/To in report tables.
 - **Cable Jacket**
 - **Cable Shield**
 - **Cable Pin**
 - **Show Current**
- **Notes**—Modifies, adds, or removes notes from a cabling assembly.
- **Parameters**—Sets up parameters for the harness.
- **Term Table**—Accesses terminator table operations.
- **Strip Table**—Accesses strip table operations.

To Delete a Harness

1. Select the harness part to be deleted from the assembly or from the model tree.
2. Click **Edit > Delete**. The harness part is removed from the assembly and all cabling data in the harness part is lost.

Note: Use the **Delete** command only when you want to completely remove a harness and start over or if you have created a harness part and never used it. If you do not delete a harness part, it appears as a part in the Bill of Materials.

To Set the Working Harness

1. Click **Cabling > Harness**. The **HARNESSES** menu appears.
2. Select the harness name you want to set as the working or active harness.

To Copy a Harness

1. Click **CABLING > Harness > Copy Harness**. The **Open** dialog box opens.
2. Select the harness that you want to copy and click **Open**. The **Open Rep** dialog box opens.
3. Select the required representation and click **OK**. The selected harness with its corresponding assembly is displayed in a separate window.

The **Copy Harness** dialog box opens and the **Select** menu appears.

4. Select a connector in an assembly to which you want to copy the harness corresponding to the reference designator of the component or connector in the assembly from which the harness is being copied.

A new reference designator is assigned to the reference model.

5. Once you have selected all connectors, specify a new harness name in the **Copy Harness** dialog box and click **OK**.

The harness is copied to the assembly.

To Create Datum Features for Harness Parts

1. Click **Insert > Model Datum**.
2. All datum features support harness design. Click an appropriate command from the following to specify the datum types.
 - **Plane**
 - **Axis**
 - **Point**
 - **Coordinate System**
 - **Default Coordinate System**
 - **Sketched Curve**
 - **Curve**

Refer to the Part Modeling module for details on how to create different datum features.

To Create Subharnesses

1. Click **CABLING > Harness > Create**. You are prompted to name a harness.

2. Do not type a name. Click  to cancel. The **SUBHARNES** menu appears.
3. Click **Setup > Add**.
4. At the prompt, type a new name for the subharness.

To Set the Default Subharness

1. Click **CABLING > Harness > Create**. You are prompted to name a harness name.
2. Do not type a name. Click  to cancel. The **SUBHARNES** menu appears.
3. Click **Set Default**.
4. Select the name of the subharness that you want to set as the default subharness.

Defining Report Object Names

Use the **CABLING > Set Up > Report Defs** command to assign names to the following objects for use in Pro/REPORT:

- **Cable Jacket**—Name for unshielded cable type locations
- **Cable Shield**—Name for shielded cable type locations
- **Cable Pin**—Pin name for cable type locations

Clicking **Show Current** displays the currently assigned objects.

Adding Harnesses to a Family Table

About Adding Harnesses to a Family Table

When you add a family table to a part, the part is first opened in the Part mode before the creation of part instances. If you choose the parent assembly as the reference model, it is added as a reference to the base part object in the part family table.

The assembly instance in the assembly family table is modified to include the part instance. For more information on family tables, refer to Managing Model Composition under Pro/ENGINEER Fundamentals.

To Add a Harness Part to a Family Table

1. After you set the working harness, click **CABLING > Family Tab**. The **Family Table : <harness_name>** dialog box opens.

2. Click  to add or delete table columns. The **Family Items, Generic :<harness name>** dialog box opens. You can add dimensions, parameters, features, components, groups, a feature from a table driven pattern, a merged part, or a referenced model to the family table.
3. Under **Add Item**, you can select **Ref Model**. The **REFERENCES** menu appears with a list of available reference models.
4. Select a reference assembly from the list and click **DONE**.
5. Click **OK** in the **Family Items, Generic :<harness name>** dialog box.
6. If you do not want to add more reference models to the family table, click **OK** in the **Family Table <harness_name>** dialog box.

To Use Family Tables with Harnesses

1. Click **Applications > Standard**.
2. Click **Tools > Family Tab**. The **Family Table <assembly_name>** dialog box opens.
3. Add a family table to the parent assembly.
4. Click  to create an instance of a part by opening it in the Part mode.
5. When you have completed adding the required number of instances, click **OK**. You can modify the assembly instance in the assembly family table to include the harness part instance.

Creating Spools

About Spools

A spool has a unique set of parameters and values that are passed to the wire or cable when a wire or cable is created. Each wire and cable in the design must be created from a predefined spool. The spool, when defined, is saved within the assembly and can be written to a text file with the .spl extension. You can reference this file in a new diagram or cabling designs.

The types of spools are:

- **Wire spools**—You must set wire spool parameters such as `NAME` and `TYPE`. The name of the wire spool must be unique. The `TYPE` parameter is set to `WIRE` to differentiate it from a cable spool. Other common but optional parameters defined in the wire spool are `COLOR` and `WIRE_GAUGE`. The parameter, `MIN_BEND_RADIUS` is automatically generated. This parameter provides values for calculations in Pro/HARNESS.
- **Cable spools**—Cable spools have parameters that are similar to those required for wire spools. You must set the `NUM_CONDUCTORS` parameter to define the

number of insulated conductors in the cables that are created from the spool. Assign an appropriate integer value to the `NUM_CONDUCTORS` parameter.

- **Sheath spools**—Sheath spools are created for bundles. You must set the sheath spool parameters such as `SHEATH_TYPE` and `PRESHRINK_INNER_DIAMETER`. Other common parameters are `NAME`, `TYPE`, and `UNITS`. The parameters, `MIN_BEND_RADIUS` and `WALL_THICKNESS` are automatically generated.

Note:

- Setting the `pro_spool_dir` configuration option allows you to specify a directory from which spools can be read. If you do not specify a directory, the current working directory is the default.
- You can retrieve the spool parameters using the Pro/REPORT table and also use these parameters to display the wire or cable labels.
- If you create spool files in a text editor outside of Pro/ENGINEER, the name of the spool file must be identical to the spool name in Pro/ENGINEER. You can also save a spool file in the ASCII format to be retrieved for use in other diagrams or a Pro/CABLING assembly.

Cable Spool Parameters

The following is a list of predefined parameters that are unique to cable spools. A cable has a defined number of conductors, and each conductor has its own defining parameters, for example gauge or color, within the spool file. All wire spool parameters are valid for cable spools. Required parameters are shown in bold.

<code>NAME</code>	The name of the spool file. Format: <code>NAME text_string</code>
<code>TYPE</code> (read only)	Determines a cable spool or wire spool. Cables are of type <code>PREFAB</code> .
<code>NUM_CONDUCTORS</code>	The number of conductors present in a cable. The default value is zero.
<code>DENSITY</code>	The linear density of the spool (in mass/unit length). Not used in Pro/DIAGRAM, but used in Pro/CABLING when referencing a diagram to determine Mass Properties.
<code>INSUL_TYPE</code>	Insulation type. (Text string) <code>NONE</code> is the default.
<code>SHIELD_TYPE</code> (cable spools only)	The shield type for a cable. (Text string) If you set this parameter, it alters the cable symbol and makes it a dashed line to specify shielding.

CABLE_JACKET_REPORT_NAME	Use the default value <code>DEFAULT</code> , in which case the name of the cable shows in the report table. Any other value is interpreted as plain text. For example, <code>&cable_name</code> has no special meaning.
CABLE_SHIELD_REPORT_NAME	Use this name for the cable symbol if the cable is shielded. The default value is <code>SHIELD</code> .
CABLE_NODE_REPORT_NAME	Use this name for the nodes of a cable symbol. The default value is <code>"_"</code> .
SHIELD_LINEAR_RESISTANCE	Specifies linear resistance of the wire's shielding.
OUTER_SHIELD_LINEAR_RESISTANCE	Specifies linear resistance of the wire's outer shielding.
LIN_CAP_TO_ITEM	Specifies linear electric capacity between items.
LIN_CAP_ITEM_TO_SHIELD	Specifies linear electric capacity between items and shield.
LIN_CAP_ASSEM_ITEM_TO_SHIELD	Specifies linear electric capacity between assembly items and shield.
OUTER_SHIELD_THICKNESS	Specifies thickness of the outer shielding of the wire.

To Create a Spool

1. Click **CABLING > Spools > Create**. The **CREATE SPOOL** menu appears.
2. Select one of the following filters to specify the type of spool that you want to create.
 - **Wire**—Creates a wire spool.
 - **Cable**—Creates a pre-fabricated cable spool.
 - **Sheath**—Creates a sheath spool. Sheath spools are referenced by bundles.
 - **From Logical**—Creates a spool using a logical reference. The **REF SPOOL** menu appears.

Select the required spools from the list and click **Done Sel** to copy spool parameters from the referenced diagram. You can select one or more wire

or cable spool names from the list or click **Select All** to select all the available spools.

Note: If the logical data is imported from a wire list, the **From Logical** command is available for wire creation only. This is because the imported wire list format does not support cables.

If you use **CABLING > Spools > Create > From Logical**, Pro/CABLING automatically creates all spools associated with the created cables and wires. The information that is used to create these spools is taken from the logically referenced data from which the wires and cables were created.

If the cabling assembly does not have a reference to a diagram or a wire list, select a spool from which the cable or wire draws its characteristics.

3. To create a new spool for a wire, cable, or sheath, at the prompt, type a new spool name and press ENTER. The **Electrical Parameters** dialog box opens.
4. Under **Display For**, select **Connections** to add parameters to wires or cables or **Conductors** to add parameters to conductors of a cable spool.
5. If you have selected **Conductors**, select a parameter in the `NUM_CONDUCTORS` column, specify a value for the number of conductors for a cable spool in the **Value** box, and click **Apply**.
6. Click **View > Columns**. The **Model Tree Columns** dialog box opens.
7. Add the required parameters to the spool, click **Apply**, and then **OK**.
8. When finished, click **File > Save** or **File > Save As** to save the file to disk.
9. Click **OK** to apply the spool parameters.

To Modify Spools

1. Click **CABLING > Spools > Edit**. The **Select Spools** dialog box opens.
2. Select one of the following filter to specify the type of spool to be modified:
 - **All Spools**
 - **Wire Spools**
 - **Cable Spools**
 - **Cable Spools by Num Cond**
3. Click a spool name from the spool list. You can filter the spools by spool type or number of conductors.

or

Click  or  to select or remove all spools from the list, respectively.

4. Click **OK**. The **Electrical Parameters** dialog box opens.

5. Modify the required spool parameters, click **Apply**, and then **OK**.
6. Click **Done/Return** in the **SPOOLS** menu.

Note:

- Use **CABLING > Spools > Read** to read in spool parameter data from a previously created spool that was saved to disk. This file must have a `.spl` extension.
- Use **CABLING > Spools > Write** to write the selected spool parameter data to your current directory. These files have a `.spl` extension.

To Rename Spools

1. Click **CABLING > Spools > Rename**. The **Select Spools** dialog box opens.
2. Select one of the following filters to specify the type of spool to be modified:
 - **All Spools**
 - **Wire Spools**
 - **Cable Spools**
 - **Cable Spools by Num Cond**
3. Click a spool name from the spool list. You can filter the spools by spool type or number of conductors.
4. Click **OK** in the **Select Spools** dialog box. You are prompted for a new spool name.
5. At the prompt, type a new spool name and press ENTER. The spool is renamed.
6. Click **Done/Return** in the **SPOOLS** menu.

Note: If you have renamed spools using the **Select Spools** dialog box, all wires or cables in the Pro/CABLING assembly reference the renamed spool. If the assembly logically references a diagram, then renaming a spool in the diagram and updating the logical reference also causes the corresponding spools in the assembly to be renamed. Use **CABLING > Logical Ref > Update** to overwrite the existing cabling spool parameters with the modified spool parameters of the logically referenced diagram.

In a comparison between an assembly and a logically referenced diagram, if a spool in the assembly has the same parameter values as the spool in the logically referenced diagram, then both these spools are recognized to be the same.

To Write Spools

1. Click **CABLING > Spools > Write**. The **Select Spools** dialog box opens.
2. Select one of the following filters to specify the type of spool to be modified:
 - **All Spools**

- **Wire Spools**
 - **Cable Spools**
 - **Cable Spools by Num Cond**
3. Click a spool name from the spool list. You can filter the spools by spool type or number of conductors.
 4. Click **OK**. The selected spool file is output to the directory from where you started the Pro/ENGINEER session or to the `Spool Dir` directory, if you have set the `pro_spool_dir` configuration option.

To Remove Spools

1. Click **CABLING > Spools > Remove**. The **Select Spools** dialog box opens.
2. Select one of the following to specify the type of spool to be modified:
 - **All Spools**
 - **Wire Spools**
 - **Cable Spools**
 - **Cable Spools by Num Cond**
3. Click a spool name from the spool list. You can filter the spools by spool type or number of conductors.

or

Click  or  to select or remove all spools from the list, respectively.

4. Click **OK**. If the spool has cables referencing to it, you are prompted to confirm the deletion.
5. Click **Yes** at the prompt, the selected spools are deleted successfully.

To List Spools

1. Click **CABLING > Spools > List** to view the list of spools in the diagram or cabling assembly. The INFORMATION WINDOW opens listing the spools and the spool type.
2. Click **File > Save As** to save the spool information to the required location.
3. Click **Close** to close the INFORMATION WINDOW.

Modifying Spool Color

If you set the `COLOR` parameter of a spool to a color that is available in the `color.map` file or one that was created and named in the **Appearance Editor** dialog box, all cables or wires created from this spool are created with the specified color. If you change the value of the `COLOR` parameter and regenerate the model, the color is

replaced with the new color. You can change the color of a cable in the design without editing the spool. Use the **Appearance Editor** dialog box to set a color for the cables without editing the spool. All the segments of the selected cable take the new color. Any new segments you create for this cable after setting a new color will use the new color. However, the spool remains unedited. The `COLOR` value for the spool is overridden at the design level.

To Add Spools from a Logical Reference

1. Click **CABLING > Logical Ref > Ref Diagram** to reference Pro/DIAGRAM information directly in Pro/CABLING. This information includes the reference designator, pin-to-pin connection information, and parameter values of connectors, pins, spools, wires, and cables.
2. Click **CABLING > Spools > Create > From Logical**. The **REF SPOOL** menu appears.
3. Select the spools that you want to add to the cabling file or click **Select All** to add all spools.
4. Click **Done Sel**.

Note: If you use **From Logical**, Pro/CABLING automatically creates all spools associated with the created cables and wires. The information used to create these spools is taken from the logically referenced data from which the wires were created.

Adding Wires and Cables

About Wires and Cables

A wire has one conductor. When wires or cables are first created, only the name and a set of default parameters inherited from the spool are present for the wire or cable. Later, you can add additional features or parameters to further define the physical shape of the connection. The wires or cables in the database must also be named and defined before you route them.

Spools contain physical properties and a set of parameters for the cables or wires that can be reused as needed, the same way as the length of a reel wire is rolled off a reel spool. You can import wires, cables, and spools from Pro/DIAGRAM or create them within the Pro/CABLING assembly.

To Create a Wire or Cable

1. After creating or activating a harness part, click **CABLING > Feature > Create**. The **CABLE FEAT** menu appears.
2. Click **Cable** to create a cable or **Wire** to create a wire and associate parameters with it. If the assembly has no reference to a diagram, you are prompted to specify a new cable name. The **SPOOL NAMES** menu appears.
3. Select an existing spool from which the cable or wire draws its characteristics or create a new spool. A new wire or cable is created.

4. Click **Done/Return** in the **CABLE FEAT** menu.

To Add a Wire or Cable to the Database by Reference

1. After creating or activating a harness part, click **CABLING > Feature > Create**. The **CABLE FEAT** menu appears.
2. Click **Cable** to create a cable or **Wire** to create a wire and associate parameters with it. If the assembly has a reference to a diagram, the **CREATE CBL** menu appears.
3. Click **From Logical**, select one or more wire or cable names from a list of names generated from the diagram or wire list displayed in the **REF CABLE** menu, and click **Done Sel** to select the referenced cable or wire.

or

Click **Enter Name** to create a cable or wire without using referenced information from a diagram or wire list and type a name for the cable or wire when prompted.

Note: If the logical data is imported from a wire list, the **From Logical** command is available for wire creation only. This is because the imported wire list format does not support cables.

4. Click **Done/Return** in the **CABLE FEAT** menu.

To Modify Cables and Wires

1. Click **CABLING > Modify** to edit dimensions, parameters, lengths, or locations of cables and wires. The **MOD CABLE** menu appears.
2. Click one of the following commands to modify cables or wires:
 - **Mod Dim**—Modifies the values of cable dimensions.
 - **Parameters**—Modifies parameters of a cable, wires, or bundle for selected cables, wires, or bundles through the **Electrical Parameters** dialog box.
 - **Length**—Allows you to access commands to measure any continuous portion of the selected cable or to fix the length of cable between locations.
 - **Split Cable**—Creates a new cable or bundle by splitting the existing cable or bundle into two parts.
 - **Mod Bundle**—Adds or deletes cables or wires from a bundle.
 - **Cosmetics**—Modifies tiwrap, tape, and marker features through the **Electrical Parameters** dialog box.

About Strip Length Table

A strip length table automatically assigns wire strip information to wires in a cabling assembly. The strip length table is similar to a terminator table.

The first column of the table must contain the required strip length of a conductor. By default, the second column specifies the terminator whose connected wire is stripped. You can modify this value.

As in terminator tables, the valid objects are `COND`, `SPOOL`, `CONN`, or `PIN`. Items specified in the parameter heading can be any parameter from the object heading above it. An asterisk (*) indicates that it applies to all values for the specified parameters.

Wire Strip Information

Use the `STRIP_LENGTH` parameter to specify the amount of insulation to be stripped off from the end of the wire. The `STRIP_LENGTH` parameter specifies the name of the wire to be stripped, the amount of insulation to be removed from the wire, and the end of the wire to be stripped.

You can use a strip length table to automatically assign strip lengths in a batch process.

To Create a Strip Length Table

1. Click **CABLING > Set Up > Strip Table**. The **Strip Table** dialog box opens.
2. Click in a cell in the **COND STRIP_LENGTH** column and click **Insert > Insert Row After**. A new row is added to the table.
3. Type the required strip length in the new cell in the **COND STRIP_LENGTH** column. This column is reserved for the strip length parameter value.
4. Identify a valid object and parameter that you want to associate the strip length with. The valid objects are `COND`, `SPOOL`, `CONN`, or `PIN`. Depending on the object that you specify, you can select a parameter from the list for the second cell.
5. Add new columns to refine the selection. For example, in the following table, the strip length value `0.5` is assigned to all pins that have the term name, `GND`, and have 18-gauge wires.

COND STRIP_LENGTH	PIN TERM_NAME	SPOOL WIRE_GAUGE
COND	PIN	SPOOL
STRIP_LENGTH	TERM_NAME	WIRE_GAUGE
0.5	GND	18

Note: Wildcards such as an asterisk (*) indicate that it applies to all values for the specified parameters.

6. Click **File > ExportTable** to save the strip table file to disk.

7. Click **Execute** to update all the connector parameters with the valid strip information. The **Execute** command updates all wire parameters with an `AUTO_ASSIGN` value of the `STRIP_LENGTH` parameter set to `true`.

Note: Strip table files are run every time you regenerate a cabling assembly.

8. Click **OK**.

Adding Components

About Components

In Pro/CABLING, components are divided into the following types. All components referenced by a 3D harness can be assembled into a flattened harness assembly for the selected 3D harness.

- **Connector**—A part or assembly that serves to connect the harness electrically into a piece of equipment or another harness. You must designate a part as a connector, and define one or more c-systems to use as an entry port before you can route the wire or cable to it. Wires or cables need not end at a connector, they can be ended in space. However, if you move the connector in the assembly and you want the cable to move with it, you must route the wire or cable to a connector entry port.
- **In-line connector**—An inline connector is a male-female pair assembly. It must have an assembly level coordinate system on either side representing an entry point and an exit point. When placed on an existing cable, an inline connector splits the cable into two separately-named entities. The length of each entity is calculated from the entry or exit location point. Internal lengths are added through the entry port `int_length` parameter on the component.
- **Splice**—A part or assembly that electrically connects wires within a harness. Wires can stop at or pass through a splice. A splice does not split the wire into two entities.
- **Mechanical Component**—A part or assembly that serves as a mechanical support for the wiring harness. A mechanical component is a necessary part or assembly in manufacturing a grommet or a formed support piece. Wires must pass completely through this type of component to be electrically meaningful. If you stop routing at a component, the wire is considered incomplete.

The subtypes of mechanical components are:

- **Custom Component**—A component that has no parameters but, in other capabilities, is exactly like a splice. Used for items that would not need a reference designator.
- **Attached Component**—A component that was referenced while routing a harness and then attached to the harness using the **Attach to Harn** command on the **CABLE COMPONENTS** menu. The referenced component at the routing location must be of the type **Along Axis** or **Pnt/Vtx/Csys**,

with a coordinate system as the reference so that you can attach it to a harness.

About Component Parameters

Components use parameters to indicate electrical information at the connector- and pin-level.

When you first designate a connector using the **CABLE COMPONENTS > Designate** command, the connector-level default parameters are established. The pin-level parameters can be specified when you either use a logical reference to read them from a diagram or manually specify them in the **Electrical Parameters** dialog box.

When you designate a connector, Pro/CABLING sets the component parameters at the assembly-level. This allows you to use the same model to represent different electrical components.

Note:

- You can view only the following cabling parameters in the Model Tree:
 - Designation
 - Diameter
 - Logical From
 - Logical To
 - Physical From
 - Physical To
 - Length
- You cannot access or show component parameters in the Model Tree because they are stored differently from the normal parameters.
- The custom components and attached components do not have electrical parameters.
- Splices can use all connector parameters, but the `TYPE` parameter value is `SPLICE`, not `CONNECTOR`.

Component Parameters for Splices before Pro/ENGINEER Release 20

For splices inserted before Release 20.0, a `LINK` connector parameter is necessary to indicate that the two entry ports are connected for routing.

To route a wire through a splice, add the following connector parameter to the splice's parameters:

```
LINK <entryport1> <entryport2>
```

where, `entryport1` and `entryport2` are the entry ports on the connector through which the wire enters and exits. Multiple entries of the `LINK` connector parameter can be used to indicate multiple links between entry ports in a splice.

For example, a splice with a connector parameter, `LINK ENTRY_2 ENTRY_7`; links `ENTRY_2` and `ENTRY_7` so that when you route a wire to the `ENTRY_2` entry port, the wire can automatically be continued from the `ENTRY_7` entry port. If a wire is routed through a splice, the information is not displayed in the wire list.

For splices and custom components assembled in Release 20.0 or later releases, all the entry ports are considered to be automatically linked, without using the `LINK` parameter.

Cable Paths Setup Options

The **Cable Paths Setup** dialog box contains the following options:

- **Component Items**—Displays a list of all cables that touch the component. In the **Components Item** group, the cable name and path (entry ports) are displayed. Each network segment is listed as a row with information in the **Entry** and **Exit** boxes. You can, therefore, control every network segment.
- **Items to Modify**—Displays selected items in the **Components Item** group that you want to modify. For all selected items, you can set the following in the **Path** group:
 - **Entry**—Click  to select a location where you want to place the component or select an item from the **Component Items** group to display its entry port name in the **Entry** box. If all items do not have the same setting for the entry, the text defaults to `As Is` to allow you to keep individual settings for one while setting another value for the other items. You can specify an entry port to override this setting.
 - **Exit**—Click  to select the exit port or select an item from the **Component Items** group to display its exit port in the **Exit** box.

In the **Show** group, you can set and modify the visibility of cable paths inside a splice:

- **Yes**—The internal segment for a cable is displayed between the entry and exit location in the 3D view.
- **No**—The wire display terminates at one entry port and resumes at another entry port. This option is selected by default.
- **As Is**—The visibility of the cable path remains unchanged.

Note:

- The minimum bend radius is ignored for internal segments. Pro/CABLING assumes that the selected path is the shortest path that maintains tangency between the entry and exit z-axis unless the internal path has and shows manually added locations.

- Pro/HARNESS cannot flatten loops. However, it can flatten the internal portions of splices and components even if they form a loop. Pro/HARNESS can also flatten a virtual loop.

To Edit Component Parameters

1. Click **CABLING > Components > Modify Parameters**. The **SELECT** dialog box opens.
2. Select one or more components to modify. Press CTRL to select multiple components.
3. Click **OK**. The **Electrical Parameters** dialog box opens.
4. Modify the required parameters.
5. Click **Apply** and then **OK**.

To Modify Placement of Splices and Custom Components

1. Click **CABLING > Components > Modify Component**.
2. Select one or more Cabling components to modify. To select multiple components, press CTRL and select the required components. The **MOD CONN** menu appears.
3. Click **Placement**. The **COMPONENT WINDOW** and the **Place Component** dialog box open.
4. Modify the component placement by applying one of the following constraint types.
 - **Tangent to Cable**—The attachment coordinate system attaches to the location that you selected earlier with its z-axis tangent to the cable segment at the location.
 - **Perpendicular to Plane**—Select a **Plane, Crv/Edg/Axis**, or **Csys** from the **GEN SEL DIR** menu as a constraint to assemble the splice or custom component. The z-axis of the coordinate system is normal to the selected plane or aligned to the selected curve, edge, axis, or coordinate system. You can modify the direction of the attachment location to be parallel to the z-axis of the attachment entry port.
5. Click **OK**.

To Modify the Internal Portions of Splices or Custom Components

1. Click **CABLING > Components > Modify Component**. The **SELECT** dialog box opens.
2. Select the cabling component you want to modify. The **MOD CONN** menu appears.
3. Click **Internal Portions**. The **Cable Paths Setup** dialog box opens.

4. Modify the internal portions of a component.
5. Click **Apply** and then **OK**.

Note: You can modify internal portions of a inline connector, splice, or a custom type of component.

To Add a Component to a Flat Harness

1. Ensure that the referenced component at the routing location is of the type **Along Axis** or **Pnt/Vtx/Csys**, with a coordinate system as the reference so that you can attach the component to a flat harness.
2. Click **CABLING > Components > Attach to Harn > Add**. The **SELECT** dialog box opens.
3. Select one or more components to attach to the flat harness.
4. If the component references multiple locations, select a location along the axis on the component at the routing location. The component is attached to the flat harness.
5. Click **OK**.

To Modify the Attachment Location of a Component to a Harness

1. Ensure that the referenced component at the routing location is of the type **Along Axis** or **Pnt/Vtx/Csys**, with a coordinate system as the reference so that you can attach the component to a flat harness.
2. Click **CABLING > Components > Attach to Harn > Mod Location**. The **SELECT** dialog box opens.
3. Select a component to modify its attachment location.
4. Select a location to which you want to attach the component and click **OK**. The attachment location of the component is modified.

To Remove a Component from a Flat Harness

1. Ensure that the referenced component at the routing location is of the type **Along Axis** or **Pnt/Vtx/Csys**, with a coordinate system as the reference so that you can attach the component to a flat harness.
2. Click **CABLING > Components > Attach to Harn > Remove**. The **SELECT** dialog box opens.
3. Select one or more components to remove from the flat harness and click **OK**.

Splices and Inline Connectors

About Adding Components to the Cable Path

You can add the following components to a location along the cable path, between the two cable-terminating entry ports:

- **Splice components**—These are components added along the length of a cable. The cable name and the length from connector to connector is not affected after the splice component is installed. You can insert both **Through** and **Butt** type of splice components that are created in Pro/DIAGRAM.
- **Custom components**—These are similar to splice components, except that they do not use parameters. You can use custom components for cabling objects that do not need a reference designator.
- **In-line connectors**—This is a pair of male and female connectors that break the cable into two physical cables but maintain signal continuity between them. The connector assembly must have an assembly-level coordinate system on either side representing an entry point and an exit point. When placed on an existing cable it splits the cable into two separately named entities. The length of each entity is calculated from the entry or exit location point. To add internal lengths, use the entry port parameter `int_length` on the component.

To Insert a Splice or Custom Component

1. Click **CABLING > Components > Create**. The **Place Component** and the **Open** dialog boxes open.
2. In the **Open** dialog box, select a part or assembly and click **Open**. You can select a part that has only a surface or datum curve or atleast a datum coordinate system to define the entry ports. The selected part or assembly appears in the **Component Window** and its name appears in the **Model Name** box.
3. Select an attachment entry port on the model being assembled.
4. Click **Edit > Find** in the **Component Window** to select a coordinate system from the model or directly select the coordinate system in the model that is displayed in the **Component Window**. The selected coordinate system appears in the **Entry Port** box of the **Place Component** dialog box.
5. Select the component type. If you select the **Custom** component type, you cannot select a reference designator.
6. Select the constraint type.
 - If you select **Tangent to Cable**, the attachment coordinate system attaches to the location that you selected earlier with its z-axis tangent to the cable segment at the location.
 - If you select **Perpendicular to Plane**, select a **Plane, Crv/Edg/Axis**, or **Csys** from the **GEN SEL DIR** menu as a constraint to assemble the splice or custom component. The z-axis of the coordinate system is normal to the

selected plane or aligned to the selected curve, edge, axis, or coordinate system. You can modify the direction of the attachment location to be parallel to the z-axis of the attachment entry port.

7. Select an existing location for the component placement in the **Cable Location** box.
8. Select an existing reference designator or create a new one.
9. Click **OK** in the **Place Component** dialog box as well as in the **Component Window**. The component is placed at the specified location and the **ORIENT FIT** menu appears. This menu determines the orientation of the component.
10. Specify the required orientation.
 - **Flip**—Changes the orientation from the positive-z to the negative-z of the coordinate system, or vice-versa.
 - **Twist**—Type an angle value and change the orientation of the splice or component by aligning the x- or y-axis of the attachment entry port to a selected reference. A dimension is created for the twist. You can modify the dimension using the **CABLING > Modify** command.
11. Click **Done** or **Quit**. The **Cable Paths Setup** dialog box opens.
12. Use the **Cable Paths Setup** dialog box to select the paths that each cable takes inside the component or splice. Initially, all items are shown as routed to or through the attachment location.

Note:

- Whole cables cannot enter the component while individual conductors exit the components. Bundles must stop at the last location that is common to all cables. A new bundle that is manually created cannot continue after the component unless all cables in the bundle enter and leave together.
- If the item goes into a component and does not stop at an entry port, it must come out as itself, and not as part of a cable or bundle. Items that go in and continue must exit as well.

To Delete a Splice or Component

1. Click **CABLING > Components > Delete**. The **DELETE/SUPP** and **SELECT FEAT** menus appear. By default, **Select** is selected.
2. Select one of the following commands from the **DELETE/SUPP** menu:
 - **Normal**—Deletes or suppresses selected features.
 - **Clip**—Deletes or suppresses selected features and all features created later.
 - **Unrelated**—Deletes or suppresses all features other than those selected.
3. Use the **SELECT** dialog box to select one or more components to delete and click **Done** in the **SELECT FEAT** menu.

To Insert Multiple Splices at a Single Location

1. Click **CABLING > Components > Create**. The **Place Component** and the **Open** dialog boxes open.
2. In the **Open** dialog box, select a part or assembly and click **Open**. You can select a part that has only a surface or datum curve or at least a datum coordinate system to define the entry ports. The selected part or assembly appears in the **Component Window** and its name appears in the **Model Name** box.
3. Select the attachment entry port on the model being assembled.
4. Click **Edit > Find** in the **Component Window** to select a coordinate system from the model or directly select the coordinate system in the model that is displayed in the **Component Window**. The selected coordinate system appears in the **Entry Port** box of the **Place Component** dialog box.
5. Select **Splice** as the component type.
6. Select the constraint type.
 - If you select **Tangent to Cable**, the attachment coordinate system attaches to the location that you selected earlier with its z-axis tangent to the cable segment at the location.
 - If you select **Perpendicular to Plane**, select a **Plane, Crv/Edg/Axis**, or **Csys** from the **GEN SEL DIR** menu as a constraint to assemble the splice or custom component. The z-axis of the coordinate system is normal to the selected plane or aligned to the selected curve, edge, axis, or coordinate system. You can modify the direction of the attachment location to be parallel to the z-axis of the attachment entry port.
7. Select an existing location for the component in the **Cable Location** box.
8. Select an existing reference designator or create a new one. To select multiple reference designators, hold down the SHIFT or CTRL key and select the required reference designators.
9. Click **OK**. Multiple splices are placed at the selected location.

Note: If you insert multiple splices at a location using this procedure, you cannot modify the orientation of the individual splices. To modify the orientation of the splices, insert single splices and not multiple splices.

To Insert an Inline Connector

1. Click **CABLING > Components > Create**. The **Place Component** and the **Open** dialog boxes open.
2. Select a part or assembly and click **Open** in the **Open** dialog box. The selected part or assembly appears in the **Component Window** and its name appears in the **Model Name** box.
3. Select the attachment entry port on the model being assembled.

4. Click **Edit > Find** in the **Component Window** to select a coordinate system from the model or select a model name from a list of all coordinate system in the model. The selected attachment entry port appears in the **Entry Port** box of the **Place Component** dialog box.
5. Select **In-line connector** as the component type.
6. Select an existing reference designator or create a new one. If you do not specify a reference designator, the part name is used as the reference designator.
7. Select the constraint type.
 - If you select **Tangent to Cable**, the attachment coordinate system attaches to the location selected earlier with its z-axis tangent to the cable segment at the location.
 - If you select **Perpendicular to Plane**, select a **Plane, Crv/Edg/Axis**, or **Csys** from the **GEN SEL DIR** menu as a constraint to assemble the splice or custom component. The z-axis of the coordinate system is normal to the selected plane or aligned to the selected curve, edge, axis, or coordinate system. You can modify the direction of the attachment location to be parallel to the z-axis of the attachment entry port.
8. Select an existing location for the component in the **Cable Location** box.
9. Click **OK** in the **Place Component** dialog box as well as the **Component Window**. The component is placed at the specified location and the **ORIENT FIT** menu appears. This menu determines the orientation of the component.
10. Specify the required orientation.
 - **Flip**—Changes the orientation from the positive-z to the negative-z of the coordinate system, or vice-versa.
 - **Twist**—Type an angle value and change the orientation of the splice or component by aligning the x- or y-axis of the attachment entry port to a selected reference. A dimension is created for the twist. You can modify the dimension using the **CABLING > Modify** command.
11. Click **Done** or **Quit**. You are prompted to name one of the new wires that you have created. The new wire is highlighted.
12. At the prompt, type a new name for the wire. The other section of the wire is highlighted and you are prompted to name it.
13. At the prompt, type a name for this section of the wire. You are prompted to select another entry port on the splice.
14. Select the second entry port on the other side of the connector assembly. A new location is created for the exit point.
15. If there are any locations inside the connector, manually delete them.

Note: Cable lengths are calculated from the originating terminals to the inline connector's coordinate system of the entry port. To add internal lengths, use the component entry port parameter, `int_length`, on the inline connector.

Designating Connectors

About Designating Components as Connectors

An assembly component must be designated as a connector before you use it as a connector. This makes the component eligible to carry pin and entry port information for starting and ending cables.

If you are using a logical reference from a Pro/DIAGRAM file, you can use the `model_name` diagram connector parameter to automatically designate specific components as connectors. After the part is designated, you can route wires or cables to it.

Subconnectors

Pro/CABLING also supports the concept of subconnectors. An example of subconnectors would be the ports in a PC case. The logical reference refers to the case as a connector and uses a node or pin for each port.

In a 3D assembly, the case is referred to as a connector and the parts assembled to the case representing the ports are designated as subconnectors.

Note: The coordinate systems for the entry ports must be defined in the part before the part is designated as a connector in the Pro/CABLING mode.

You can change the designation of a connector by designating it again or by modifying it. When you undesignate a connector, you must strip it of all connector data. Use **Undesignate/Designate** to replace all information in the connector parameters.

To Designate a Component as a Connector

1. Click **Cabling > Components > Designate**.
2. Select the part or subassembly geometry that you want to represent or designate as a connector. The **DES CONN** menu appears.
3. Click **Logical Ref** to select a component from the components defined in the logical reference diagram.

or

Click **Enter Name**. You are prompted for a filename (`.con`) to read connector parameters from. At the prompt, type the name and press ENTER. If you do not specify a filename, the connector is defined using the default values. When the connection is defined, you can modify parameters or entry ports.

Specify the entry ports, that is, the coordinate systems where cables enter the connector.

After the part is designated, you can route wires or cables to it.

Note: If you select a component that is logically referenced from a Pro/DIAGRAM file, components are automatically designated as connectors.

About Autodesignating Components as Connectors

If the Pro/CABLING assembly has a logical reference to a diagram file or an XML file, you can automatically designate a component in the assembly as a connector. Automatic designation of a component as a connector is possible only when the corresponding connector in the diagram file or the XML file has the `<model_name nnn>` parameter associated with it. In this parameter, `nnn` is the model name of the 3D part in the Pro/CABLING assembly that you want to represent as the connector.

If you have not set the `model_name` parameter in your logical reference, you can still use the **Auto Designator** dialog box to select and match assembly components to connectors from the logical reference.

Note: Select **Allow Mismatch** in the **Auto Designator** dialog box to match the components manually.

To Autodesignate Connectors

1. Click **CABLING > Components > AutoDesignate**. The **Auto Designator** dialog box opens.
2. Match a listed connector with a selected 3D component.
3. After matching a component with a diagram connector, use the editing tools for parameters and entry ports from the **Auto Designator** dialog box.

To Undesignate a Connector

1. Click **CABLING > Components > Undesignate**.
2. Select the connector or splice.
3. Click **Done/Return**.

To Edit Parameters for Components, Pin, or Entry Port Data

1. Click **CABLING > Components > Modify Parameters**. The **SELECT** dialog box opens.
2. Select one or more connectors to modify and click **OK**. The **Electrical Parameters** dialog box opens. The selected connectors appear in the left panel in a tree navigation format.
3. Click the plus sign for a component to expand the component you want to edit. The parameters of the pins and entry ports are displayed in the right panel.
4. Under **Display For**, click **Components**, **Pins** or **Entry Ports** to show the parameters associated with them.
5. Modify the required parameters.
6. Click **Apply** and then **OK**.

To Redefine Splice Placement

1. Open the assembly in Standard mode to change the orientation of the splice.
2. Click **CABLING > Components > Modify Components**.
3. Select the splice that you want to redefine. The **MOD CONN** menu appears.
4. Click **Placement**. The **COMPONENT WINDOW** and **Place Component** dialog box open.
5. Change the constraint type to one of the following:
 - **Tangent to Cable**—The attachment coordinate system attaches to the location that you selected earlier with its z-axis tangent to the cable segment at the location.
 - **Perpendicular to Plane**—select a **Plane, Crv/Edg/Axis**, or **Csys** from the **GEN SEL DIR** menu as a constraint to assemble the splice or custom component. The z-axis of the coordinate system is normal to the selected plane or aligned to the selected curve, edge, axis, or coordinate system. You can modify the direction of the attachment location to be parallel to the z-axis of the attachment entry port.
6. Click **OK** in the **Place Component** dialog box. The splice is repositioned to the new location.

About Replacing Connectors

Use the functional interchange group members (**INTRCH GROUP**) to replace the assembly components that represent a connector with wires routed to it. In a functional interchange assembly, each coordinate system that has a wire routed to it on the connector being replaced must have a corresponding coordinate system on the replacement connector with an assigned tag.

Note:

- The harness part must be regenerated to show the wires being routed to the new entry ports on the new connector.
- If the connector is a subassembly, any entry port of any connector must be represented by a coordinate system that resides on the top level of that connector.

To Replace a Connector

1. Select the connector you want to replace.
2. Click **Edit > Replace**. The **Replace Comp** dialog box opens. You are prompted to select the components to replace.
3. Click the **By Model** tab and select one of the following:
 - **Reference Model**—Replaces a component model with a model containing an external reference of the model.

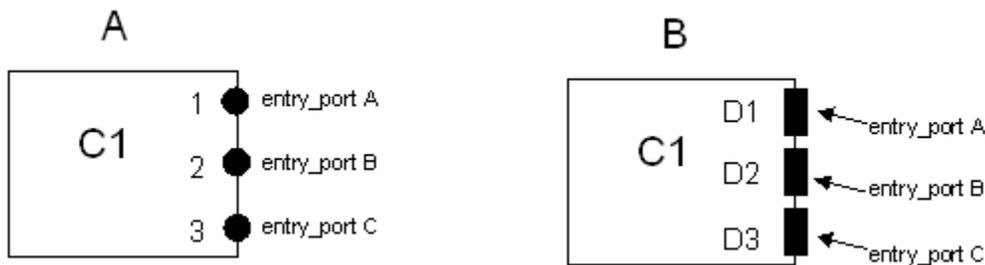
- **Layout**—Replaces a component model with a model associated through a layout.
 - **Manually**—Replaces a component model by specifying placement of a new model.
4. In the **Selected Model** box, select a model to replace.
 5. Click **OK**. The **Component Placement** dialog box opens.
 6. Select the constraints and references.
 7. Click **OK**.

Using Subconnectors

About Subconnectors

When you attach a subconnector to a connector, the entry port information on the subconnector is associated with the parent connector.

A 3D assembly may have components designated as connectors, but no entry ports. The entry ports for these connectors may be on separate parts or subassemblies on the assembly, for example, a PC case and its serial and parallel connectors. Subconnectors are necessary when a Pro/DIAGRAM file represents the PC case as one reference designator, and the connectors as pins. In the 3D assembly, if each serial or parallel connector part is assigned as a subconnector of the case part, the logical reference searches the subconnectors for entry port names listed in the diagram connector's pin information.



A represents a Pro/DIAGRAM connector C1, for example a computer case, showing the case's ports as pins 1, 2, and 3. Each pin has the `entry_port` parameter with a value matching the `entry_port` pin parameter on the subconnector part.

B represents the 3D assembly with parts D1, D2, and D3 as subconnectors of C1. C1 has no entry ports of its own, but each subconnector has one. Each subconnector entry port is available to be assigned to the `entry_port` parameter for pins on C1. Use **Modify Parameters** to assign the parameters to the subconnectors.

To Assign a Connector as a Subconnector

1. Click **Cabling > Components > Modify Component**.

2. Select the component that will receive the subconnector. The **MOD CONN** menu appears.
3. Click **Add SubConn**. Select the connector to be added as a subconnector. A message appears that the subconnector has been added successfully.

Note:

- To remove the subconnector, click **Remove SubConn**.
- To highlight a connectors assigned subconnectors, click **Show Subconns**.
- To list all connector with subconnectors, click **List Subconns**.

To Assign Subconnector Entry Ports

1. Click **CABLING > Components > Modify Parameters**. The **SELECT** dialog box opens.
2. Select the parent component and click **OK**. The **Electrical Parameters** dialog box opens.
3. Under **Display For**, select **Pins**.
4. Expand **Pins** to display the pins used in the assembly.
5. In the right panel, select an **ENTRY_PORT** parameter.
6. In the **Value** box, type a new value or select the entry port name from the list.
7. Repeat this procedure for each pin. Subconnector entry ports are assigned to pin numbers on the parent connector:

Note: Entry ports must first be defined on each subconnector.

Using Entry Ports

About Entry Ports

An entry port is a coordinate system of a connector that you designate as accessible to route a wire, cable, or bundle. Entry ports represent the pins in pin-to-pin cabling connections. A conductor always leaves an entry port tangent to the positive z-axis at the origin of the entry port coordinate system. You can define any coordinate system on a part as an entry port. Typically, you do this when you designate the part as a connector.

Note: To be an entry port, the coordinate system must be at the top level of the model representing the component, that is, at the part level for a part or at the top level of a subassembly.

After a coordinate system is designated as an entry port, you can assign this coordinate system to a specific pin on the connector. You do not have to do this to route a cable to an entry port manually, but the coordinate system is required for autorouting individual conductors within a cable.

To Designate a Coordinate System as an Entry Port

1. Click **CABLING > Components > Designate** to designate a component as a connector.
2. Select the component. The **MOD CONN** menu appears.
3. Click **MOD CONN > Entry Ports**. The **ENTRY PORT** menu appears.
4. Click **Add/Modify** and select a coordinate system.
5. You are prompted to enter the internal length of the cable.
6. Type the length and press ENTER. The **PORT TYPE** menu appears.
7. Select **WIRE, ROUND** or **FLAT** as the port type. Wire denotes a single wire whereas, round or flat refer to the packing for cables. You cannot route a cable to a wire entry port.

You can also use the Autodesignate dialog box to designate the entry ports.

8. Continue to define all the entry ports. When finished, exit the command sequence.

The internal length is not actually routed, but is specified in a numerical parameter for each entry port. These lengths are automatically added to the routed length of a wire when Pro/CABLING provides length information.

When you have designated one or more coordinate systems as entry ports on a connector, the parameters `EP_TYPE` (round, flat, wire) and `INT_LENGTH` (integer) are added for each at the entry port level of the connector in the **Electrical Parameters** dialog box.

To Assign an Entry Port to a Pin Number

1. Click **Cabling > Components > Modify Parameters**.
2. Select one or more connectors with the pins you want to associate. The **Electrical Parameters** dialog box opens.

Ensure that the `NUM_OF_PINS` parameter has an appropriate value to be assigned to the entry ports.

3. Under **Display For**, click **Pins**.
4. If no column are displayed, click **View > Columns** to add the required parameters.
5. In the **Value** box, for each pin, type the name of the coordinate system you want to associate with the pin as a value for the `ENTRY_PORT` parameter or select the entry port name from the list.

To Redefine an Entry Port

1. Click **CABLING > Components > Modify Component**.

2. Select the target component from the assembly.
3. Click **MOD CONN > Entry Ports > Add/Modify**. The **SELECT** dialog box opens.
4. Select an entry port or ports to modify.
5. Click **OK**. You are prompted to assign new values for the internal length and the wire, flat, and round designation for all selected entry ports.

Using Terminators and Terminator Tables

About Terminators and Terminator Tables

A terminator is a cabling object with a set of parameters. Terminators are attached to the pins of a connector to represent contact, crimp, or other terminator style information. The `term_name` parameter is used to assign a terminator to a pin.

Use **CABLING > Terminators > Create** to create a terminator. For a given terminator, you can create any number of user-defined parameters using the **Electrical Parameters** dialog box.

You must first create a terminator and then attach its name to a pin either manually or through a terminator table. In a large design, instead of manually attaching terminators to each pin, you can create a terminator table to automatically assign specific terminators to a specific group of pins.

A terminator table defines the criteria for a terminator to be matched to a particular pin. Specify such criteria in the **Terminator Table** dialog box and run the table. You can add or modify the default criteria to include any connector, connector pin, spool, or spool conductor parameter. By specifying combinations of parameters in a terminator table, any connector pin that meets the required criteria is automatically assigned the specified terminator.

The terminator table columns define the connector and spool parameters. The first column contains the `term_name` value. This value is the terminator name that is to be associated with connections that match the criteria in the remaining columns.

Use the terminator table to assign terminators based on the diameter of the wire and the connector model name. When you regenerate the cabling assembly, the connector terminator statement is updated according to the latest terminator table that you have run. Settings in a terminator table do not override terminators that you have manually assigned, unless you specify otherwise.

Note: Before you assign a terminator to a connector manually or by a table, the `NUM_OF_PINS` parameter must be defined in the connector parameter file.

To Use a Terminator Table

1. Click **CABLING > Set Up > Term Table**. The **Terminator Table** dialog box opens.
2. Type the required terminator name values in the first column as shown in the table below.
3. Use the remaining columns to set the criteria for the `term_name` assignment. Asterisks mean that the value applies to all values of the parameter in the column header. For example:

SET	MATCH	MATCH	MATCH
PIN	CONN	SPOOL	COND
TERM_NAME	MODEL_NAME	WIRE_GUAGE	MATERIAL
mil_14324	conn_A	18	*
mil_14587	conn_B	*	copper

The terminator value `mil_14324` is assigned to all pins that have 18 gauge wires of any material, using the `conn_A` part model. The `mil_14587` value is assigned to all pins on connectors that use the `conn_B` part model that uses a copper conductor of any wire gauge.

4. After specifying the required parameter constraints in the terminator table, click **OK**.
5. Click **File > ExportTable** to save the terminator table to a file.
6. Click **Execute** to update all the connector terminator parameters. The `term_name` parameter with the appropriate value is attached to each pin that meets the set criteria.
7. If required, click **Insert** or **Edit** to add or delete columns.

To Create a Terminator

1. Click **CABLING > Terminators > Create**.
2. At the prompt, type a new name for the terminator. The **Electrical Parameters** dialog box opens.
3. Click **View > Columns**. The **Model Tree Columns** dialog box opens.
4. Move parameters back and forth from the **Not Displayed** to the **Displayed** windows as necessary.
5. If required, define a new parameter by typing the parameter name in the **Name** box and clicking **>>** to move it to the **Displayed** window. The new parameter is added to the current list with `Nonexistent` as the value. This means a value has never been assigned. You can change the value for a single parameter or for multiple parameters.

6. Click **Apply** and then **OK** in the **Model Tree Columns** dialog box.
7. Assign a value to the new parameter in the **Electrical Parameters** dialog box.
8. Click **Apply** and then **OK** to add parameters.

To Read a Terminator

1. Click **CABLING > Terminators > Read**. The **Open** dialog box opens.
2. Select the terminator that you want to read from the disk.
3. At the prompt, type **Yes** if you want to overwrite the existing terminator.

To Modify Terminators

1. Click **CABLING > Terminators > Edit** to modify the terminator parameters. The **Select Terminators** dialog box opens.
2. Select the required terminator or all the terminators from the list and click **OK**. The **Electrical Parameters** dialog box opens.
3. Modify the value of the required parameter in the **Electrical Parameters** dialog box and click **OK** to apply the changes.

Note:

- Use **CABLING > Terminators > Read** to read in terminator parameter data from a previously created terminator that was saved to disk. This file must have a `.trm` extension.
- Use **CABLING > Terminators > Write** to write the selected terminator parameter data to your current directory. These files have a `.trm` extension.

To Write a Terminator

1. Use **CABLING > Terminators > Write** to write the terminator parameters to the disk. The **Select Terminators** dialog box opens.
2. Select the required terminator or all the terminators from the listed that you want to save to the disk and click **OK**.
3. Modify the value of the required parameter in the **Electrical Parameters** dialog box and click **OK** to apply the changes.

Note:

- Use **CABLING > Terminators > Read** to read in terminator parameter data from a previously created terminator that was saved to the disk. The file must have a `.trm` extension.
- Use **CABLING > Terminators > Write** to write the selected terminator parameter data to your current directory. These files are given a `.trm` extension.

To Rename a Terminator

1. Use **CABLING > Terminators > Rename** to rename the terminator. The **Select Terminators** dialog box opens.
2. Select the required terminator from the list and click **OK**.
3. At the prompt, type a new terminator name and press ENTER. The terminator is renamed.

To Remove a Terminator

1. Click **CABLING > Terminators > Remove**. The **Select Terminators** dialog box opens.
2. Select the required terminator from the list and click **OK**. The selected terminators are removed from the diagram or cabling assembly.

To Get Information on Terminators

1. Click **Info > Bill of Materials**. The **BOM** dialog box opens.
2. Select the required options and click **OK**.

A list of all the terminators used and the quantity of these terminators is displayed in the embedded browser.

3. To retrieve terminator information for a given harness or for the cabling assembly, ensure that you have set the required harness part or the cabling assembly as the active model.

The report symbols available to retrieve complete terminator information for a given harness are:

- &harn.term.name
- &harn.term.qty
- &harn.term.<User Defined>

The report symbols available to retrieve complete terminator information for the cabling assembly are:

- &asm.mbr.connprm.name
- &asm.mbr.connprm.pin.name
- &asm.mbr.connprm.pin.term.name
- &asm.mbr.connprm.pin.term.<User Defined>

Where <User Defined> is the parameter defined by you as per your requirement.

Note: Use **CABLING > Terminators > List** to view the list of terminators in the diagram or cabling assembly.

Assigning a Terminator Based on the Sum of Multiple Wire Widths

Use the `MIN_SUM` and `MAX_SUM` parameters to choose terminators based on the total summed area of potentially multiple wires routed to a pin.

SET PIN TERM_NAME	MATCH CONN NAME	MATCH CONN VENDOR	MIN_SUM SPOOL AREA	MAX_SUM SPOOL AREA
term1	conn1	amp	.1	.25
term2	conn1	amp	.25	.5
term3	conn1	amp	* *	

Note: The `term_auto_assign` pin parameter determines whether the pin's `term_name` value can be automatically changed. Before you run a terminator table, ensure that all the terminators you want to edit are eligible for modification, or whether all or some terminators are protected when this parameter is set to `False`. If the `term_auto_assign` parameter has never been set for a terminator, its value does not exist in the **Electrical Parameters** dialog box. You can update such a terminator.

Using Parameters in Pro/CABLING

About Modifying Cable and Wire Parameters

You can modify cable, wire, or bundle parameters by

- Editing the current values
- Retrieving a file containing new parameter data
- Updating the parameters from logical reference data for any matched spool, cable, or wire

When modifying the `THICKNESS` value, remember the following:

- If you modify surface offsets for locations individually, then when you modify the dimension, the offset values are considered fixed and are no longer automatically updated.

You must change the offset dimension manually to ensure that the cable does not interfere with the surface as the diameter at the location increases. Either make the surface offset larger to raise the cable above the surface, or change the location offset back to **Free Height**, and let Pro/CABLING do so.

- If the bundle parameter, `THICKNESS`, is less than the sum of the thickness of all its packed components, Pro/CABLING issues a warning.
- Changing the thickness of a spool does not automatically update the `THICKNESS` parameter of a bundle created from the cables referencing the spool. To update the bundle thickness, you must regenerate the harness part from the **CABLING** menu. This is valid for regular cables as well.

You can add columns to the Model Tree that display the status of a particular feature, the type of feature, or information about a feature (such as parameters). Feature parameters can apply to any feature in an object, while model parameters apply only to an entire model.

Note: You can use the Wire, Cable, or Sheath spool parameters to define the shape of the cabling objects, such as wires and cables. For example, `THICKNESS`, `MIN_BEND_RADIUS`, and so on.

To Add or Delete Parameters of the Cabling Objects

1. Click **CABLING > Modify**. The **MOD CABLE** and **MODIFY** menus appear.
2. Click **Parameters** from the **MOD CABLE** menu. The **SELECT** dialog box opens.
3. Select the wires, cables, components, or connectors to modify.
4. Click **OK**. The **Electrical Parameters** dialog box opens.
5. Under **Display For**, select appropriately to show or hide the display of parameters for different levels of the tree in the left panel. For example, for a component, pin, conductor, entry port, and so on.
6. Click **View > Columns**. The **Model Tree Columns** dialog box opens.
7. Move parameters back and forth from **Not Displayed** to **Displayed** windows as necessary.
8. If required, define a new parameter by typing the parameter name in the **Name** box and clicking **>>** to move it to the **Displayed** window. The new parameter is added to the current list with `Nonexistent` as the value. This means a value has never been assigned. You can change the value for a single parameter or for multiple parameters. If you click **<<**, the parameter name in the **Displayed** window is moved to the **Not Displayed** window and removed from the current list of parameters.
9. Click **Apply** and then **OK** in the **Model Tree Columns** dialog box.
10. Assign a value to the new parameter of any object in the **Electrical Parameters** dialog box.
11. To delete a parameter, select the parameter that you want to delete in the right panel of the dialog box and click **Delete Parameter**.
12. Click **Apply** and then **OK**.

To Specify Parameter Values Individually

1. Click **CABLING > Modify**. The **MOD CABLE** and **MODIFY** menus appear.
2. Click **Parameters** from the **MOD CABLE** menu. The **SELECT** dialog box opens.
3. To ensure that you select a correct cabling object, click  or **Edit > Find**, and select the required object.

or

Select the wires, cables, components, or connectors that you want to modify.

4. Click **OK** in the **SELECT** dialog box. The **Electrical Parameters** dialog box opens.
5. Under **Display For**, select appropriately to show or hide the parameters display for different levels of the tree in the left panel. For example, for a component, pin, conductor, entry port, and so on.
6. Select the parameter you want to edit in the right panel of the dialog box.
7. Type a new value for the parameter in the **Value** box and press ENTER. The new value for the selected parameter appears in the right panel of the **Electrical Parameters** dialog box.
8. Click **Apply** and then **OK**.

Note: Both the cable or wires and bundles cannot be displayed together in the **Electrical Parameters** dialog box.

To Specify Parameter Values for Multiple Objects

1. Click **CABLING > Modify**. The **MOD CABLE** and **MODIFY** menus appear.
2. Click **Parameters** from the **MOD CABLE** menu. The **SELECT** dialog box opens.
3. To ensure that you select a correct cabling object, click  or **Edit > Find**, and select the required object.

or

Select the wires, cables, components, or connectors that you want to modify.

4. Click **OK** in the **SELECT** dialog box. The **Electrical Parameters** dialog box opens.
5. Under **Display For**, select appropriately to show or hide the parameters for different levels of the tree in the left panel. For example, for a component, pin, conductor, entry port, and so on.
6. In the right panel of the dialog box, click `multi-select` for the object type that you have selected. If a parameter for the object has any nonexistent values, you can change all values, including nonexistent values, to the new value or ignore the `Nonexistent` values.

If the column has any nonexistent values, you can choose between:

- **Create and Set**—Changes all values, including nonexistent values, to the new value.
- **Set Existing Only**—Ignores nonexistent fields.

7. Type a new value for the parameter in the **Value** box and press ENTER. The new value for the selected parameter appears in the right panel of the **Electrical Parameters** dialog box.
8. Click **Apply** and then **OK**. All values in a column for a selected object are changed to the new value.

Note: Both the cable or wires and bundles cannot be displayed together in the **Electrical Parameters** dialog box.

To Specify a Value for Pin Names Across Multiple Connectors

1. Click **CABLING > Modify**. The **MOD CABLE** and **MODIFY** menus appear.
2. Click **Parameters** from the **MOD CABLE** menu. The **SELECT** dialog box opens.
3. Select the connectors that you want to modify.
4. Click **OK** in the **SELECT** dialog box. The **Electrical Parameters** dialog box opens.
5. Under **Display For**, click **Pins**.
6. Expand **Common Pins**.
7. In the right panel of the **Electrical Parameters** dialog box, select the specific pin name and the parameter value that you want to edit. Values for most columns are *As Is*. This means that values differ for the same pin name across connectors.
8. Type a new value for the parameter in the **Value** box and press ENTER. The values for the selected parameter are updated across all listed connectors for the specified pin.
9. Click **Apply** and then **OK**.

To Specify a Single Value for Parameters Across Multiple Connectors

1. Click **CABLING > Modify**. The **MOD CABLE** and **MODIFY** menus appear.
2. Click **Parameters** from the **MOD CABLE** menu. The **SELECT** dialog box opens.
3. Select the connectors that you want to modify.
4. Click **OK** in the **SELECT** dialog box. The **Electrical Parameters** dialog box opens.
5. Under **Display For**, click **Pins**.
6. Expand **Common Pins**.
7. In the right panel of the dialog box, click *multi select* to the right of **Common Pins** and above the parameter you want to edit. A prompt appears above the tree headings. Values for most columns are *As Is*. This means that values differ for the same pin name across connectors.

8. Type a new value for the parameter in the **Value** box and press ENTER. The values for the selected parameter are updated across all listed connectors for the specified pin.
9. Click **Apply** and then **OK**.

To Flip Cable Ends

1. Click **CABLING > Modify**. The **MOD CABLE** and **MODIFY** menus appear.
2. Click **Parameters** from the **MOD CABLE** menu. The **SELECT** dialog box opens.
3. Select a cable or wire that you want to modify. The cable or wire must have both from and to connectors.
4. Click **OK** in the **SELECT** dialog box. The **Electrical Parameters** dialog box opens.
5. Click **Flip Cable Ends** to flip the from/to connectors of the selected cable.

To Display Cable or Wire Parameters in the Model Tree

1. In the left pane of the Pro/CABLING window, click **Settings > Tree Filters**. The **Model Tree Items** dialog box opens.
2. Select the features of the Pro/CABLING assembly that you want to view in the left pane of the Pro/CABLING window.
3. Click **OK** to apply the changes and close the **Model Tree Items** dialog box.
4. Click **Settings > Tree Columns**. The **Model Tree Columns** dialog box opens.
5. In the **Type** list, select **Cabling Params**.
6. Type a parameter name in the **Name** box at the bottom left of the dialog box or select a parameter name from the list of available parameters and press ENTER. The new parameter is added to the **Displayed** list.
7. Click **Apply** and then **OK**. The dialog box closes and the new parameter column is added to the model tree.
8. If required, add more columns to the model tree to display the status of a particular feature.

Adding a Parameter to a Note

The value of a parameter evaluated in your cabling assembly can be placed as a note in drawings of 3D or flat harnesses. You can attach the note to a connector, bundle, cable, or wire with or without a leader.

A parameter can be included in the note by adding `¶m_name:att` to the note where `param_name` is the name of the parameter whose value you want to use in the note. The parameter value must exist before the note is created. By default, the

value of the parameter that is extracted is based on the item to which the note is connected.

If you attach `&name:att` as a note to a cable, the name of the cable appears in the note. However, if you want a specific type of parameter to be used, add `_param_type` to the note, where `param_type` is the type of parameter you want to use. For example, you can use the name of a spool used for a wire in a note by using the note, `&name:att_spool`. Attaching this note to a wire shows the name of the spool that was used to create the wire instead of the name of the wire itself.

Cabling Parameters

The following is a list of cabling parameters arranged by object.

Component Parameters

REF_DES (Text String)
 MODEL_NAME (Text String)
 NUM_OF_PINS (Integer)
 TYPE (Text String)
 CONN_PLUG (Text String)
 USER_DEFINED (Text String)
 OBJ_TYPE (Text String)
 DEF_INTERNAL_LENGTH (Integer)
 DEF_GROUPING
 TABLE_AUTO_ASSIGN (True/False)

Connection Parameters

NAME (Text String)
 SPOOL (Text String)
 REF_DES_FROM (Text String)
 ENTRY_PORT_FR (Text String)
 REF_DES_TO (Text String)
 ENTRY_PORT_TO (Text String)
 MAX_ALLOWED_LENGTH (Integer)
 MAX_ALLOWED_DST (Integer)
 TARGET_LENGTH (Integer)
 FROM_TO_CABLE_REPORT_NAME (Text String)
 TO_CABLE_REPORT_NAME (Text String)
 FROM_CABLE_REPORT_NAME (Text String)

Cabling - Help Topic Collection

USE_PATH (Text String)

NETWIRE (Integer)

MIN_ALLOWED_DIST (Integer)

TYPE (Text String)

Pin Parameters

SIGNAL_NAME (Text String)

SIGNAL_VALUE (Text String)

ENTRY_PORT (Text String)

TERM_NAME (Text String)

TERM_AUTO_ASSIGN (True/False)

CABLE_NAME_SHIELD (Text String)

PLUG_MODEL_NAME (Text String)

PIN_NAME (Text String)

CABLE_NAME_# (Text String)

COND_ID_# (Integer)

TABLE_AUTO_ASSIGN (True/False)

TERM_PIN_ID (Integer)

Entry Port Parameters

TYPE_EP (ROUND, FLAT, WIRE)

INT_LENGTH (Integer)

LINK (Text String, another entry port's name)

Bundle Parameters

NAME (Text String)

BUNDLE_TYPE (ROUND, FLAT)

MIN_BEND_RADIUS (Integer)

THICKNESS (Integer)

SPOOL(Text String)

SHEATH SPOOL(Text String)

ADDITIONAL NAMES (Text String)

ASSIGNED_MIN_BEND_RADIUS

ASSIGNED_THICKNESS(Integer)

LINESTYLE

MATERIAL_DENSITY (Integer)
 MIN_ALLOWED_LENGTH (Integer)
 MAX_THICKNESS (Integer)
 MIN_ALLOWED_DIST (Integer)
 OVERLAP_FACTOR

Conductor Parameters

WIRE_NAME (Text String)
 STRIP_LENGTH_FROM (Text String)
 AUTO_ASSIGN_FROM (Text String)
 STRIP_LENGTH_TO (Text String)
 AUTO_ASSIGN_TO (Text String)
 COLOR (Text String) (If a string matches the string present in **View > Colors and Appearance**, cables are displayed in that color.)
 LINSTYLE (Text String) (solidfont, dotfont, and so on)

Routing Cables

About Routing Cables

When you have defined spools in the database and created wires or cables from the spools, you are ready to add actual cables to the design. You can do this manually or automatically.

Routing manually, you first add a cable or cables to a routing set, then install dimensionally constrained points called locations to identify the path of the set. The first location can reference a point on an assembly surface or can be attached to an entry port on a connector. To capture the design intent, the locations may be fixed on surfaces, offset, or dependent on other locations. Therefore, if the assembly surfaces are moved, the cable moves with them.

As you are routing, you can click **Switch End** to activate the opposite end of the cable or **Reset Loc** to reset the start location. Use **Del Portion** to remove a segment between two selected locations.

Autorouting

Using autorouting, you first lay out a default backbone or network of locations where you want the main trunk of the harness. Like cables, the network is associated with the current working harness. The network location points have the same properties as location points used in interactive routing. Additionally they can be assigned priorities by connection to better define the paths for the autorouter.

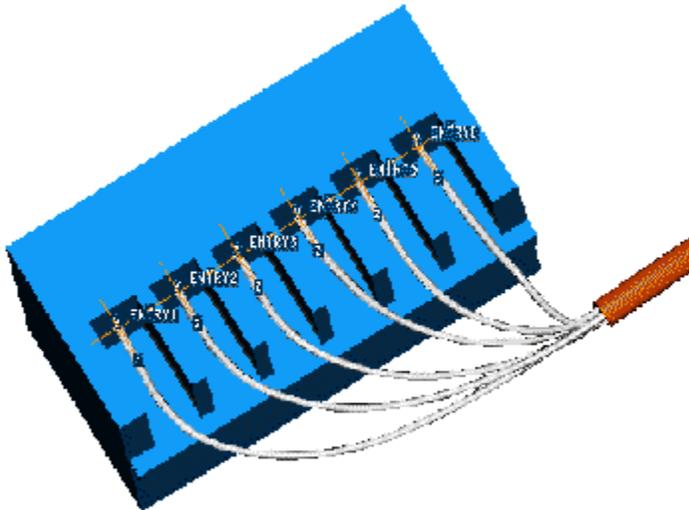
Networks are by default harness-specific, but you can elect to share a network across harnesses, if one network will define a suitable path for two harnesses.

When you have added the network, use the **Cabling > Route** menu to select cables to route, then use the **Autoroute** subcommand to complete the routing.

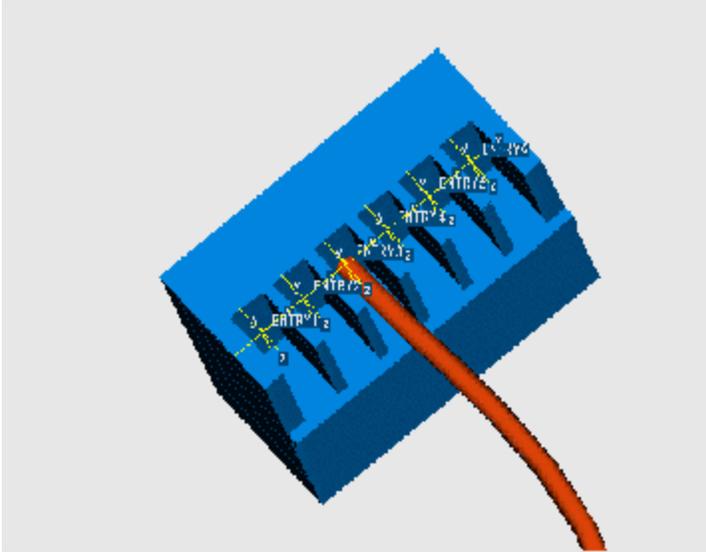
Manual Routing

To add cables interactively, you first select the cable or cables you want to route, then you use the cursor to add locations along the path of the cable. The first and last locations are usually entry ports on connector components.

Locations along the path of the cable can be free or dependent. Free locations move when you move the cable. Dependent locations associate the cable with an offset from a component, so if the component dimensions are edited, the cable length or position is edited accordingly. Cables that you route are added to the current working harness. Use the commands on the **Cabling > Route** menu to start the interactive routing process. In the following figure, the individual conductors of a cable routed to entry ports represent pins.

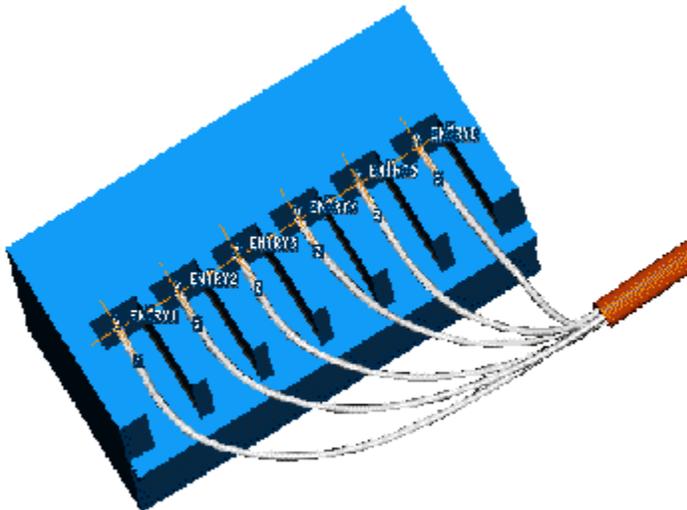


In the following figure, all conductors of a cable are routed to one entry port.



Routing Individual Cable Conductors

You may need the detail of information provided when you route each conductor of the cable separately, as shown in the illustration below. You can accomplish this interactively or automatically through autorouting, if you provide the pin-to-pin from-to and conductor ID information. The most common source of from-to information is a logically referenced ASCII wire list or a Pro/Diagram binary file. To route individual conductors, you must set the `NUM_CONDUCTORS` parameter with an appropriate value for the respective cable spool.



To Control Cable Shape

1. Click **CABLING > Route**.

2. Select the items to route. The **CABLE SEL**, **ADD CABLES**, and **ADD CBL SEL** menus appear.
3. Use the **ADD CBL SEL** menu to select or create a cable.
4. Click **Done Sel**. The **CBL ROUTE** menu appears.
5. Click **Cable Envr**. The **CABLE ENVR** menu appears.

This command lets you control the shape of the routed cable. The settings are stored with the assembly and are recalled from one routing operation to the next, and for all harnesses, until you change the setting. The settings are applicable only for the current Pro/ENGINEER session.

When you route the cable using straight conditions, the cable appears more as a line/arc combination with straight line segments and connecting splines alternating as close to arcs as possible. However, smooth routing is used by default when straight conditions cannot be maintained.

The **Round** or **Flat** type of grouping does not control the grouping of individual items within a bundle routed through a location; that is determined by the bundle-type parameter value.

6. Select the appropriate shape of the routed cable and click **Done/Return** to establish the shape of the cable's cross section.

To Select Cables to Route

1. Click **CABLING > Route**. The **CABLE SEL** menu appears. The **SELECT** dialog box also opens.
2. Select a wire.
3. Click **OK** to complete your cable selection. The **CABLE SEL**, **ADD CABLES**, and **ADD CBL SEL** menus appear.

Using the **CABLE SEL** menu, you can:

- Add selected cables to the routing set
- Remove selected or all cables currently being routed from the set. Use the **Add** command to create a new set before routing.
- Create a bundle containing the selected cables.

You can select conductors from several cables before choosing and routing wires. Use the multiple selection operations, for instance, you can select conductors from several cables before choosing and routing wires.

To filter your selection for routing using the **ADD CABLES** menu, you can:

- Route a continuous segment between two connectors

- Limit your choice of cable selection to only those cables or wires not yet completely routed (that is, attached to two connectors with a continuous length of cable).

Note: If a component of a bundle is completely routed, it is considered complete and does not show in the selection menu if you specify the **Incomplete** command.

If any conductor in a cable is completely routed, that cable is considered complete and does not show in the selection menu.

Any segment of a wire or cable that is already partially routed either between an entry port and a location or between two locations using manual routing is considered incomplete.

- You can select entire cable features for routing or select individual cable conductors for routing if they have been defined individually in the cable spool.

Using the **ADD CBL SEL** menu you can:

- Select existing wires, cables, bundles, or cable conductors to add to the set and a location that will select all items routed through the location.
- Select wires routed to the connector if there is no logical reference; selects those wires routed to the connector in the reference data if there is a logical reference
- Create a cable or wire to be added to the set

If you select the **Create** command, you are prompted to create a new wire or cable. After choosing either **Wire** or **Cable**, you can continue creating the new feature as if you had chosen **CABLE FEAT > Wire** or **CABLE FEAT > Cable**. Wires or cables created now are automatically added to the set of cables you select to route.

The **Create** command allows you to remove all the cables in the set to select a completely new set of cables to route without leaving the **CBL ROUTE** menu or remove the selected cables.

If the current set contains more than one wire or cable, you can create a bundle from any of the cables in the current set. The current set remains active until you change it or end the routing process.

4. Click **Done/ Return** to end the routing process.

To Add or Remove Cables from the Routing Set

1. Click **CABLING > Route**.
2. Select the items to route. The **CABLE SEL**, **ADD CABLES**, and **ADD CBL SEL** menus appear.

3. Use the **ADD CBL SEL** menu to select or create a cable.
4. Click **Done Sel**. The **CBL ROUTE** menu appears.
5. Click the **Change Cbls** command from the **CBL ROUTE** menu.
6. Click the **Add** or **Remove** command from the **CABLE SEL** menu.

To Split Cables

1. Click **CABLING > Modify**. The **MOD CABLE** and **MODIFY** menus appear.
2. Click **Split Cable** from the **MOD CABLE** menu. The **SELECT** dialog box opens.
3. Select a cable that you want to split.
4. Select a location at which you want to split the cable and click **OK**. The harness is regenerated.
5. At the prompt, type different names for the two separate cables. The cable is successfully split into two separate cables at the selected location.

To Route Along an Existing Cable

1. Click **CABLING > Route**.
2. Select the items to route. The **CABLE SEL**, **ADD CABLES**, and **ADD CBL SEL** menus appear.
3. Click **Select** from the **ADD CBL SEL** menu. The **SELECT** dialog box opens.
4. Select the cable and click **OK**.
5. Click **Done Sel** in the **CABLE SEL** menu. The **CBL ROUTE** menu appears.
6. Route to a network location that is used by the cable that you want to follow.
7. Click **Follow Cable** and select the cable to follow the same path as an existing cable.
8. To ensure that you follow the correct cable, click  or **Edit > Find**, and select the required cable.
9. Select another location on the cable that you are following. The cable being routed is now automatically routed upto this second location through all the intermediate locations that the cable being followed goes through.

To Route Through a Hole

1. Click **CABLING > Route**.
2. Select the items to route. The **CABLE SEL**, **ADD CABLES**, and **ADD CBL SEL** menus appear.
3. Click **Select** from the **ADD CBL SEL** menu. The **SELECT** dialog box opens.

4. Select the cable and click **OK**.
5. Click **Done Sel** in the **CABLE SEL** menu. The **CBL ROUTE** menu appears.
6. Click **Along Axis** and select an axis to route the cable along an existing axis. When you select the **Along Axis** command as the first routing command for a segment, you are prompted to select the direction in which the cable is to be routed.
7. To ensure that you select the correct axis, click  or **Edit > Find**, and select the required axis. The **Along Axis** command creates two locations at the endpoints of an axis in one step. The cable is tangent to the axis at the locations. This guarantees that the cable entity passes through the hole normal to the surfaces. (The assumption is that the axis is in a hole.)

If a cable must pass through a hole but does not have to be tangent to the axis, you can create a Free location and select the axis. Only a single location on the axis is created.

Note: If you want to route through clamp-down clips, these parts should be created with an axis and assembled first. Then, route from axis to axis. This method guarantees a smooth trajectory through the parts.

To Route Through an Axis Offset From Surfaces

1. Click **CABLING > Route**.
2. Select the items to route. The **CABLE SEL**, **ADD CABLES**, and **ADD CBL SEL** menus appear.
3. Click **Select** from the **ADD CBL SEL** menu. The **SELECT** dialog box opens.
4. Select the cable and click **OK**.
5. Click **Done Sel** in the **CABLE SEL** menu. The **CBL ROUTE** menu appears.
6. Click **Offset Axis**. You are prompted to select an axis and wire/cable direction.
7. To ensure that you select the correct axis, click  or **Edit > Find**, and select the required axis. The wire/cable is created along this axis and keeps the offset from the surfaces that define the axis placement.

Single flat surface

When creating an axis using a surface and a datum plane, the selected surface will be considered as a boundary for the cable. The cable moves perpendicular to this surface and keeps no interference and no clearance.

"V" shape

When creating an axis using two surfaces, the selected surfaces are considered a boundary for the cable. The cable moves perpendicular to those surfaces and keeps no interference and no clearance.

To Copy a Cable Path

1. Click **CABLING > Route**.
2. Select the items to route. The **CABLE SEL**, **ADD CABLES**, and **ADD CBL SEL** menus appear.
3. Click **Select** from the **ADD CBL SEL** menu. The **SELECT** dialog box opens.
4. Select the cable and click **OK**.
5. Click **Done Sel** in the **CABLE SEL** menu. The **CBL ROUTE** menu appears.
6. With a routing set selected, click **Copy Routing**.
7. Select the cable whose path you want to copy.

The path and the sources of the new cable follow the selected cable.

Each cable selected for routing is routed along a path of the same shape as the source cable. If the source cable has a fixed length, the newly-routed cables have the same fixed length. If a portion of the source cable has a fixed length, the corresponding portion of the newly-routed cables have the same fixed length.

The copied cable starts at one of its designated entry ports. It goes through locations offset from the entry port's coordinate system; these locations are offset from each other by the same amount as the source cable's locations are.

If any cables cannot be completed you are prompted about the problem.

Tip: Updating Locations During Routing

When a cable is routed, Pro/CABLING creates new features to represent both the cable and the locations that the cable passes through.

A cable retains information about where it passes through a location, but the location is responsible for packing all entities that pass through it. Subsequent cables routed to the same location are arranged by the location relative to their neighbors.

During routing, Pro/CABLING may display a message prompting you to regenerate the cables, to see the correct packing on the screen.

In this case, Pro/CABLING is updating the location, but it is possible that some of the earlier cable features are not aware of the location's changes. As a result, not all cables are updated immediately on the screen.

When you regenerate the cables, Pro/CABLING goes back and updates not only each location, but each cable entity as well, giving a completely updated model.

To Set a Cable to a Fixed Length

1. Click **CABLING > Modify**. The **MOD CABLE** and **MODIFY** menus appear.
2. Click **Length** from the **MOD CABLE** menu. The **CABLE LENGTH** menu appears.
3. Click **Fix**. You are prompted to select locations that are the endpoints of the segment.
4. Select the two required locations on the cable to apply a fixed length to a segment. The system displays the current length of the segment which you can accept or modify. You can then use this dimension in a relation.

Note:

- Use **CABLING > Modify > Length > Show** to display the dimensions of a fixed length cable.
- Use **CABLING > Modify > Length > Change** to modify the value of a fixed length cable.
- Use **CABLING > Modify > Length > Free** to remove the fixed length constraint of the cable so that the cable no longer has a fixed length.

To Calculate the Current Length of a Segment

1. Click **CABLING > Relations**. The **Relations** dialog box opens.
2. Click **Utilities > Evaluate**. The **Evaluate Expression** dialog box opens.
3. Specify Cabling mode function `cable_len()` in the **Expression** box to calculate the length of a cable.

To calculate the current length of any segment of a cable or bundle, use the following syntax:

```
cable_len ("cable_name",location_id1,location_id2)
```

To calculate the length for the complete cable from end to end, use the following syntax:

```
cable_len ("cable_name",-1,-1)
```

where

- `cable_name` is the name of the cable being measured in the current harness.
 - `location_id` is the internal feature ID of the location through which the `cable_name` is routed. Where 1 and 2 denote the two cable locations between which the length is to be calculated.
4. Click **Evaluate**. The length of the cable or bundle is displayed in the **Result** box.
 5. Click **Close** in the **Evaluate Expression** dialog box.
 6. Click **OK** in the **Relations** dialog box.

Using Bundles During Manual Routing

You can automatically place wires in a bundle during manual routing, thus reducing the number of picks required to clean-up the routing, and simplify the display of the harness.

When you select **CBL ROUTE > Cable Env** and check the **Rt in Bndls** checkbox, manually routed wires behave the same way as autorouted wires. If wires are manually routed using **FOLLOW CABLE** along a path occupied by a bundle, these wires are automatically added to that bundle. If more than one bundle exists along that path, you are prompted to select which you want to use.

To Delete All or Part of an Individual Cable Segment

1. Click **CABLING > Feature > Delete**. The **DELETE CABLE** menu appears.
2. Click **Cable Portion** and select the bounding locations of the portion to be deleted.
3. If more than one cable is routed through the selected locations, a **DEL CABLE** menu appears that contains a list of all the cables that are routed through the selected locations.
4. Select the cables to be deleted from the menu, and a check mark appears next to the selections.

Note: Deleting a location using **Del Feature** deletes all children (cables and bundles that are routed through the location).

5. If you want to remove a location from the harness but not delete the cable, or to remove a single wire or wires from a location, use **CABLING > Location > Remove**.
6. When you select a cable to delete, the first entity that appears is the cable segment. If you want to delete the entire cable, use **Edit > Find** and select the cable you want to delete.

Note: If you delete all cable or bundle portions from the assembly, this still does not remove it from the list of available cables that can be routed. The cable or bundle feature still exists.

To Get Cable and Wire Info from the Display

1. Click **Info > Cabling** from the menu bar to get information about selected harnesses and cables in the assembly. The **Cabling Information** dialog box opens.
2. Use the **Information Type** options and the associated options to set up an information readout. When you have set up the query, click **Apply**.
3. If you choose any one of the harness options say, **Individual** or **Partial**, the **HARNESSES** menu appears with a list of existing harnesses in the part. Select

the harness whose information you want to display, the INFORMATION WINDOW appears.

Using this procedure, you can get the name, conductor contents, and highlight the path of any selected cable or harness feature.

To Check Cable Clearance

1. Click **Analysis > Model Analysis** from the main menu line. The **Model Analysis** dialog box opens.
2. Select **Pairs Clearance** in the **Type** box.
3. Select **From/To** items to measure the clearance.
4. Select whether you want to check the clearance along the whole cable feature by **Cable** or only for a segment by choosing **Single** entity.
5. Select a cable from the assembly. If you chose the **Cable** command in the previous step, the entire harness is used to measure clearance. If you chose the **Single entity** command in the previous step, the segment you select is used to measure clearance.
6. You can select the **Whole subasm**, **Whole part**, and **Surface** commands as items to measure clearance From and/or To. You can select a harness part using the **Whole Part** command.
7. When you select a harness part with the **Whole part** selection type, Pro/CABLING calculates the clearance between the entire harness part and the other selection (such as a second harness part) in the same way an individual cable is chosen with the **Cable** command.
8. Select the second cable or assembly component.
9. Click **Compute**. The clearance between the selected items is displayed in the **Results** box. If you click **Info**, an INFORMATION WINDOW opens with the clearance results displayed in the window.
10. Click **Close** to exit the **Model Analysis** dialog box.

Note: You can determine the clearance between cables, and between cables and assembly members.

To Check Global Clearance for Harnesses

1. Click **Analysis > Model Analysis**. The **Model Analysis** dialog box opens.
2. Select **Global Clearance** in the **Type** box.
3. Click the **Include** to include harness parts in the **Harness** box.
4. Click **Compute**. Pro/CABLING calculates the global clearance of each harness part and the information appears in the **Clearance** window in the **Setup** box.
5. Click **Close** to exit the **Model Analysis** dialog box.

Note: You can include harness parts in the global clearance checking process. The checking of harness parts does not include the checking of interferences between cables within a harness part. The check is between the entire harness part and other parts, such as assemblies and harness parts.

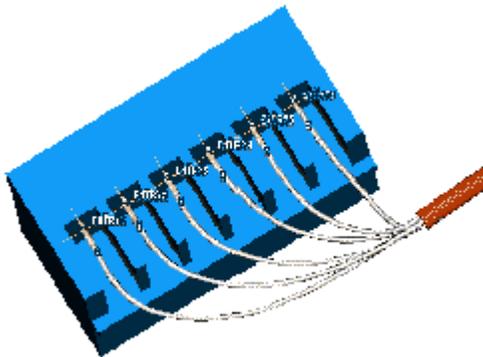
To Route Individual Conductors

Use this procedure to route individual cable conductors to selected entry ports. Cables must be in the Centerline mode. Ensure that you set the `NUM_OF_CONDUCTORS` parameter for the respective cable spools before routing.

1. Click **CABLING > Route**.
2. Select the items to route. The **CABLE SEL**, **ADD CABLES**, and **ADD CBL SEL** menus appear.
3. Select **Conductors** from the **ADD CABLES** menu.
4. Click **Select** from the **ADD CBL SEL** menu. The **SELECT** dialog box opens.
5. Select the wire or cable to route conductors from and click **OK**. The **SEL CONDUCTORS** menu appears.
6. Check the boxes for the conductor or conductors you want to select an entry port for. To route a wire to an entry port, check the wire before you proceed. If you want to route more than one wire to an entry port, check the multiple boxes.
7. Click **Done Sel**, and **CABLE SEL > Done Sel**. The **CBL ROUTE** menu appears.
8. Click **Entry Port** and identify the connector and entry port you want to connect to. A location is established on the entry port.
9. Click **Location** and click on the location at the end of the cable segment, where you want the conductor to come from. The conductor is routed to the entry port.

Repeat from step 1 for each separate conductor you want to route.

In the following figure, individual conductors are routed to different entry ports.



Using Locations

About Locations

Locations are points, similar to datum points, that cables follow during routing. The locations describe the offsets and fixed points of the cable's path through an assembly. Use the locations to capture the design intent in case of changes to the assembly's dimensions. You can assign dimensions and constraints to capture the design intent of the cable and harness if changes are made to the assembly. Locations are also required for autorouting. When you define a path, multiple wires and cables can follow the path during autorouting. Two kinds of location points are:

- **Free**—You can place a free location as a point on any surface, curve, axis, or network segment. It is automatically offset by 0.6 of the radius of the cable or cables passing through the location. Free locations on curves, axes, or other cable entities are always centered on the entity and tangency is not constrained.
- **Dependent**—A dependent location has the same default offset and directional values as the free location, but you are prompted for a dimension from the previously specified location, and an offset from a second selected surface. When the reference entity is moved, the locations move with it.

Use the dependent location when you must route along a complex surface, when the cable must follow a straight line as seen from the top or side view. If you must move the cable, you can move only the reference location, and the dependent locations move with it.

An offset location is similar to a dependent location, but lets you define the location relative to three dimensions. Other location types place the path along a selected axis or parallel to a channel. Use an Offset location to place cables in a tight passage or cramped volume; the cables move relative to assembly geometry. You can use the **Offset** command to create locations relative to a global coordinate system, but you should avoid this exclusively because the resulting trajectory has no reference to the assembly and does not capture your design intent. As with Dependent type locations, Offset locations cannot be used as a starting point for routing a cable set.

After specifying locations of any kind, you can optionally attach a width constraint to them for the cable to be routed through the location.

To Create a Dependent Location

1. Click **CABLING > Route > Select** or **CABLING > Network Ops > Route**. For the both routines, the **CBL ROUTE** menu appears.
2. Click **Free**. Use the pointer to make a new free location as a reference for the Dependent location.

or

Select an existing free location.

3. Click **Dependent** to route from or to a point and dimension to the previous location.
4. Select a point on a curve or surface where you want to install a dependent location. You are prompted to select the first direction.
5. Click  or **Edit > Find** to select an edge, axis, or curve. You are prompted to select a reference for the second direction.
6. Similarly, select an edge, axis, or curve for the second direction. The location is installed where you clicked in step 3.

To Create a Use Dir Location

1. Click **CABLING > Route > Select** or **CABLING > Network Ops > Route**. For the both routines, the **CBL ROUTE** menu appears.
2. Click **Use Dir**.
3. Select a straight curve or cable portion, axis, edge, or channel. The **OFFSET** menu appears.
4. Select one of the following:
 - **Thru_Point**—Creates a parallel datum plane that intersects the point that you specify on the part surface.
 - **Enter Value**—Type an exact dimensional value for the datum to be offset from the plane in the direction of the red arrow. If the arrow points in a direction that is opposite the one where you want to send the location, type a negative number.

Note: To modify the dimensional value of the offset, choose **CABLING > Modify** and then select the location. Click **Regenerate** to view the results.

To Create an Offset Location

1. Click **CABLING > Route** or **CABLING > Network Ops > Route** routines. The **CBL ROUTE** menu appears.
2. Click **Offset**. The **OFFSET LOC** menu appears.
3. Select three straight edges or a coordinate system. No tangency is enforced at the location.
4. Specify the point from which the offsets are measured.
5. Select a command from the **OFFSET LOC** menu.
 - **Specify Dirs**—Specify directions for dimensioning a new location.
 - **Specify Portion**—Select a portion of network to create a offset copy.

- **Specify From**—Specify the point you are offsetting from. Note that this point need not refer to the location from which you are routing. The commands in the **Specify From** menu are:
 - Previous Loc**—(default) The new locations are offset from the previous location. The previous location is highlighted. You can use the **Reset Loc** command to specify the location to be referenced before you start creating the offset location.
 - Select Pnt**—The new locations are offset from a selected datum point or edge or curve vertex.
6. After specifying reference directions and origins, select **Make Locs** from the **OFFSET LOC** menu.

This begins the creation of offset locations based upon given references. An arrow appears for each direction. Type the offset value for each as required. After every three offsets, a new location is created. The command is modal, that is, you can continue to enter offset values using the current directions.
 7. When you are finished entering offsets, or want to specify a new direction or from point, press ESC. This exits the prompt and you can begin again with Step 1.
 8. To return to the **CBL ROUTE** menu, click **Done/Return**.

To Get Information on Locations

1. Click **Info > Location**.
2. Select a location that you want to query. An INFORMATION WINDOW appears that displays the location ID, the cables passing through the location, the feature the location is attached to, and other design-specific information.

To Redefine Location Types

1. Click **CABLING > Location > Redefine**.
2. Select the location to redefine. The **REDEFINE_LOC** menu appears.
3. Redefine the location using one of the following commands:
 - **Pt/Vtx/Csys**
 - **Free**
 - **Dependent**
 - **Use Dir**
 - **Offset**
4. Select a new type of location and recreate the feature.

Note:

- You cannot redefine one free location to be multiple offset locations.

- You cannot redefine a connector location.

Note: Use **CABLING > Location > Reroute** to reroute locations to other references.

To Redefine a Location Along an Axis

1. Click **CABLING > Location > Redefine**. The **SELECT** dialog box opens.
2. Select a location along an axis that you want to redefine. You are prompted to move all the wires, cables, or bundles to another axis instead of redefining the selected location.

Note: You cannot redefine connector type locations.

3. At the prompt, click **Yes**. You are prompted to select an another axis.
4. To ensure that you select the correct axis, click  or **Edit > Find**, select the required axis, and click **OK**. The **DIRECTION** menu appears. You are prompted to define the required direction for the selected axis.
5. Click **Flip** to specify the reverse direction of the arrow. The selected end of the new axis is highlighted.
6. Click **Okay** to accept the direction of arrow. The harness is regenerated and the cables and wires are autorouted to follow the new direction that you have defined.

Note:

- You can switch between **Flat** and **Round** to indicate the grouping of cables as they pass through locations.
- You can modify the rotation of a location with the **Rotate** command on the **MOD LOCATION** menu.

To Edit Location Dimension Values

1. Click **CABLING > Modify**. The **MOD CABLE** and **MODIFY** menus appear.
2. Click **Mod Dim**. The **SELECT** dialog box appears.
3. Select one or more locations to show its dimensions. You can select any number of locations.
4. After they are displayed, you can select the dimension and type a new value in the box.

The dimensions for locations that can be modified are:

- **Free location**—The offset from the surface. If you modify this dimension directly, free locations will not automatically maintain their relative distance off the surface as they change thickness.

- **Dependent location**—For dependent locations on surfaces, there are three dimensions: two for the references selected and one for the offset from the surface. For a dependent location on a curve or cable, only the offset from the previous location, in the direction specified, can be modified.
- **Offset location**—The three dimensions for the three directions specified.
- **Use Dir location**—The offset dimension from the previous location in the direction of the reference.

Note: Alternatively, you can also use **CABLING > Location > Free Height** to adjust the height of the location to the thickness of the cables.

To Constrain Cable Thickness at Locations

1. Click **CABLING > Location > Max Diameter**. The **MAX DIAMETER** appears. The **SELECT** dialog box opens.
2. Click **Set**. You are prompted to select a location or locations to specify a maximum diameter for.
3. Select a location and click **OK**. The current diameter is displayed at the prompt.
4. Accept the current diameter or type a new maximum diameter for the selected location or locations and press ENTER. The selected locations are now constrained to the specified diameter. If adding wires to the cables passing through these locations results in a violation of the constraint, the wires are not added and you are notified of the violation.

Use the **MAX DIAMETER** menu to remove the limit on the maximum diameter for items routed through locations or from all locations in the active harness.

To Move a Location

1. Click **CABLING > Location > Move**. The **MOVE LOC** menu appears.
2. Click **Move** to select a location to move, select a different point on the current surface, and position it using the **SELECT** dialog box. Use **Discard** to undo any relocated positions. When the new positions have been defined, click **Update** to regenerate the new path.

or

Click **Drag** to select a location and visibly drag the location along current direction to a new position. Click to select a new placement and middle-click to cancel. **Drag** allows you to make small adjustments in a rubberband view by selecting a location on the surface or axis.

3. Click **Done** in the **MOVE LOC** menu.

Note: Movement in either move or drag mode depends on the location type and any directional constraints associated with it. You can not move fixed length segments

beyond the length constraint. If you drag a parent location, the child locations follow accordingly.

To Modify Packing at Locations

1. Click **CABLING > Location > Packing**.
2. Select a flat location to modify packing, the **Modify Packing** dialog box opens.
3. Modify the **Cable name** or **Thickness** and set the other settings appropriately.

You can cut and paste items in the list of items at that location to modify the shape of the flat grouping. When you select an item in the list, the item is highlighted in Pro/ENGINEER.

4. Click **Apply** to preview the visual changes to the packing in your assembly without making them permanent and **OK** to close the dialog box.

To Add Locations to Previously Routed Cables

1. Click **CABLING > Location > Add**. You are prompted to select a point on a segment.
2. After selecting a point, click the type of location from the **ADD LOCATION** menu. You are prompted to select a point to add a location between existing locations on previously routed cables.
3. Click to select a point on a curve or surface. You are prompted to define the necessary constraints for the location type.

To Modify the Size of Location Nodes in Drawing

1. Specify a value in drawing units of the detail setting, `location_radius`. This sets the radius of the nodes displayed. You can specify the word `DEFAULT` as the value of this setup option. The default value is 2.
2. If you specify the setting as 0.0, the location nodes are displayed, but do not print.

Note: Modifying the size of the location nodes is helpful when printing drawings.

To Remove Locations

1. Click **CABLING > Location > Remove**. The **LOC REMOVE** menu appears and the **SELECT** dialog box opens.
2. Click **ItemFromLocs** to remove a single item from multiple locations. Select a location to remove. If you select a location at a branch, Pro/ENGINEER identifies a segment, and the **SELECT SEG** menu appears. Click **OK**, choose the second location at the end of the selected segment to delete both the locations, and click **Yes** at the prompt to accept the changes. To remove a single location, select the location and middle-click. the location is removed.

or

Click **ItemsFromLocs** to remove multiple items from multiple locations. Use the **SEL CABLES** menu to select all the items passing through the selected locations and click **Done Sel**. Select another location and click **OK** in the **SELECT** dialog box. To remove the items and the locations, you are prompted to click **YES** at the prompt.

Note: Any location except a **Use Dir** or **Connector** location can be removed as long as at least two locations remain to define a cable segment. A location with children cannot be removed.

Using Locations with Channels

About Channels

Channels are conduits for defining direction, or for terminating cables to a conductive strip (as a ground or as part of the live circuit). Channels are also the basis for modifying cables using cross sections. You can remove an existing channel with the **Remove** command.

To Define a Channel

1. Click **CABLING > Channel > Add**.
2. At the prompt, type a new name for the channel.
3. Select the type of channel you want to add:
 - **Ground**—The channel is a conductive strip connected to ground.
 - **Conduct**—The channel is a conductive strip.
 - **Direction**—The channel establishes a direction for the cable being routed with respect to it.
4. Click **Done** or **Quit**.
5. Select the geometry representing the channel. You can select a surface for a conductive surface and a straight edge, axis, or curve to establish the direction.

To Place Locations in Channels

1. Click **CABLING > Route** or **CABLING > Network Ops > Route**. The **CBL ROUTE** menu appears.
2. Click **Free** or **Dependent** to place a location on a surface in the channel. This surface need not actually belong to the channel.
3. To use the direction of the channel, edge, or axis to route parallel to, click **Use Dir** and select the channel as the reference.

Note: The method to create a loopback in the channel, that is, change direction is to add a Free location on the surface and then continue with more **Use Dir** type locations selecting the channel.

To Move Locations in a Channel Cross Section

If you have created a network using a free location and the second location using the **CABLING > Network Ops > Route > Use Dir** command based on the direction of the channel:

1. Click **CABLING > Location > ByXsec**.
2. Select a channel to cross-section. You can modify locations routed through a channel by cutting a cross-section through the channel, selecting the cables, and moving them to new location.

Note: You can redefine only a free location.

3. Select a point through which the cross-section must pass. You are prompted to confirm the selection point.
4. Click **Yes**. The model appears in a drawing window. You are prompted to select the top edge to orient the cross-section.
5. When you select the top edge, the model orients to the cross-section of the channel. Use the **Zoom** dialog box to zoom in or out of the view and click **OK** in the dialog box.
6. Use the **CABLING > Location > Move**. The **MOVE LOC** menu appears. Use these move commands to move locations in the cross-section.
7. When the edits are complete, click **Done**.

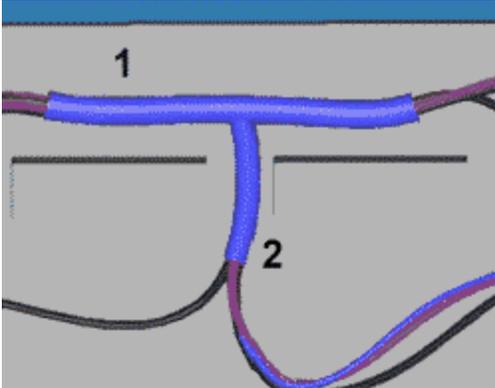
Note: The location must be routed such that there are no other locations that lock the location into a specific place, such as an **Offset** or **Dependent** location.

Using Bundles

About Bundles

A bundle is a sheath feature containing one or more cables, wires, or other bundles that pass through the same location. Bundles are created from sheath spools. You can also create a bundle without a SHEATH spool. They have thickness, color, and minimum bend radius parameters of their own. In lists of cables or wires, bundle names are marked by an asterisk.

You can add cables to a bundle after they are routed or create the bundle as unrouted, add the cables to it, and then route the bundle (and all its cables) at once. To sheath wires that branch from the bundle in a T-junction, you can add a branch bundle at a selected location along the main bundle. The following figure displays a single bundle with two branch bundles.



Branch Bundles

Branch bundles allow cables to leave a bundle in sheathed branches. Use the following ways to determine the branch location:

- Select an existing location on the original bundle, or
- If cables diverge in two directions, select both the locations where the cables diverge from the original bundle and find a midpoint between the two locations.

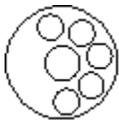
Note:

- While creating bundles before routing, it is not necessary to create a bundle to route multiple cables at the same time.
- The minimum bend radius of bundled wires is the larger of one-half of the bundle diameter or the largest `min_bend_radius` of any component of the bundle.

Bundle Grouping Property

Bundles can be grouped as round or flat. After you create a bundle, you must group the cables. Use one of the following grouping types:

- **Round**—Packs the cables together in a round grouping. This corresponds to the `ROUND` value for the `BUNDLE_TYPE` parameter.



- **Flat**—Packs the cables together in a flat grouping. This corresponds to the `FLAT` value for the `BUNDLE_TYPE` parameter. The bundle diameter is equivalent to the total thickness of each individual component of the bundle.



To Bundle Existing Cables

1. Click **CABLING > Feature > Create.> Bundle.**
2. At the prompt, type a new name for the bundle. The **GROUPING** menu appears.
3. Select the grouping type that is the way the bundle components are packed together in the bundle. The bundle diameter is equivalent to the total thickness of each individual component of the bundle.

The **SPOOL NAMES** menu appears, listing all sheath spools available in this cabling assembly.

4. Select an existing sheath spool to be used to sheath the new bundle or create a sheath spool for the bundle being created. If you select **None**, it specifies that no sheath spool should be used for this bundle. The bundle is created from default parameters. When you have assigned a spool, the **BUNDLE OPTS** menu appears.
5. Select a set of cables for this bundle and click **OK** in the **SELECT** dialog box.
6. At the prompt, type the file name to read bundle parameters from.
7. Click **Along Path**. The **SELECT** dialog box opens. You are prompted to select two locations defining the extents of the sheath.
8. Select the two locations. If any cables branch from the path between the two selected points, you are prompted to specify whether to include the partially routed cables in the bundle. Partially routed cables in this case are the cables that diverge from the bundle path. If the path contains cables that you want to include in the sheath as branches, type **Yes**. If you want these cables to be outside the bundle, type **No**. You are prompted for a parameter file to read into the new bundle.
9. At the prompt, type a file name containing the bundle parameters and press Enter. You can modify these parameters later. The new bundle is created. You can add a bundle sheath feature to a group of cables that share the same path. To share a path, cables use the same location. In thin Cable mode, a single line marks the path of the bundled cables. In thick Cable mode, the sheathing is displayed.

To Create an Unrouted Bundle

1. Click **CABLING > Feature > Create.> Bundle.**
2. At the prompt, type a new name for the bundle. The **GROUPING** menu appears.
3. Select the grouping type that is the way the bundle components are packed together in the bundle. For a **Flat** grouping type, the bundle diameter is equivalent to the total thickness of each individual component of the bundle.

The **SPOOL NAMES** menu appears, listing all available sheath spools.

4. Select an existing sheath spool for the new bundle, click **Create** to create a new sheath spool or **None** to specify that the bundle does not have any sheathing. When you have assigned a spool, the **BUNDLE OPTS** menu appears.
5. Click **Unrouted**.
6. Select a set of cables and bundles that are not yet completely routed, and click **OK** in the **SELECT** dialog box. A dialog box opens with a list of cables and bundles.
7. Click **Done Sel**.
8. At the prompt, type a file name containing the bundle parameters and press ENTER. You can modify these parameters later.

The new bundle is created in the database and appears as a selection when you are adding items to be routed.

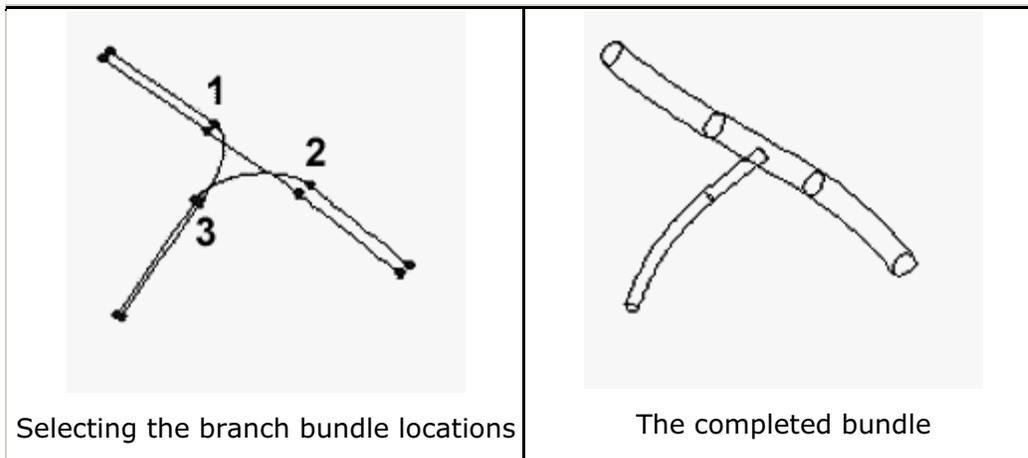
To Create a Branch Bundle

1. Ensure that you have created the main bundle before creating a branch bundle.
2. Click **Feature > Create > Bundle**.
3. At the prompt, type a new name for the bundle.
4. Select the grouping type to specify the way the bundle components are packed together in the bundle. For a **Flat** grouping type, the bundle diameter is equivalent to the total thickness of each individual component of the bundle.

The **SPOOL NAMES** menu appears listing all available sheath spools.

5. Select an existing sheath spool for the new bundle, click **Create** to create a new sheath spool or **None** to specify that the bundle does not have any sheathing. When you have assigned a spool, the **BUNDLE OPTS** menu appears.
6. Click **Branch**. You are prompted to define the extents of the branch.
 - o If your cables converge into the branch from both directions:
 - a. Select the two locations on the main bundle defining the extents of the top of the "T" connection. (Locations 1 and 2 in the figure below.) When you have selected the two locations, you are prompted to select the third location, defining the end of the branch. (Location 3 in the figure below.)
 - b. Select the third location, Pro/Cabling places a new location on the main bundle midway between the extents you defined, and draws the proposed branch bundle as a centerline. You are prompted to enter parameters for the new branch bundle.
 - c. At the prompt, type the parameters or press ENTER to use default parameters, the new branch bundle is created.
 - o If cables enter the branch from one direction:

- a. Select the location where the cable leaves the path, then click **OK**. You are prompted to select the location to be used as the outer extent of the branch.
- b. Select the location, Pro/Cabling draws the proposed branch bundle as a centerline. You are prompted to enter parameters for the new branch bundle.
- c. At the prompt, type the parameters or press ENTER to use default parameters, the new branch bundle is created.



7. Click **Done/Return**.

Note: Use this procedure to add a branch bundle to a regular bundle, where cables leave the shared bundle path. You must use **Along Path** to create the main bundle before you create the branch.

To Modify a Bundle

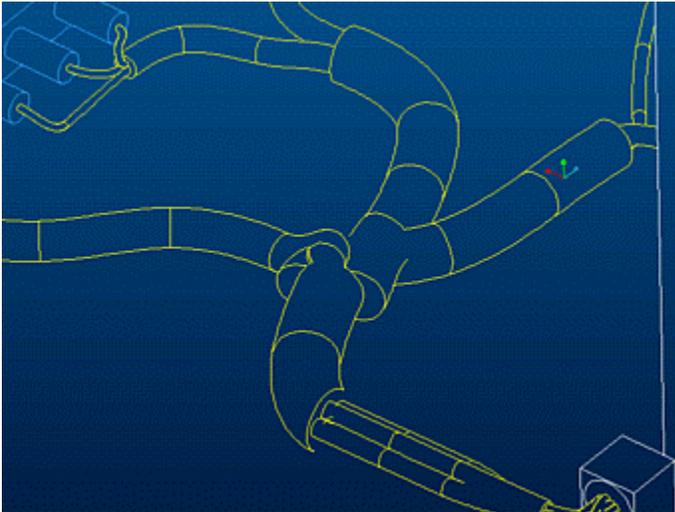
1. Click **CABLING > Modify**. The **MOD CABLE** and **MODIFY** menus appear.
2. Click **Mod Bundle** from the **MOD CABLE** menu.
3. Select a bundle to modify. The **MOD BUNDLE** menu appears.
4. Click  or **Edit > Find** to select the bundles by name.
 - o If you select a branch bundle, a warning message appears at the prompt.
Click **Yes** to extract all cables and remove the bundle itself. After extracting, the cables are left routed, running parallel to the bundle.
 - o If you select a straight bundle, use the appropriate **MOD BUNDLE** commands to add or remove cables from an existing bundle, or to extract all cables from an existing branch bundle.
NOTE: Sections only within the bundle are deleted.
5. Select the appropriate bundle using the **Search Tool** dialog box.

- Click **Apply** and then **OK** or click **OK** to apply and close the **Search Tool** dialog box.

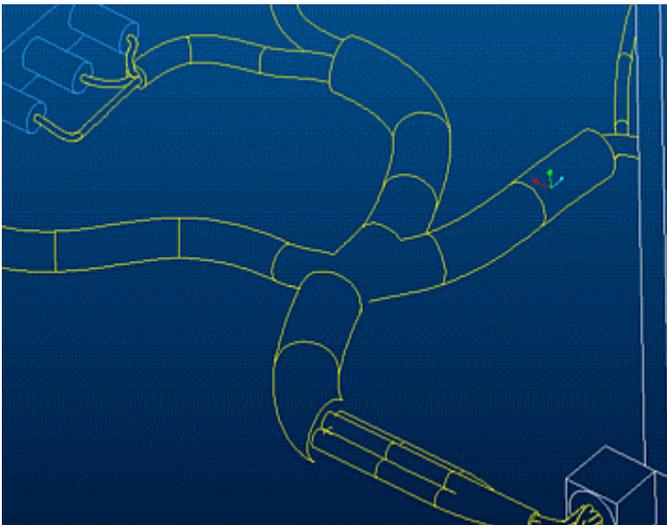
About Aligning and Unaligning Bundles

When several bundles meet at a single location, the bundle ends may be aligned or unaligned by default. If bundle ends at a single location are aligned by default, they can be unaligned. Similarly, if bundle ends at a single location are unaligned by default, they can be aligned.

The following figure shows a bundle that has been split into three smaller bundles. The bundles are unaligned.



The following figure shows bundles after they are aligned.



To Align or Unalign Bundles

- Click **CABLING** > **Location** > **Bundle Alignment**. The **Bundle Alignment** dialog box opens.

2. Select one of the following to align or unalign bundles:
 - **Locations**—Select the locations at which you want to change the alignment. The selected locations are displayed. You can select only those locations at which the bundles end or start.
 - **Align**—Aligns the bundles at the specified location.
 - **Unalign**—Unaligns the bundles at the specified location.
3. Click **Apply** to continue aligning or unaligning bundles at new locations or **OK** to close the **Bundle Alignment** dialog box.
4. Click **CABLING > Regenerate**.

Note: The `align_cable_bundles` configuration file option is used to align or unalign bundles but applies to bundle locations and not the bundles themselves. The default value is `Yes`.

When creating locations, if the `align_cable_bundles` configuration file option is set to `Yes`, bundles at such locations are aligned by default. But if this option is set to `No`, bundles at such locations are unaligned by default.

Use this procedure to align bundles at locations created with the `align_cable_bundles` option set to `No` or to unalign bundles at locations created with the `align_cable_bundles` option set to `Yes`.

To Query Bundle Contents

1. Click **Info > Cabling**. The **Cabling Information** dialog box opens.
2. Under Information Type, click **Physical Routing**.
3. Click **Bundle Items**.
4. Select a bundle to query. The bundle contents are displayed in the INFORMATION WINDOW.
5. Click **Close** to close the **Cabling Information** dialog box.

To Extract Cables from a Bundle

1. Click **CABLING > Modify > Mod Bundle**. The **SELECT** dialog box opens.
2. To ensure that you select the correct bundle, click  or **Edit > Find**, select the required bundle, and click **OK**. The **MOD BUNDLE** menu appears.
3. Click **Extract**. The **SEL BUNDLES** menu appears. All the wires and cables inside the selected bundle are available for selection.
4. Select the wires and cables that you want to extract or click **Select All** to select all the wires and cables.
5. Click **Done Sel** to extract the selected wires and cables or all the wires and cables from the selected bundle.

Autorouting

About Autorouting

Before autorouting the cables or wires, you must establish a logical reference to a Pro/ENGINEER diagram file or a wire list text file in a supported format. Cabling depends on the information in the wire list for the from-to connection information that associates conductors and cables with individual entry ports (pins) on connectors. In a logically referenced diagram, the `ENTRY_PORT` pin parameter must be assigned to the corresponding coordinate system in the selected Cabling assembly. If the `ENTRY_PORT` parameter is not defined in the logically referenced diagram, the wires or cables cannot be autorouted.

After establishing a logical reference, you must set up a network using the **CABLING > Network Ops** command. The network is defined by location points that are not routed. You can specify location priorities in the network, for example, allowed or disallowed, as needed for the network locations. Optionally, you can define paths within the network and assign specific cables to them.

You can accept or reject the resulting routing. In case of errors in your autorouting procedure, the failure information for the wires, cables, or incomplete conductors is saved in the `autoroute.fail` file.

Use the **CABLING > Network Ops > Route > Autoroute** command to route the selected wires or cables between their from and to connectors. The wires automatically follow the shortest path through the network that you set up.

Note: You can use autorouting to complete a harness part without a logical reference if you manually route a small piece of each cable from its last location to its terminating connector. This establishes the connection of the wires, cables, or cable conductors with the pins or connectors. The logical reference method is a faster method.

To Autoroute Wires and Cables

1. If you have performed the setups described in the About Autorouting topic, you can start the autorouting process.
2. Regenerate the harness before you autoroute.
3. Use the **Cabling > Route** commands to add cables and wires to the routing set as you would for manual routing. The **CABLE SEL**, **ADD CABLES**, and **ADD CBL SEL** menus appear.
4. Click **Select** from the **ADD CBL SEL** menu. The **SELECT** dialog box opens.
5. Select the cables or wires and click **OK**.
6. Click **Done Sel** in the **CABLE SEL** menu. The **CBL ROUTE** menu appears.
7. Click **Autoroute**. Pro/CABLING autoroutes the cables you have selected to be routed in the network. The **AUTOROUTING** menu appears.

8. Click **Rehighlight** to display the autorouted cables in your assembly. The wires and cables that you have added are highlighted.
9. Click **Accept** or **Reject** to accept or reject the resulting routing, respectively. In case of errors in your autorouting procedure, the **Autorouting Failure Information** dialog box opens.
10. Click **More Info**. The INFORMATION WINDOW opens and prompts you to look at the `autorte.fail` file information. This error file contains the following information:
 - Wire number
 - Explanation of the problem, for example, insufficient `entry_port` information
 - Suggested fix

Autorouting Bundles

Bundles by themselves cannot be autorouted as they do not extend completely from connector to connector and thus have no From or To logical information. If you select a bundle for routing and attempt to autoroute, the bundles are not autorouted. Use the **Route Wires** command on the **RMV BUNDLE** menu to autoroute the items contained in the bundle, but not the bundle itself.

If you select the **Ignore Wires** command on the **RMV BUNDLE** menu, the contents of the bundle are not autorouted.

Autorouting to Splices

If you designate a splice as the connector for a wire, autorouting routes the wire to the splice as it routes to any connector.

If the splice is not assembled, you can autoroute the wire with one connector and one required location, and later insert the splice at the required location.

Tip: Autorouting through Components

Wires can be autorouted through components. There need not be any network segment that contact two entry ports with an internal cable portion between them. Instead, Pro/CABLING assumes that each time a network segment is routed to a different entry port, all network segments are connected, even though the internal segments do not show.

Troubleshooting for Autorouting

If autorouting fails, read the `autorte.fail` file for solutions to the problem. Other areas that you could investigate are as in the following table:

Autorouting Failure

IF...	THEN...
Autorouting fails, some of the items may have failed or been in a state of failure before the autorouting	Regenerate these items and autoroute again.
Regeneration fails during autorouting	Ensure that you have regenerated the harness before you autoroute.
The harness was regenerated and the autorouting still fails	An autorouted wire may have caused a <code>min_bend_radius</code> violation
Autorouting fails because you added items to a bundle which increased the <code>min_bend_radius</code> , that is now violated	Decrease the <code>min_bend_radius</code> of the bundle by modifying its parameters or modify its routing and autoroute again
The cable is already completed and a continuous segment already exists between two connector entry ports.	Check if the cable is already routed according to the diagram.
The conductor logical information does not match, the cable is completely routed, but the logical information specifies that the cable conductors be routed to separate entry ports on the connector (s).	Delete cable segments nearest to connectors using <code>Del Portion</code> and reroute the individual conductors manually.
There is insufficient entry-port information, the cable cannot be autorouted because the system cannot determine where to attach to the connector(s). Either there is no entry port designated, or there is more than one entry port designated but the pin signal <code>ENTRY_PORT</code> parameter has no value.	Ensure that the entry ports are designated and the pin signal <code>ENTRY_PORT</code> parameter has a value assigned to it.
The logical and physical entry ports do not match, a cable may only attach to two connector locations (except when routing through a splice). For the cable above, the system cannot determine the correct From and To connectors because there are two correct logical connectors in addition to one or two physically routed cable segments routed from other connectors.	Ensure that the names of the logical and physical entry ports for both diagram and cabling match.

<p>No path through the network is found, the location priorities may be set so that the required or disallowed locations prevent a complete path from existing. Autorouting with a network that is not all connected may mean there is no path between connectors. Also, the tangencies of the existing network locations do not allow routing between connectors.</p>	<p>Add additional network segments and autoroute again.</p>
<p>The cable has too many ends to route too many independent segments touching the network for the system to determine which should be used for routing.</p>	<p>Ensure that the cable does not contain more than one manually routed segment.</p>

Using the MBR Failure Diagnostic

If autorouting fails because of an MBR (Minimum Bend Radius) violation, the segment where the violation occurred is highlighted, and a new dialog box opens. This dialog box provides tools to collect more information about the failure and how to fix the failure.

Use this dialog box to:

- Highlight failures in a wire
- Route the wire and ignore failures

If the routing of the wire segments fail upon regeneration, fix the failure using **Resolve** options, or fix it before regeneration.

- Select not to route the failed wire.

Autorouting Networks

About Routing with Networks

A network is a series of locations that defines default paths for autorouted cables. Use the **Network Ops > Route** commands to add locations as you would when routing a cable. The difference is that you do not have a cable attached for the path you are defining. The path is represented by a dotted line.

A network is associated with the cables of the active harness. Cables assigned to other harnesses in the assembly are not routed until their harnesses are made active. A network created in one harness may be made available for routing by another harness if you designate the network as shared. A network is associated with the harness part in which it is created. If you want to make a network available to route more than one harness part, you can share the network between selected harness parts. Sharing makes a network created in a non-active harness available to the active harness.

You can copy a network or network branches at a specified offset distance from the original network. The original locations forming the network are copied at the offset distance. The existing network portion is also copied at the new locations.

Network Properties

When you add network locations, you can also assign priorities to them to further define the autorouting process.

Network bundles and sheathing are created in the same way as other bundles, except that they exist along a branch of the network. Pro/CABLING does not automatically create bundles for the network, but you can assign bundles and sheathing to network segments even if no wires are present yet.

Sheathing on the network can be used for space planning for the harness in early stages of the design.

Assigning Location Priority for Network Locations

To control the path in the network that autorouting uses, you can assign the following priorities to network locations.

Type	Description
Primary (Default)	These network locations are used whenever possible.
Required	<p>Forces all wires being autorouted to pass through the specified location or locations in the same direction directions, even if this does not create the shortest path for each wire.</p> <p>This priority is used to handle wires that are routed through a common splice or to a splice that is to be inserted. You can also use this priority to ensure that wires travel through a specific hole or duct.</p>
Disallowed	Disallowed network locations cannot be used for autorouting. You can avoid overfilling of channels, interference with a fixed size hole in a bulkhead, or creation of loops in an individual harness.
Secondary	A network location that is used for autorouting only if no complete path using primary locations is available.

To Use Location Priority for Autorouting

1. Click **CABLING > Network Ops**. The **NETWORK OPS** menu appears.
2. Select the type of priority you want to assign.
 - **Required Loc**
 - **Primary Loc**
 - **Secondary Loc**
 - **Disallow Loc**
 - **Reset All**

If you click the **Required Loc** command, the **REQD LOCS** menu appears. Select one of the following options:

Select—Select locations through which all autorouted cables must pass. You can have multiple required locations of this type.

Optimize—System finds the shortest paths with at least one common location for autorouted items. This location becomes a required location after autorouting so that you know the location that was selected. This removes all previously set required priority locations.

Put on Wire—System finds the best and shortest paths with a common location on a selected wire. This location becomes a required location after autorouting so that you know the location that was selected. This removes all previously set required priority locations.

Click **Reset All** to reset all network locations used for autorouting to the default primary location.

Note: Modifying the location priority only affects subsequent autorouting operations. Previously autorouted wires are not affected.

To Define the Tangency Direction at Network Branches

1. Click **Cabling > Network Ops**. The **NETWORK OPS** menu appears.
2. Define the tangency of the network branch by using one of the following commands:

Flip Direction—Flips the direction of the network branch at the selected location or locations and also fixes the tangency direction.

Fix Tangency—Fixes the tangency of the network branch at the selected location or locations. The branch direction remains unchanged even if the shape of the network changes.

Display Tangency—Displays an arrow to indicate the direction of the tangency for all the network branch locations that have a fixed tangency.

Clear Fixed Tangency—Clears the fixed tangency for the selected location or locations.

Clear All—Clears the fixed tangency from all the network locations. The branches revert to the shape defined by the system, depending on the geometry of the main network segment.

3. Select the network branch location or locations for which you want to modify the tangency direction.

To Modify Network Locations

1. Click **Cabling > Network Ops > Route**.
2. Use the **CBL ROUTE** commands to edit location placing or type.

You can add, modify, or remove locations from the network.

Note:

- If assembly references are updated, so does the network shape.
- To delete a network segment, use the **CBL ROUTE > Del Portion** command. If you delete a network, the cable locations remain and the cable/network color changes.

To Copy a Network

1. Click **CABLING > Network Ops > Route > Offset**.
2. Select three straight edges or a coordinate system. No tangency is enforced at the location.
3. Specify the network direction.
4. Click **Specify Portion** to select a portion of the cable that you want to copy using the **SELECT** dialog box.
5. Specify the reference directions and network portions.
6. Click **Make Locs** to begin the creation of offset locations based on the given references.
7. An arrow appears for each direction. Type the offset value for each of the three directions. A new network is created that is offset from the original.
8. To create another network at a different offset value from the original network, click **Make Locs** again.
9. When you have finished creating copied networks, and want to return to the **CBL ROUTE** menu, click **Done/Return**.

Note:

- The copied network offset locations are parametric with the existing network locations. If the distances between the locations of the original network are changed, the distances between the locations of the copied network change similarly.
- The offset between the original network and the copied network is controlled by a single dimension that can vary.
- You can use **Modify > Make Indep** to make the dimensions of each copied location independent of the network.
- If the original network is deleted, the copied network is not deleted as the copy is independent of the original network.

To Flatten the Network of a Harness

1. Click **Tools > Options**. The **Options** dialog box opens.
2. Type a new parameter, `fan_with_network` in the **Option** box.
3. Type a value for the `fan_with_network` variable in the **Value** box. Type **Yes** if you want to see the network in flat harness else, type **No**.
4. Click **Add/Change**.
5. Click **Apply** and then **OK**.

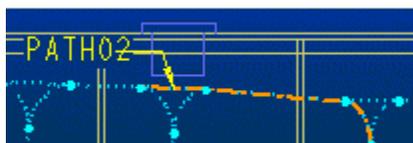
To Share Networks Between Harness Parts

1. Click **Cabling > Network Ops > Share**. You are prompted to select the harness associated with the network you want to share with the current harness.
2. When you select the harness, the model is regenerated and the locations in the shared network are available to the current harness. To directly connect networks across shared harness parts, use the **Connect** command. To undo the sharing, use the **Unshare** command.

Sharing makes a network created in a non-active harness available to the active harness

About Network Paths

After you have defined an autorouting network, you can define paths within the network and associate specific cables or wires with them to further control the autorouting. In the following figure, a part of the network is defined as a path.



When you define a path, you are prompted to name it. The name is attached to the path as a note.

To associate a cable or wire with the path, attach a parameter named `use_path` to the cable or wire and use the path name as the value.

If a path conflicts with other network location priorities, maximum diameter restrictions, or other autoroute restrictions, the autorouting of the wire fails. Autorouting the wire fails, for example, if the specified path contains disallowed locations.

As a network path acts like any other cable, you can modify it so that some portion of the path may no longer be on the network. If both ends of a path are on the network, autorouted cables follow the path (leaving the network and returning). If only one end of a path is on the network, that path is ignored.

To Define and Edit Network Paths

1. Ensure that an autorouting network exists.
2. Click **CABLING > Network Ops > Define Path**.
3. Select a start and end location to define the path.
4. Click **Done Sel**. You are prompted to name the path.
5. At the prompt, type a name and press ENTER. The path is created and the name is attached to a note on the path.
6. To extend or remove the path, click **Network Ops > Extend Path** or **Network Ops > Rename Path**.

To Associate Connections With Network Paths

To specify that a wire or cable must use a predefined network path, add a `use_path` `<path name>` parameter and its value to the wire or cable feature.

Note: To use a different name for the parameter, use the `autoroute_path_param_name` configuration option.

Adding Cabling Cosmetic Features

About Cabling Cosmetic Features

The three types of cabling cosmetic features are tie wraps, markers, and tape. Each is described below.

Note: When creating cosmetic features that represent tie wraps, markers, and tape, you can only select cables and locations that belong to the active harness.

Tie wraps, markers, and tape features have feature parameters. You can assign user-defined colors to the Pro/CABLING cosmetic features. Use the **Appearance Editor** dialog box to change the colors of the cosmetic features. By default, all cosmetic features are assigned the system color, `Sheetmetal`. You can access all tie

wrap and marker feature parameters in Pro/REPORT through the `&asm.mbr.cblprms` category.

Tie Wraps

A tie wrap is used to hold one cable or several cables together. The representation for the tie wrap is as follows:

- In centerline cable display, a circle with a circumference that corresponds to the length of the tie wrap.
- In thick cables environment, a cylinder that shows the appropriate circumference, thickness, and width.

Tie wraps have no required parameters. However, if you specify a name during tie wrap creation, Pro/CABLING implements it as a parameter in the parameter file. You can also assign user-defined parameters to tie wraps using the **Electrical Parameters** dialog box.

Tape Feature

Use this cosmetic feature to show where wires are taped to a thicker portion of the harness in the cabling assembly. The tape feature is located at a single location whereas tape sheathing is wrapped around entire bundle segments of the harness.

As in other cabling features, each tape feature has its own set of feature parameters. Tape features have three required feature parameters:

- NAME
- NUM_OF_WINDS
- SPOOL

Markers

Markers represent shrink wrap tubing that is placed on a cable during the manufacturing process to identify the cable for assembly, maintenance, and repair purposes. Pro/CABLING represents markers as cylindrical features around the selected cable with a name tag to identify the marker. The tag switches on and off with the display of the datum point.

The only required feature parameter is `NAME`. Pro/CABLING automatically includes the value of this parameter in the parameter file when you specify the name of the marker during creation. You can also assign user-defined parameters to markers.

Marker dimensions are assigned when you create the markers.

To Create a Marker

1. Click **CABLING > Feature > Create > Marker**.
2. At the prompt, type a new marker name. The marker name can have a maximum of 31 characters.

3. Select a point or location on the segment of a cable or wire to use as the center for the marker on the cable. The marker length is displayed at the prompt.
 - Click  to accept the default length. The marker expanded diameter value is displayed at the prompt.
 - Type the new marker length, if you want to change the existing length.
 - Click  to cancel the operation. The marker expanded diameter value is displayed at the prompt.
4. Click  to accept the existing value or type a new marker expanded diameter value. The marker reduced diameter value is displayed at the prompt.
5. Click  to accept the existing value or type a new marker reduced diameter value. The cable cosmetic is created successfully.

Note: The thickness of the marker is assigned automatically. You can modify the expanded diameter, reduced diameter, length, and position along the selected cable with the **Modify > Mod Dim** commands.

Marker Default Dimensions and Labels

Default Dimension Name (at Marker Creation)	Default Value	System-generated Dimension Label
length	maximum possible length*/2.0	LENGTH
expanded diameter	cable thickness	MAX Ø
reduced diameter	cable thickness	MIN Ø
thickness	cable thickness/2.0	THICK

Note: The maximum possible length is twice the distance from the point you selected to the closest segment end.

To Create a Tape Feature

1. Click **CABLING > Feature > Create > Tape**.
2. At the prompt, type a unique name for the tape feature.
3. Select an existing tape spool on which to base the new tape feature.
4. Click **Create**, if a tape spool does not exist yet or you want to create a new tape spool.
5. At the prompt, type a unique name for the tape spool. The **SPOOL NAMES** menu appears.

6. Click **Create** or select an existing spool.
7. If you click **Create**, at the prompt type a new name for the spool. The **Electrical Parameters** dialog box appears with the default spool parameters.
8. Modify the required spool parameter values.
9. Click **Apply** and then **OK** in the **Electrical Parameters** dialog box.
10. At the prompt, type the number of times the tape is to wind around the cable or set of cables. The default is 2 times.
11. Select a cable location to use as center for the new tape feature. Pro/CABLING wraps all items at the selected location with the tape feature.

Note: There are no dimensions to modify in a tape feature, as the size is determined by the tape spool parameters. You can modify the tape feature using the **Electrical Parameters** dialog box only.

To Create a Tie Wrap

1. Click **CABLING > Feature > Create > Tie Wrap**.
2. At the prompt, type a new tie wrap feature name.
3. Select a location or point on the segment of the cable to use as a center for the tie wrap. The selected segment of the cable is highlighted. The dimension that locates the tie wrap is created automatically from the start of the cable portion, and all other dimension values are assigned.
 - If a location is chosen, the tie wrap wraps around all cables at that location.
 - If a point along a segment portion is chosen, only the selected item is wrapped by the tie wrap.

The default tie wrap dimension values are assigned automatically as follows:

- **THICK**—cable thickness / 4.0
- **WIDTH**—cable thickness / 4.0
- **LENGTH**—cable thickness * pi

Note: The tie wrap length is not used to calculate the tie wrap diameter.

To Modify Cosmetic Feature Dimensions

1. Click **CABLING > Modify > Mod Dim**.
2. Select the required feature, the feature's dimensions appear.
3. Modify the dimensions press ENTER.

Note: The **Datum Point** display must be set in the **Environment** dialog box to display the tape, tie wrap, and marker labels and the location where they are placed.

To Modify Cosmetic Feature Parameters

1. Click **CABLING > Modify > Cosmetics**.
2. Select the required cosmetic feature (tie wrap, tape, or a marker) you want to edit and middle-click. The **Electrical Parameters** dialog box opens.
3. Edit the parameters of the required cosmetic features using the **Electrical Parameters** dialog box.

Exporting Cabling Geometry

To Export Cabling Geometry

You can export the complete geometry of wires and cables, or just the centerlines, from a Pro/CABLING assembly to IGES and other common formats.

1. Click **File > Save a Copy**.
2. Select the export format in the **Type** box. The **Export Environment** dialog box for the selected environment opens. If the functionality is supported in the selected format:
 - Select **Export Cable Surfaces** to export the cables as surfaces (as seen in thick Cable mode.)
 - Select **Datum Curves and Points** to export cabling harness as centerlines.

Select neither to exclude any cabling geometry information from the export.
3. Click **OK**.

Pro/Report Parameters

Pro/REPORT Parameters for Assemblies

Use the following Pro/REPORT parameters for cabling assemblies:

`&asm.mbr.connprm.<User Defined>`—Lists the values of the specified user-defined parameter for each connector.

`&asm.mbr.connprm.name`—Lists the reference designators of connectors used in the assembly.

`&asm.mbr.connprm.pin.entry_port`—Lists the entry port of the pin of the specified user-defined parameter for each connector.

`&asm.mbr.connprm.pin.name`—Lists the pin names for each connector in the assembly.

`&asm.mbr.connprm.pin.run.<User Defined>`—Lists the specified user-defined parameters for wires, cables, or bundles connected to each connector pin.

`&asm.mbr.connprm.pin.run.cond.<User Defined>`—Lists the user-defined parameters of the given conductor of wires, cables, or bundles connected to each connector pin.

`&asm.mbr.connprm.pin.run.cond.color`—Lists the color of the given conductor of wires, cables, or bundles connected to each connector pin.

`&asm.mbr.connprm.pin.run.cond.name`—Lists the name of the given conductor of wires, cables, or bundles connected to each connector pin.

`&asm.mbr.connprm.pin.run.cond.strip.len`—Lists the strip length of a wire attached to a specified pin connector.

`&asm.mbr.connprm.pin.run.len`—Lists the length of wires, cables, or bundles connected to each connector pin.

`&asm.mbr.connprm.pin.run.name`—Lists the names of wires, cables, or bundles connected to each connector pin.

`&asm.mbr.connprm.pin.run.spool.<User Defined>`—Lists the specified user-defined spool parameters of wires, cables, or bundles connected to each connector pin.

`&asm.mbr.connprm.pin.run.spool.name`—Lists the spool names of wires, cables, or bundles connected to each connector pin.

`&asm.mbr.connprm.pin.signal.name`—Lists the signal names of each connector pin.

`&asm.mbr.connprm.pin.signal.type`—Lists the signal types of each connector pin.

`&asm.mbr.connprm.pin.term.<User Defined>`—Lists the specified user-defined parameters used in the terminators attached to connector pins.

`&asm.mbr.connprm.pin.term.name`—Lists the terminator names attached to connector pins.

Pro/REPORT Parameters for Terminators

Use the following terminator-related parameters in Pro/REPORT:

`&harn.run.cond.from`—Lists the terminator name that every wire and cable is routed from, in the harness.

`&harn.run.cond.to`—Lists the terminator name that every wire and cable is routed to, in the harness.

`&harn.term.qty`—Lists the numbers of every terminator used in the harness.

`&harn.term.name`—Lists all the terminator names used in the harness.

`&harn.term.<User Defined>`—Lists the specific user-defined parameter for every terminator used in the harness.

Note: The parameters `&harn.term.qty`, `&harn.term.name`, and `&harn.term.<User Defined>` must have the harness part as the drawing model.

Harness Related Pro/Report Parameters

You can use several Pro/REPORT parameters in Pro/CABLING cabling assemblies and flat harnesses. You can also use an item's parameters as user-defined parameters in a Pro/REPORT table.

Perform the following steps to set the model before creating a table. Using this table you can access the Pro/REPORT parameters.

1. Open the required drawing and click **File > Properties**. The **FILE PROPERTIES** menu appears.
2. Click **Drawing Models**. The **DWG MODELS** menu appears.
3. Click **Add Model** to add a new part or assembly to the drawing. The selected model becomes the default or current model.
4. If you have added multiple parts or assemblies, click **Set Model** to set the required harness part or flat assembly as the current model.

The report parameters in the following table use `cond` to refer to all wires and cable conductors in a harness and `run` to refer to all wires and cables in a harness. The term `from/to` indicates that the parameter exists for both From and To direction. The `User Defined` category indicates that you can specify user-defined parameters and also system parameters.

For example, although `&harn.run.spool.color` is not specifically defined, you can specify `&harn.run.spool<User Defined>` and type the system parameter `color`.

Pro/REPORT Parameter Name	Definition
<code>&asm.mbr.cblprm<User Defined></code>	Lists the specified user-defined parameter in the cabling components
<code>&asm.mbr.cblprms.name</code>	Lists names of all the user-defined parameters in the cabling components
<code>&asm.mbr.cblprms.value</code>	Lists values of all user-defined parameters in the cabling components
<code>&asm.mbr.connprm.name</code>	Lists reference designators of connectors in the assembly
<code>&asm.mbr.connprm.pin.name</code>	Lists names of pins for each connector in

	the assembly
<code>&asm.mbr.connprm.pin.run.cond.color</code>	Lists the color of the wires and conductors connected to each connector pin
<code>&asm.mbr.connprm.pin.run.cond.name</code>	Lists names of the conductors and wires connected to each pin
<code>&asm.mbr.connprm.pin.run.cond.strip.len</code>	Lists the strip length of wires and conductors attached to each pin
<code>&asm.mbr.connprm.pin.run.cond<User Defined></code>	Lists the specified user-defined parameter in conductors or wires, connected to each pin
<code>&asm.mbr.connprm.pin.run.len</code>	Lists the length of wires and cables connected to each pin
<code>&asm.mbr.connprm.pin.run.name</code>	Lists the names of wires, cables, or bundles connected to each connector pin
<code>&asm.mbr.connprm.pin.run.spool.name</code>	Lists spool names of wires or cables connected to each connector pin
<code>&asm.mbr.connprm.pin.run.spool<User Defined></code>	Lists the specified user-defined spool parameter of wires and cables connected to each pin
<code>&asm.mbr.connprm.pin.run<User Defined></code>	Lists the specified user-defined parameter for wires and cables connected to each pin
<code>&asm.mbr.connprm.pin.signal.name</code>	Lists signal names of each pin

<code>&asm.mbr.connprm.pin.signal.type</code>	Lists signal types of each pin
<code>&asm.mbr.connprm.pin.term.name</code>	Lists the terminator name of each pin
<code>&asm.mbr.connprm.pin.term<User Defined></code>	Lists the user-defined parameters set for the terminators
<code>&asm.mbr.connprm<User Defined></code>	Lists values of the specified user-defined parameter for each connector
<code>&asm.mbr.connprm.pin.entry_port</code>	Lists the entry port parameter of each pin for each connector
<code>&asm.mbr.connprm.pin.signal<User Defined></code>	Lists the specified user-defined parameter for each pin
<code>&asm.mbr.cparams.name</code>	Lists the names of all user-defined parameters in the assembly components
<code>&asm.mbr.cparams<User Defined></code>	Lists the specified user-defined parameters used in the assembly components
<code>&asm.mbr.cparams.value</code>	Lists the values of all user-defined parameters in an assembly component
<code>&asm.mbr.name</code>	Lists the names of assembly members
<code>&asm.mbr.param.name</code>	Lists the names of all user-defined parameters in an assembly member
<code>&asm.mbr.param.value</code>	Lists the values of all user-defined parameters in an

	assembly member
<code>&asm.mbr.type</code>	Lists the type (part, assembly, or terminator) of an assembly member
<code>&asm.mbr<User Defined></code>	Lists the specified user-defined parameter for the respective assembly components
<code>&harn.run.cond.color</code>	Lists the color for every routed wire or cable in the harness
<code>&harn.run.cond.from/to.conn.name</code>	Lists connector reference designators that every wire or cable conductor is routed from connector to connector
<code>&harn.run.cond.from/to.pin.name</code>	Lists pin names that every wire or cable conductor is routed from connector to connector
<code>&harn.run.cond.from/to.conn<User Defined></code>	Lists the specified user-defined connector parameter for every wire or cable conductor
<code>&harn.run.cond.spool<User Defined></code>	Lists the specified user-defined spool parameter for every wire or cable conductor
<code>&harn.run.cond.from/to.pin.entry_port</code>	Lists the pin entry port for every wire or cable conductor
<code>&harn.run.cond.from/to.pin.sig.name</code>	Lists the pin signal name for every wire or cable conductor

<code>&harn.run.cond.from/to.pin.sig.type</code>	Lists the pin signal type for every wire or cable conductor
<code>&harn.run.cond.from/to.pin.sig<User Defined></code>	Lists the specified user-defined pin parameter for every wire or cable conductor
<code>&harn.run.cond.from/to.strip.len</code>	Lists the strip length of each wire or cable conductor
<code>&harn.run.cond.from/to.term.name</code>	Lists the terminator name for every wire or cable conductor
<code>&harn.run.cond.name</code>	Lists every routed wire or cable conductor in the harness
<code>&harn.run.cond<User Defined></code>	Lists the specified user-defined parameter for every wire or cable conductor in the harness
<code>&harn.run.from/to.conn.name</code>	Lists connector reference designators that every wire and cable is routed from connector to connector
<code>&harn.run.from/to.conn<User Defined></code>	Lists the specified connector user-defined parameter that every wire or cable is routed from connector to connector
<code>&harn.run.len</code>	Lists lengths of routed wires or cables
<code>&harn.run.name</code>	Lists names of routed wires or cables

&harn.run.spool.name	Lists the spool name for every routed cable or wire
&harn.run.spool<User Defined>	Lists the specified user-defined spool parameter for every wire or cable
&harn.run.subharn.name	Lists subharness names used
&harn.run<User Defined>	Lists the specified user-defined wire, cable, or bundle parameter for every harness
&harn.spool.len	Lists the total lengths of routed wire spools and cable spools
&harn.spool.name	Lists the wire or cable spool names used
&harn.spool<User Defined>	Lists the specified user defined parameter for every wire or cable spool used

The following conditions apply:

Report Parameter	Active Model	Attribute
&harn...	harness part	NA
&asm.mbr.cblprm(s)...	3D assembly	Cable Info
&asm.mbr.connprm...	3D assembly/Flat assembly	NA

Note: The **Cable Info** attribute is available only when you select at least one cable parameter in the repeat region.

To assign a different model to a region:

1. Click **Table > Repeat Region**. The **TBL REGIONS** menu appears.
2. Click **Model/Rep** to display a different model or simplified representation to control a region. If you change the model associated with the repeat region, data in the repeat region reflects the new model data.
3. Select a region. The **Open** dialog box opens.

4. Select a new part or assembly to be associated with the repeat region.

Glossary

Glossary of Terms

Term	Definition
Harnesses	A harness is a cable part you create within the assembly context and has a complete information of wires, cables, and cosmetic features in a cabling assembly. One assembly can contain several harnesses, and one harness can contain different cables, as connected or unconnected segments.
Bundles	A bundle is made up of one or more cables, wires, or other bundles that are represented as sheathed or wrapped. Bundles are created from spools that carry properties like <code>min_bend_radius</code> , <code>color</code> , and <code>bundle_type</code> (round or flat shape). You must add cables to a bundle after you have added them to the routing set and before you begin to route.
Channels	Channels are conduits for defining direction or for terminating cables to a conductive strip as a ground or as part of the live circuit. Channels are also the basis for modifying cable or network locations in cross sections.
Connectors	Connectors are parts that have parameters you can use to define the beginning and end of cables. You must designate a part as a connector, and define one or more coordinate systems to use as an entry port before you can route the wire or cable to it.
Custom components	Custom components are similar to spliced components except they do not use parameters. You can use custom components in place of cabling components that do not need a reference designator.
In line connectors	In line connectors are pairs of male-female connectors which break the cable into two physical cables but maintain signal continuity between them.
Locations	Locations are the user-defined points within the assembly that describe the path that cable routing should follow. Locations can be free-floating on one selected surface.
Network	A network is a string of locations without cables attached to them that define a path for autorouting. Networks are not cable specific; any number of cables or wires can share a

	network.
Network Paths	A network path is a portion of the network with a name, defined between two locations, and passes through a series of locations in an network. This path can be assigned to a specific cable using the <code>use_path</code> config option.
Splice components	The splice components are components added somewhere along the length of a cable. The cable name and length from connector to connector is not affected after the splice component is installed.
Through Splice	Added along the length of the cable and will not split the cable.
Butt Splice	Splits the cable into two parts
Spools	A spool is similar to an actual spool from which you roll off an amount of cable or wire, as needed. Spools are collections of parameters like color, gauge, and so on that you reference when you create individual wires and cables. When you edit the spool, the edits are passed to the wires or cables that were created from it.
Wire spools	Contains the basic descriptive parameters such as color, thickness, and so on for one conductor or import them.
Cable spools	Describes the cable sheath and certain number of conductors. You must create spools in the cabling assembly or import them from a logical reference before you route the wire or cable. Spool files are saved with the <code>.spl</code> extension.

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