Pro/ENGINEER®
Wildfire™ 2.0

Pro/DIAGRAM™
Help Topic Collection

Parametric Technology Corporation

Third-Party Technology Information

Parametric Technology Corporation, 140 Kendrick Street, Needham, MA 02494 USA

Table Of Contents

Pro/DIAGRAM

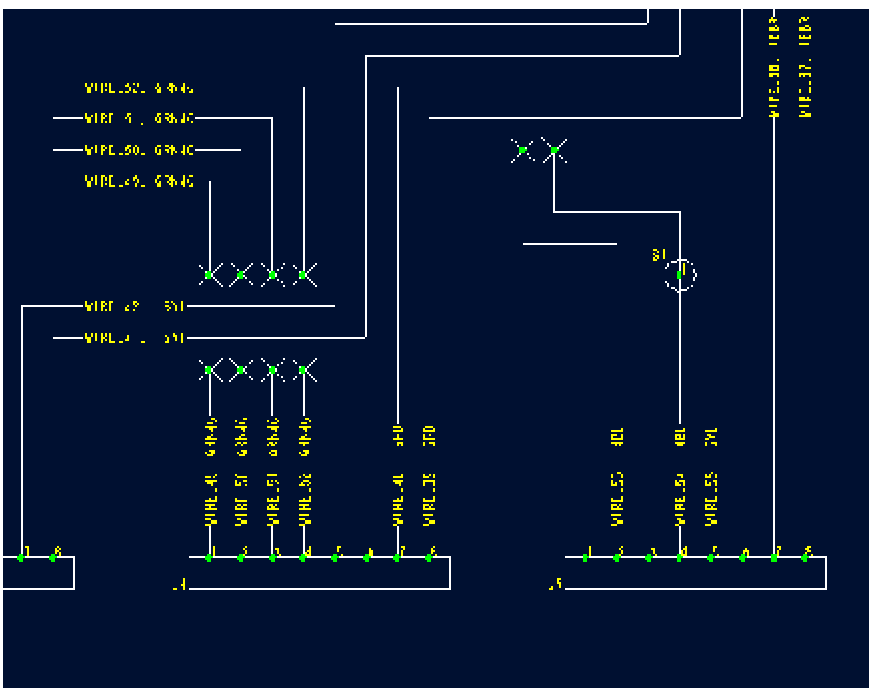
Using Pro/DIAGRAM

About Pro/DIAGRAM

Pro/DIAGRAM captures logical connection information and represents it graphically as wires, cables, components and pins. You can use Pro/DIAGRAM to output component lists and wire lists, either as schematic reports or as formatted ASCII text files.

Diagram-specific tools allow you to:

- Use symbol libraries of electrical and mechanical components.
- Perform quick and easy routing of connections between components with automatic generation of wire lists.
- Compare the diagram to its corresponding cabling assembly or wirelist information from an ASCII file.
- Organize the diagram by layers. You can place wires, connectors, and cables on a layer, even if they are on different sheets.
- Produce BOMs and wire and connection lists that you can pass to Pro/CABLING.



A basic wiring diagram showing connectors, nodes, wires wire labels and wire breaks.

Additional standard tools in Pro/DIAGRAM include the ability to add formats, draft geometry, and notes. Also, all standard Pro/ENGINEER import and export functions, such as IGES and DXF, and all standard plotting options are available within the Pro/DIAGRAM environment.

To Create a New Diagram

Before you start to create diagram objects:

- Make sure you have access to the necessary symbols. Set up your configuration file to point to the appropriate library directories.
- If you are going to work with Grid Snap On, modify your grid spacing to correspond to the required pin spacing in the diagram.

To Create a New Diagram:

1. Click **File** > **New**. The **New** dialog box opens.
2. Click **Diagram** in the **Type** box to start the Diagram module.
3. Type a new diagram file name or use the default file name format (`dgm000#`) and click **OK**. The **New Diagram** dialog box opens.
4. Click one of the following options. The **New Diagram** dialog box contains the following group boxes:

   Specify Template

   - **Empty With Format**— Lets you retrieve a saved format.
   - **Empty**—Lets you use the **Orientation** and **Size** Controls.

   Orientation

   - **Portrait**—Uses standard sizes for portrait, for example, 8.5 inches x 11 inches.
   - **Landscape**—Uses the standard sizes for landscape, for example, 11 inches x 8.5 inches.
   - **Variable**—Lets you create a size. If you select this command, the **Inches** and **Millimeters** command buttons become available as do the **Width** and **Height** boxes.

   Size

   - **Standard Size**—You can click a size, for example, Size A. As you change the standard size, the values of the width and height change in the grayed boxes.
   - **Width**—When you select the Variable button, you can set the width in inches or millimeters.
   - **Height**—Set the height in inches or millimeters.
5. Click **OK**. The **DIAGRAM** window opens. You are now ready to create objects for your diagram.

Note: If the Diagram menu structure ever disappears from view, click **Window** > **Activate** or click the current diagram, and the module menu reappears. This is true only if you have one diagram in session.

Tip: Case in Filenames

The diagram can be saved to disk using the standard **Save** and **Save As** options.

Diagram files are saved as `diagramname.dgm.#`, where `diagramname` is the name of the diagram, and `#` is the version number.

Note: In UNIX, Windows NT, and Windows 95, the actual filenames on disk must use lowercase characters only.

In both UNIX and Windows NT, directory names can contain a mix of uppercase and lowercase characters. However, if Pro/ENGINEER encounters two or more directories in a path that have the same parent and the same name except for a different mix of uppercase and lowercase characters, it accesses only the directory with the earliest uppercase letters (because their ASCII values are lower). This occurs even if you enter the full pathname with the correct case sensitivity.

Drawing Setup File Options for Diagram

The following drawing setup options can also apply to diagrams.

Drawing Option	**Description**
`drawing_text_height`	Sets the default text height for all text in the drawing using the value set for `drawing_units`.
`text_thickness`	Sets the default text thickness for new text after you regenerate and existing text whose thickness has not been modified. Enter the value in drawing units.
`text_width_factor`	Sets the default ratio between the text width and text height. The system maintains this ratio until you change the width with the **Text Width** option.
`default_font`	Sets the default text fonts to be those fonts listed in the specified font index. Do not include the `.ndx` extension. The fonts *font* and *filled* are in the setup file.
`draw_arrow_length`	Sets the length of the leader line arrows.
`draw_arrow_style`	Sets the style of arrows.
`draw_arrow_width`	Sets the width of leader line arrows. This drawing setup file option drives these other

	drawing setup file options: • `draw_attach_sym_height` • `draw_attach_sym_width` • `draw_dot_diameter`
`draw_attach_sym_height`	Sets the height of leader line slashes, integral signs, and boxes. If you specify default, uses the value set for `draw_arrow_width`.
`draw_attach_sym_width`	Sets the width of leader line slashes, integral signs, and boxes. If you specify default, uses the value set for `draw_arrow_width`.
`draw_dot_diameter`	Sets the diameter of leader line dots. If you specify default, uses the value set for `draw_arrow_width`.
`leader_elbow_length`	Determines the length of the leader elbow (the horizontal leg attached to the text).
`decimal_marker`	Determines which character marks the decimal point in secondary dimensions.
`drawing_units`	Sets the units for all drawing parameters.
`line_style_standard`	Controls text color in drawings. Unless you set this option to `STD_ANSI`, shows all text in drawings in blue, and shows the boundary of the detail view in yellow.
`max_balloon_radius`	Sets the maximum allowable balloon radius. If set to `0`, balloon radius depends only on text size.
`min_balloon_radius`	Sets the minimum allowable balloon radius. If set to `0`, balloon radius depends only on text size.
`sym_flip_rotated_text`	Flips any text in a **Rotate Text** symbol that is upside down, making it right-side up. If set to `yes`. and the symbol orientation is +/- 90 degrees, flips the text, rotating it along with the symbol.
`text_thickness`	Sets the default text thickness for new text after you regenerate and existing text whose thickness has not been modified. Enter the value in drawing units.

`pos_loc_format`	Controls appearance of `&pos_loc` text in notes and report tables. `%%=<a single %>,%s=<sheet number>, %x=<horizontal position> %y=<vertical position> %r=<end of repeatable substring>` (Default is `%s%x%y,%r`)

To Display Diagram Information

The following commands in the **Info** menu can help you obtain information about your diagram:

- **Bill of Materials**—Lets you generate a bill of materials (`dgm_bom.inf`) of all the objects in the diagram.
- **Bill of Materials for Layer**—Lets you generate a bill of materials only for one layer.
- **Wire List**—Lets you generate a from-to wire list for all the wire connections in the diagram.
- **Wire List for Layer**—Lets you generate a from-to wire list for only one layer.
- **MVC Instance**—Displays information of a multi-view component.
- **Component Group**—Displays information of a component group instance.
- **Draft Entity**—Displays entity information.
- **Draft Grid**—Displays grid information in the message area.
- **Save Note**—Writes note to a file.
- **Session Info**
    - **Object List**—Displays names of all objects in the current session.
    - **Message Log**—Displays old messages.
    - **Date and Time**—Displays date and time.

To Search for Diagram Objects

To locate objects in a large, complex diagram:

1. Click **Edit** > **Search in diagram**. The **Select By Type** dialog box opens.
2. Use the dialog box to set up a search by parameter or parameters. Define each parameter with its value and use the down arrows to add them to the search list. Select the **And** and **Or** option as required.

   Note: Reference objects are components or connectors.
3. Click **Find** to highlight any of the found items on the current sheet.

4. Click **Info** to open an INFORMATION WINDOW that displays object details and the sheet number location.

Using Layers in Pro/DIAGRAM

You can manage diagram objects by placing them on layers You can then produce report tables, a wirelist or a BOM for all layers, or all objects on selected layers. Use the **Edit** tab options from the left pane of the Drawing window to control layers.

Tip: Selection Shortcut: the Modify Item Pop Up

1. Select a diagram item and right-click. The shortcut menu appears.
2. Select **Properties**. The commands that are displayed depend on the item you have selected.
3. Click **Done** to complete the task and close the shortcut menu.

Undo Redo Operations in Pro/DIAGRAM

To limit the number of undo and redo action to fewer than the default of 50, set the `general_undo_stack_limit` configuration option to a lower number. When available, Undo and Redo commands appear on the menus.

Undo and redo operations are not available for the following commands:

Edit Menu Commands

- **Edit** > **Transfer All Connections**
- **Edit** > **Cable Contents** > **Add Wires**
- **Edit** > **Cable Contents** > **Remove Wires**
- **Edit** > **Highway Wires** > **Add Wires**
- **Edit** > **Highway Wires** > **Remove Wires**
- **Edit** > **Search in diagram**

View Menu Commands

- **View** > **Update Labels**

Insert Menu Commands

- **Insert** > **Wire**
- **Insert** > **Cable**
- **Insert** > **Highway**
- **Insert** > **Rail**

- **Insert** > **Ladder**
- **Insert** > **Connector** > **Single View**
- **Insert** > **Connector** > **Multi-View**
- **Insert** > **Connector** > **Parametric**
- **Insert** > **Connector** > **Inline**
- **Insert** > **Component** > **Single View**
- **Insert** > **Component** > **Multi-View**
- **Insert** > **Component Group**
- **Insert** > **Splice** > **Butt**
- **Insert** > **Splice** > **Through**
- **Insert** > **MVC View**
- **Insert** > **Component Group View**
- **Insert** > **Cable Symbol**

Format Menu Commands

- **Format** > **Spools**
- **Format** > **Default Wire Spool**
- **Format** > **Default Cable Spool**
- **Format** > **Conductor Direction**
- **Format** > **Multi-View Connector Gallery**
- **Format** > **Multi-View Component Gallery**
- **Format** > **Component Group Gallery**
- **Format** > **Rung Labels**
- **Format** > **Reference Zones**

Info Menu Commands

- **Info** > **Wire List**
- **Info** > **Wire List for Layer**
- **Info** > **MVC Instance**
- **Info** > **Component Group**

Tools Menu Commands

- **Tools** > **Parameters**
- **Tools** > **Diagram**
- **Tools** > **Pattern**
- **Tools** > **Objects**
- **Tools** > **Terminator Tables**
- **Tools** > **Logical Reference**

Configuring Pro/DIAGRAM

About Configuring Pro/DIAGRAM

You can preset environment options by setting configuration options and their values in the `config.pro` file.

A list of configuration options arranged in alphabetical order for Pro/CABLING, Pro/DIAGRAM, and Pro/HARNESS is available under the **Electromechanical** category in **Current Session**. 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

From this list, only the following configuration options are available for Pro/DIAGRAM:

- Diagram_pin_edit_multi_line
- Orthogonal_snap
- pro_spool_dir

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

Many of the Drawing configuration options also apply to Pro/DIAGRAM.

To Set Pro/DIAGRAM 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** check box to see all configuration options or select this option to see currently loaded configuration options.

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 Diagram-specific configuration option from the list or type the 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. You can modify this value.

   or

   Type a 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 symbol.

   Note: The **Add/Change** option is enabled only when you try to 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/DIAGRAM, click **Apply** or **OK**.

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

diagram_pin_edit_multi_line

`no`, `yes`

`yes`—You can add user defined pin parameters by including them between the **DEFINE** and **ENDDEF** statements for each pin.

`no`—The system uses a columnar format. Determines the Pro/TABLE format used when you are modifying pin parameters.

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.

general_undo_stack_limit

`50`

Sets the number of undo or redo operations. If the number of operations exceeds 50, the first operation in the stack is removed first, and so on.

orthogonal_snap

yes, no

Controls non-90 degree routing in Pro/DIAGRAM.

yes—Turns the Ortho Snap function on; allowing you to sketch wires only at the default horizontal and vertical orientations.

no—Allows you to sketch wires in drawings at angles other than the default horizontal and vertical orientations.

Logical Referencing

About Logical Referencing

Logical referencing is transferring electrical design data between a diagram and a.corresponding 3D cabling model, so that they can both store and use the same information, and reflect engineering change orders accurately.

Cabling assemblies can directly reference diagram (`.dgm`) files. You can also import a wirelist to a diagram or cabling assembly in Mentor Graphics, PTC Neutral format (`.nwf`), or XML format (`.xml`). You can export data from a diagram in PTC Neutral format or XML format (`.xml`). During the design process, you can compare the diagram and the 3D model object by object, to be sure that data is synchronized for both designs.

To Import or Export a Wire List

1. Click **Tools** > **Logical Reference** > **Import** (or **Export**).
2. If you choose **Import**, select the format from the **WIRELIST IMP** menu. You can import logical information from a Mentor Graphics format wire list file, PTC neutral wire list file or an XML format (`.xml`) file.
3. If you choose **Export**, use the browser to name the export file.

To Compare Logical References

You can compare the logical connections of the diagram with either a cabling assembly or an ASCII wire list. This allows you to see if any logical connection in the diagram is missing or incorrect in the 3D harness.

1. Click **Tools** > **Logical Reference** > **Compare**. The **COMPARE** menu appears with checkboxes for each diagram object on the wirelist. Check each object you want to compare:
   - **Wire and Cable Spools**—A match occurs if they all have the same parameters with the same values. A subset match, indicated by the word `SUBSET` in parentheses in the comparison output file, occurs if all the parameters common to both referenced and design data have the same values. The spools are identified by the spool names.

- **Conns/Comps**—Connectors and components can 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 connection of a wire to a connector or component is compared. A wire is matched if it runs between the same matched connectors and pins, even if the names of the wires are not the same. Wires connected to rails are also listed in the logical reference information as being connected to rails. Wires are identified by the wire name.

Note: Cabling assemblies do not contain pin attachment information until it is explicitly entered when you edit the connector parameters.

If you check **Matched**, matched items are listed first in the listing.

2. Click **Execute**. The comparison list appears in a text reader.

Tip: Layers and Logical Referencing

When you reference a diagram in Pro/CABLING, you can reference the whole diagram file or only specified layers, so it can be convenient to create harness specific layers in Pro/DIAGRAM.

Example: PTC Neutral Wire List

This is an incomplete PTC neutral wire list, meant to show examples of each object type:

```
! Wire and cable spools

NEW WIRE_SPOOL 14BL
PARAMETER COLOR BLUE
PARAMETER MIN_BEND_RADIUS 0.125
PARAMETER THICKNESS 0.06
PARAMETER WIRE_GAUGE 14
PARAMETER UNITS INCH
PARAMETER MASS_UNITS POUND
PARAMETER WIRE_CONSTRUCTION STRANDED

NEW CABLE_SPOOL GRN4C 4
PARAMETER COLOR GREEN
PARAMETER MIN_BEND_RADIUS 0.18
PARAMETER THICKNESS 0.08
PARAMETER MASS_UNITS POUND
PARAMETER UNITS INCH
CONDUCTOR 1
PARAMETER COLOR BLACK
PARAMETER WIRE_GAUGE 8
CONDUCTOR 2
PARAMETER COLOR RED
PARAMETER WIRE_GAUGE 7
```

```
CONDUCTOR 3
PARAMETER COLOR ORANGE
PARAMETER WIRE_GAUGE 8
CONDUCTOR 4
PARAMETER COLOR WHITE
PARAMETER WIRE_GAUGE 7

! Components and connectors

NEW CONNECTOR J1
PARAMETER MODEL_NAME CONN_Y15
PARAMETER NUM_OF_PINS 15
PARAMETER GENDER FEMALE
PIN 1
PARAMETER SIGNAL_NAME X24-1
PARAMETER SIGNAL_VALUE +5V
PARAMETER ENTRY_PORT ENTRY1
PARAMETER GROUPING ROUND
PARAMETER INTERNAL_LEN 0
PIN 2
PARAMETER SIGNAL_NAME X24-1
PARAMETER SIGNAL_VALUE +5V
PARAMETER ENTRY_PORT ENTRY2
PARAMETER GROUPING ROUND
PARAMETER INTERNAL_LEN 0
PIN 3
PARAMETER SIGNAL_NAME X24-1
PARAMETER SIGNAL_VALUE +5V
PARAMETER ENTRY_PORT ENTRY3
PARAMETER GROUPING ROUND
PARAMETER INTERNAL_LEN 0
PIN 4
PARAMETER SIGNAL_NAME X24-1
PARAMETER SIGNAL_VALUE +5V
PARAMETER ENTRY_PORT ENTRY4
PARAMETER GROUPING ROUND
PARAMETER INTERNAL_LEN 0

! Wires and cables

NEW WIRE WIRE_01 14BL
ATTACH P1 1 C1 1

NEW WIRE WIRE_02 14GRY
ATTACH P1 2 C1 2

NEW WIRE WIRE_03 14GRY
ATTACH P2 1 C1 3
```

```
NEW WIRE WIRE_04 16RD
ATTACH J7 8 C2 6

NEW WIRE WIRE_05 16RD
ATTACH J7 7 C2 5

NEW WIRE WIRE_06 16YL
ATTACH J7 6 C2 4

NEW CABLE CABLE_01 GRN4C
ATTACH J1 "" C3 ""
CONDUCTOR 1
ATTACH J1 1 C3 1
PARAMETER NAME WIRE_13
CONDUCTOR 2
ATTACH J1 2 C3 2
PARAMETER NAME WIRE_14
CONDUCTOR 3
ATTACH J1 3 C3 3
PARAMETER NAME WIRE_15
CONDUCTOR 4
ATTACH J1 4 C3 4
PARAMETER NAME WIRE_16
```

Setup Options for Pro/DIAGRAM

To Set Up Pro/DIAGRAM Preferences

To set up the diagram file with options of your choice:

1. Click **File** > **Properties**. The **Options** dialog box opens.
2. Select and edit the required configuration options.

To Set the Grid

1. Click **View** > **Draft Grid**. The **GRID MODIFY** menu appears.
2. Use the commands in the **GRID MODIFY** menu to set the grid display and spacing as follows:

- **Show Grid**—Turns on the display of the grid. This does not affect the snapping of sketched entities to grid intersections. To change the grid snap, use the **Snap to Grid** option in the **ENVIRONMENT** dialog box (**Tools** > **Environment**). This insures that all nodes lie on the drawing grid.

  Note: When routing wires, you can work with the grid on but with grid snap off.

- **Hide Grid**—Turns off the display of the grid.
- **Type**—Establishes the type of grid as Cartesian (default) or Polar. In Pro/DIAGRAM, you use the Cartesian grid.

- **Origin**—Places the grid origin anywhere on the screen and shows coordinate axes at the current grid origin.
- **Grid Params**—Establishes the spacing and angle of the grid. The **Cart Params** menu appears with the following options:
  - **X&Y Spacing**—Sets the spacing in both X- and Y-direction to the same value.
  - **X Spacing**—Sets the X-spacing only.
  - **Y Spacing**—Sets the Y-spacing only.
  - **Angle**—Modifies the angle between the horizontal and the X-direction grid.

Note: Before creating wires it is helpful to:

- Modify the grid parameters and/or the component locations so that all the nodes are located on grid intersections.
- Click **Tools** > **Environment** and check **Snap to Grid** to turn the grid snap on.

If you turn on the grid before creating connectors, it ensures that any pins (nodes) you create are on the grid and are spaced properly.

To Switch Snap to Grid

1. Select **Tools** > **Environment**. The **Environment** dialog box opens.
2. In the **Default Actions** box, check or clear **Snap to Grid** and **Snap to XY Axes** as required.

   Note: If grid snap is on, objects only snap to the grid if the grid spacing is set large enough to be visible. On an A-size drawing, the grid is not visible if the grid spacing is less than 0.7 mm or 0.07 inches.

   The visibility of the drawing also depends on the size of the window and whether you have collapsed the left panel of the window or not.
3. Click **Apply** or **OK** for the changes to affect your current diagram.

To Set Default Conductor Names

Use the Pro/DIAGRAM setup options to customize default names of the wires, cables, rails, and highways.

1. Click **File** > **Properties**. The **Options** dialog box opens.
2. Use the options below for wires as a guideline for all conductor objects; each object (highways and rails for example) uses similar setup options.
   - `wire_default_prefix` defines the prefix of wirenames. Use the characters that are allowed for parameters. The default value is `WIRE`.
   - `wire_default_suffix` defines the first number or letter of the wirename that is incremented as you add a new wire. The value must either be only

numerical or alphabetical and must not contain characters such as underscores or hyphens, or a mix of numerical and alphabetical characters. The default value is 0001.

- wire_default_increment defines the increment between two consecutive wires. The increment value must be numerical. The default value is 1.

Diagram-Specific Drawing Setup Options

These Drawing Setup options apply specifically to Pro/DIAGRAM.

Label Position

label_default_position	Position of labels on wire lines. Can be START, MIDDLE, END or any combination separated by commas.
label_ends_gap	Measures the gap erased from a wire on each side of a label, when labels are inside wires. Real number (floating point).
label_parallel	If YES, the wire label is always parallel to the wire, if NO, labels are always parallel to the orientation of the sheet.

Wires

break_point_size	Sets the size of the wire break symbol. There is a certain limit for the value of break_point_size. If the value is larger or equal to 0.0014 (=**<**0.0014), the cross (wire break symbol) at the wire break point displays even if it is very small. If the value is less than 0.0014 (<0.0014), the cross (wire break symbol) does not display, but the number in the setup file is the one entered by the user.
def_wire_break_label	Sets the default wire break label name. To create leading spaces in default wire-break labels, you must enter two sets of quotation marks before the spaces (such as, " sht&wire_opp_sheet"); otherwise, the system does not accept the syntax as input, and does not create the leading spaces in the label.

`def_wire_label`	Sets the default string to display in the wire label.
`default_wire_label_style`	Sets the default wire label style, which is a user-defined text style.
`default_wire_line_style`	Sets the default wire line style, which is a user-defined text style or a system line style.
`wire_default_increment`	Defines the increment numerical value between two consecutively created wires. Used with wire default-prefix and wire-default suffix.
`wire_default_prefix`	Defines the text prefix of default wire names.
`wire_default_suffix`	Defines the start number or letter of the default wirename. If `wire_default_suffix` is `1`, and `wire_default_increment` is `2`, consecutive wires are named `wire1`, `wire3`, `wire5`, and so on. If `wire_default_suffix` is `A`, and `wire_default_increment` is `2`, consecutive wires are named `wireA`, `wireC`, `wireE`, and so on. (Cannot contain underscores, dashes, or a mix of numerical and alphabetical characters.)
`wire_notes_above`	If `Yes`, the wire notes are set above the wire. If `No`, they intersect the wire. You can also enter a real number (floating point) for exact offset from the wire.

Cables

`cable_cond_delimiter`	If `cable_cond_from_cable` is set to `Yes`, the value of `cable_cond_delimiter` is inserted between the cable and spool conductor name when forming the name of the cable conductor.
`cable_default_increment`	Defines the increment numerical value between two consecutively created cables. Used with `cable_default_prefix` and `cable_default_suffix`.

cable_default_prefix	Defines the text prefix of default cable names.
cable_default_suffix	Defines the start number or letter of the default cable name. If cable_default_suffix is 1, and cable_default_increment is 2, consecutive cables are named cable1, cable3, cable5, and so on. If cable_default_suffix is A, and cable_default_increment is 2, consecutive cables are named cableA, cableC, cableE, and so on. (Cannot contain underscores, dashes, or a mix of numerical and alphabetical characters.)
cable_jacket_report_name	If you use the default value (DEFAULT), the name of the cable shows in the report table. Any other value is interpreted as plain text. For example, &cable_name has no special meaning.
cable_shield_report_name	Use this name for the cable symbol if the cable is shielded. The default value is SHIELD.
cable_node_report_name	Use this name for the nodes of a cable symbol. The default value is "-".
cond_name_from_cable	If Yes, the conductor name is derived by concatenating the cable name, the value of cable_cond_delimiter, and the spool conductor name.
def_cable_label	Defines the default cable note contents that appears when a cable symbol is created.
def_cond_label	Sets the default cable conductor label name.

Rails

def_rail_label	Sets the default contents of the rail label.
default_rail_label_style	Sets the default rail label style, which can be any valid named text style that is user-defined.

`default_rail_line_style`	Sets the default rail line style, which is a user-defined text style or a system line style.
`rail_default_increment`	Defines the increment numerical value between two consecutively created rails. Used with `rail_default_prefix` and `rail_default_suffix`.
`rail_default_prefix`	Defines the text prefix of default rail names.
`rail_default_suffix`	Defines the start number or letter of the default railname. If `rail_default_suffix` is `1`, and `rail_default_increment` is `2`, consecutive rails are named `rail1, rail3, rail5`, and so on. If `rail_default_suffix` is `A`, and `rail_default_increment` is `2`, consecutive rails are named `railA, railC, railE,` and so on. (Cannot contain underscores, dashes, or a mix of numerical and alphabetical characters.)
`rail_node_radius`	Sets the size of the nodes displayed at attachment points between two rails or between a wire and a rail for printing and display.
`rail_notes_above`	Places the rail notes at a specified real number (floating point) above the highway. The default is `.01`. Enter `NO` to center the label on the rail.

Highways

`def_highway_label`	Defines the highway label contents. This sets the highway label name. The default is no label. Enter `&hway_name` as a value to show the name; with the assigned prefix, suffix, and increment.
`default_highway_label_style`	Sets the default label style, which can be any valid named text style that is user-defined.
`default_highway_line_style`	Sets the default highway line style, which is a user-defined text style or a system line style.

highway_default_increment	Defines the increment numerical value between two consecutively created highways. Used with highway_default_prefix and highway_default_suffix.
highway_default_prefix	Defines the text prefix of default highway names.
highway_default_suffix	Defines the start number or letter of the default highway name. If highway_default_suffix is 1, and highway_default_increment is 2, consecutive highways are named highway1, highway3, highway5, and so on. If highway_default_suffix is A, and highway_default_increment is 2, consecutive highways are named highwayA, highwayC, highwayE, and so on. (Cannot contain underscores, dashes, or a mix of numerical and alphabetical characters.)
highway_notes_above	Places the highway notes at a specified real number (floating point) above the highway. The default is .01. Enter NO to center the label on the highway.

Connectors

def_para_conn_text_height	Sets the default parametric connector text height.
node_radius	Sets the radius of the nodes. By default, the node radius value is DEFAULT. You can change this value as required.
para_conn_mirror	Allow or disallow mirroring for PCs or associated text. (Yes, No, geom_only, text_only)
pc_ident_label	Sets content of parametric connector label.
pc_node_label	Sets the content of the node label for parametric connectors. The value of this option can be text, a pin parameter, or any combination, including a user-defined pin parameter.

Setup Options: Default Report Names

The setup options determine the report name of the default cable symbol for shielded and unshielded cable symbols and the default name for nodes on cable symbols. These options apply to all cable symbols in a diagram.

Parameter or Option	Description
`cable_jacket_report_name`	Use the default value `DEFAULT`, in which case the name of the cable shows in the report table. Any other value is interpreted as plain text. For example, `&cable_name` has no special meaning.
`cable_shield_report_name`	Use this name for the cable symbol if the cable is shielded. The default value is `SHIELD`.
`cable_node_report_name`	Use this name for the nodes of a cable symbol. The default value is "-".
`cable_shield_report_name` `cable_node_report_name`	Use the same names for cables of the same type, for example, a coaxial cable or a two wire, twisted-pair cable.

`from_cable_report_name`
`to_cable_report_name`
`from_to_cable_report_name`
	Use these to represent mechanical differences that can be important for manufacturing. Override cable spool parameters and Drawing setup options.

Defining Symbols in Pro/DIAGRAM

About Defining Pro/DIAGRAM Symbols

Components and fixed (non-parametric) connectors in Pro/DIAGRAM are represented by drawn symbols. Symbols can be defined and saved in both Drawing and in Diagram mode. When you place a symbol in the drawing, you place an instance of the symbol based on a definition that has already been created. There can be many symbol instances referencing the same symbol definition, each with a different reference designator to distinguish between them.

You can select cable symbols or break symbols using the Diagram Item filter and the drawing or custom symbols using the Symbol filter.

A connector or component symbol in Pro/DIAGRAM must have a certain set of parameters. Two parameters types can be included:

- Symbol definition parameters to identify the symbol.
- Node parameters identifying nodes. (If nodes are necessary in the symbol.)

Symbols may be assigned to any component or connector parameters.

You can modify symbolic representations of components and fixed connectors (such as the shape of the symbol, nodes, and the location of notes) only by redefining the reference symbol definition. If the symbols are defined as Variable Drawing Units in the symbol definition, then you do not need to redefine the symbol to change its size.

If you redefine a symbol, all component/connector instances in the diagram that reference the same symbol also change.

Note: You cannot redefine a Parametric connector.

Symbol Instances

When you add a symbol instance, you specify:

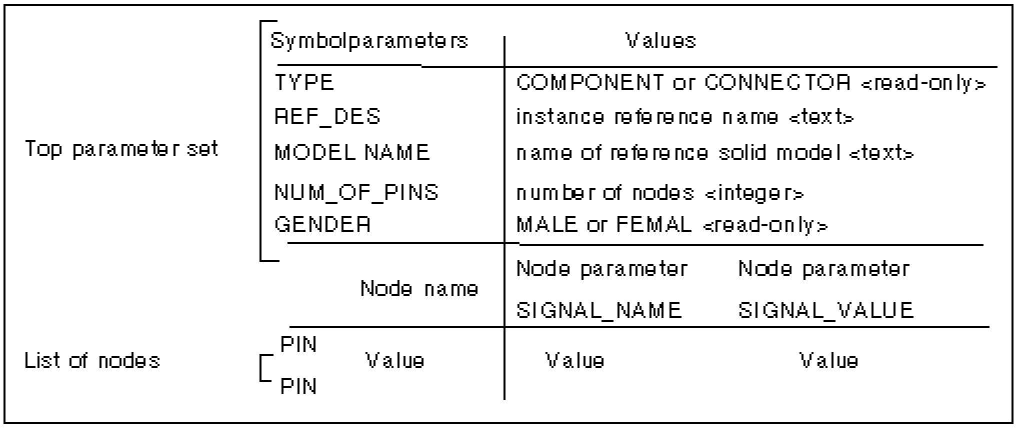
- The symbol definition to be used
- Where to place it in the diagram
- How large to make the symbol
- Its parameters (diagram objects only)

To add symbol instances you can:

- Place them as you would in a drawing.
- Place them as a diagram object such as a single-view component, a single-view connector, or a parametric connector. In the case of the parametric connector, the definition is actually a part of the instance.
- Place them as part of a multiple view component or a multiple view connector.

If an application uses a symbol with required parameters, but they are missing, the system issues an error message and you can edit the symbol parameter file. The next figure shows the format of the parameter file.

Parameter Set Format



Pro/ENGINEER uses symbols containing an individual parameter set to create electrical diagrams. You can define Pro/DIAGRAM symbols in Drawing mode and Diagram mode. However, if you want to use them in Pro/DIAGRAM, you must define them as components or connectors by providing the parameter set appropriate for the type of object the symbol represents (component or connector). The following tables list component and connector parameters (the required parameters are shown in bold type).

To Define a Drawing Symbol for a Component or Connector

1. Click **Format** > **Symbol Gallery**. The **DEFINITION** menu appears.
2. Click **Define**.
3. At the prompt, type a new symbol name and press ENTER. The (**SYM_EDIT_<NAME>**) drawing window opens and the **SYMBOL EDIT** menu appears.
4. Click **Sketch** in the drawing window to access the Pro/ENGINEER drawing tools to create the shape for the connector.
5. When you have created the symbol shape, click **Insert** > **Node**. The **SYMBOL NODE** menu appears.
6. Click **Make Node**. You are prompted to specify a node name.
7. Type a name or number for the node at the prompt in the drawing window and press ENTER. You are prompted to name the next node. Name the node or middle-click to end the command. Node points are added as small circles. The **Symbol Definition Attributes** dialog box opens.

   Use the **General** tab to set origin points for the symbol and leaders you may attach to it. Use the **Symbol Instance Height** options to determine whether the symbol can be scaled or should remain of fixed dimensions. Use the **Attributes** check boxes to allow angle or mirror attributes.

8. When you are finished with the **Symbol Attributes** dialog box, click **OK**.

9. To add connector or component parameters to the symbol definition, click **Symbol Edit** > **Parameters** > **Read**. The **READ SYM PRM** menu appears. Use any of the following commands to read in parameters of the diagram objects:
   - **Comp Default**—Allows to read in default parameter set for a component.
   - **Conn Default**—Allows to read in default parameter set for a connector.
   - **Other**—Opens a browser to read in a previously-saved parameter file.
10. Click **Done** in the **SYMBOL EDIT** menu. The symbol is created.

Note: You cannot use this procedure to add a symbol instance to the diagram. Use **Insert** > **Drawing Symbol** > **Custom** to add instances.

To Add a Drawing Symbol from a Standard Palette to the Diagram

1. Click **Insert** > **Drawing Symbol** > **From Palette**. The **Symbol Instance Palette** dialog box opens.
2. Select the required symbol from the palette. The **SELECT** dialog box opens.
3. Select an edge, entity, dimension, dimension witness line, coordinate system, curve, or a symbol entity to insert the symbol and click **OK**. The symbol is inserted at the selected location.
4. Click **Close** in the **Symbol Instance Palette** dialog box.

To Redefine a Symbol

1. Click **Format** > **Symbol Gallery**. The **DEFINITION** menu appears.
2. Click **Redefine**. The **GET SYMBOL** menu appears. Use the Diagram Item or Symbol filters to select the required symbol.
3. Select an instance of the symbol to edit or click **Name** to select a predefined symbol from the **Select Symbol Definitions** dialog box and click **OK**. The symbol is placed in the drawing.
4. Use the **SYMBOL EDIT** menu to make the edits.

   If you edit the drawn symbol, all instances are updated. If you edit symbol parameters, you can check the **Propagate** option to globally update the symbol instances in the design.
5. Click **Done/Return**, when you finish.

To Modify Symbol Size

1. Select a symbol to modify. You can use the Diagram Item or Symbol filters to select the required symbol.
2. Click **Edit** > **Properties**. The **EDIT CONN** and **CONN VIEW** menus appear.
3. Click **Resize** from the **CONN VIEW** menu.

4. Enter a value, for instance, height. Pro/DIAGRAM displays the current value as the default and rescales the symbol.

Note: If the symbols are defined as Fixed, you cannot modify them.

To Modify a Drawing Symbol

1. Select a drawing symbol to modify. You can use the Symbol filter to select the required drawing symbol.
2. Click **Edit** > **Properties** or double-click on the symbol. The **Custom Drawing Symbol** dialog box opens.
3. Under **Placement**, you can change the placement type.
4. Under **Properties**, specify the height, angle, or color of the symbol instance according to the specifications.
5. Under **Origin**, specify the origin as defined in the symbol definition or specify a custom origin for the symbol instance.
6. Click **OK**.

Note: If the symbols are defined as Fixed, you cannot modify them.

Using Parameters in Pro/DIAGRAM

About Parameters in Pro/DIAGRAM

Diagram reference objects (components and connectors) and connections (wires, cables, and rails) have parameters associated with them. For connections, some parameters are passed from the spool that the wire or cable is added from, and other parameters are specified for the individual wire or cable after it is added to the diagram. Every entity has the following parameter types:

- **Required**—The conductor or component must have values defined for these parameters.
- **Optional**—These are not required to define the object, but are included as commonly used informational fields, for example, `COLOR_CODE` (`Wht, Blk, Rd`, and so on.). If the value is numeric, the parameter may be used in manufacturing design calculations, for example, `MINIMUM_BEND_RADIUS` for a cable.
- **Optional user-defined**—You can define a new parameter and value for informational output as required, for example, `SUPPLIER_NAME`.

Double clicking on a wire, rail, cable conductor, cable symbol, component view, or a connector view for a MVC, Parametric, inline, or a component group opens the **Electrical Parameters** dialog box. Use the **Electrical Parameters** dialog box to add or edit parameters for the selected diagram objects. You can edit a single parameter or make global edits. For example, you can assign a common value to all parameters of the same name, across selected objects.

Components vs. Connectors

A component is a symbol with the `OBJ_TYPE = COMPONENT` parameter associated with it. Components use reference designators as unique identifiers and include properties for pins.

Connectors are a subtype of components with the `OBJ_TYPE` value of `CONNECTOR`. A connector can use two additional parameters:

- `GENDER` (`MALE` or `FEMALE`)—At the component level

  The gender is a design reference for the intended solid model and is required for connectors.

- `ENTRY_PORT`—At the pin level

  The entry port is the name of a coordinate system on the solid model that specifies the beginning or end of a connection.

These parameters are specifically used to pass information to Pro/CABLING. In Pro/CABLING, the `ENTRY_PORT` parameter designates pin to pin wire and cable connections for manual cabling or autorouting.

To Add or Delete Parameters on the List

1. Click **Tools** > **Parameters** > **Objects**. The **Select By Type** and **SELECT** dialog boxes open simultaneously.
2. In the **Object Type** box, select the appropriate object type.
3. Select the wires, cables, components, or connectors to modify.
4. Click **OK** in the **SELECT** dialog box.
5. When you set all the required options, click **Modify** in the **Select By Type** dialog box.

   or

   Select the wires, cables, components, or connectors to modify and right-click to select **Parameters** from the shortcut menu.

   or

   Double-click the wires, cables, components, or connectors that you want to modify.

   The **Electrical Parameters** dialog box opens.
6. Set the appropriate **Display For** option to show or hide the display of parameters for different levels of the tree. For example, for a component, pin, conductor, entry port, and so on.
7. Click **View** > **Columns**. The **Model Tree Columns** dialog box opens.
8. Use the **Model Tree Columns** dialog box to move parameters back and forth from the **Not Displayed** to the **Displayed** windows as necessary.

To define a new parameter, type it in the **Name** box and click **>>** to move it to the **Displayed** window. The new parameter is added to the current list with `<Nonexistent>` as the value. Nonexistent means no value has been assigned to the parameter. You can change the values individually or globally.

9. Click **Apply** and then **OK**.
10. Add the new parameter to any object in the **Electrical Parameters** dialog box.
11. Click **Apply** and then **OK** in the **Electrical Parameters** dialog box
12. Click **Close** in the **Select By Type** dialog box.

To Edit Electrical Parameters

1. Click **Tools** > **Parameters** > **Objects**. The **Select By Type** and **SELECT** dialog boxes open simultaneously.
2. In the **Object Type** box, select the appropriate object type.
3. Select the objects in the diagram to modify.
4. Click **OK** in the **SELECT** dialog box.
5. Click **Modify** in the **Select By Type** dialog box.

   or

   Select the wires, cables, components, or connectors to modify and right-click to select **Parameters** from the shortcut menu.

   or

   Double-click the wires, cables, components, or connectors that you want to modify. You can also select multiple wires, cables, components, or connectors that you want to modify and double-click.

   The **Electrical Parameters** dialog box opens.

   The left pane of the dialog box contains a tree representing the selected objects and any associated subobjects, for example, connectors and their associated pins. The right pane contains columns for the parameters associated with each object.

   Depending on the object that you have selected, use the **Display For** box to show or hide the display of parameters for different levels of the tree.
6. Click **Apply** and then **OK** in the **Electrical Parameters** dialog box.
7. Click **Close** in the **Select By Type** dialog box.

To Specify Parameter Values Individually

1. Click **Tools** > **Parameters** > **Objects**. The **Select By Type** and **SELECT** dialog boxes open simultaneously.

2. In the **Object Type** box, select the appropriate object type.
3. Select the wires, cables, components, or connectors to modify.
4. Click **OK** in the **SELECT** dialog box.
5. When all the options are set, click **Modify** in the **Select By Type** dialog box.

   or

   Select the wires, cables, components, or connectors to modify and right-click to select **Parameters** from the shortcut menu.

   or

   Double-click the wires, cables, components, or connectors that you want to modify.

   The **Electrical Parameters** dialog box opens.
6. Set the appropriate **Display For** option to show or hide the parameters display for different levels of the tree. For example, for a component, pin, conductor, entry port, and so on.
7. Select the parameter you want to edit in the right panel of the dialog box.
8. 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.

To Specify Parameter Values Globally

1. Click **Tools** > **Parameters** > **Objects**. The **Select By Type** and **SELECT** dialog boxes open simultaneously.
2. In the **Object Type** box, select the appropriate object type.
3. Select the wires, cables, components, or connectors to modify.
4. Click **OK** in the **SELECT** dialog box.
5. When all the options are set, click **Modify** in the **Select By Type** dialog box.

   or

   Select the wires, cables, components, or connectors to modify and right-click to select **Parameters** from the shortcut menu.

   or

   Double-click the wires, cables, components, or connectors that you want to modify.

   The **Electrical Parameters** dialog box opens.

6. Set the appropriate **Display For** option to show or hide the parameters display for different levels of the tree. For example, for a component, pin, conductor, entry port, and so on.
7. In the right panel of the dialog box, select **<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 to a new value.
8. 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.

To Change a Value for Pin Names across Multiple Connectors

1. Click **Tools** > **Parameters** > **Objects**. The **Select By Type** and **SELECT** dialog boxes open simultaneously.
2. In the **Object Type** box, select the appropriate object type.
3. Select the components or connectors to modify.
4. Click **OK** in the **SELECT** dialog box.
5. When all the options are set, click **Modify** in the **Select By Type** dialog box.

   or

   Select the wires, cables, components, or connectors to modify and right-click to select **Parameters** from the shortcut menu.

   or

   Double-click the wires, cables, components, or connectors that you want to modify.

   The **Electrical Parameters** dialog box opens.
6. Click **Pins** under **Display For** in the **Electrical Parameters** dialog box.
7. Expand the **Common Pins** tree.
8. In the right panel of the **Electrical Parameters** dialog box, select the specific pin name line and the parameter value you want to edit.

   Note: Values for most columns are `<As Is>`. This means that values differ for the same pin name across connectors.
9. 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.

Note: Do not click **Connections** or **Spools**.

To Change a Single Value for Parameters across Multiple Connectors

1. Click **Tools** > **Parameters** > **Objects**. The **Select By Type** and **SELECT** dialog boxes open simultaneously.
2. In the **Object Type** box, select the appropriate object type.
3. Select the components or connectors to modify.
4. Click **OK** in the **SELECT** dialog box.
5. When all the options are set, click **Modify** in the **Select By Type** dialog box.

   or

   Select the components or connectors to modify and right-click to select **Parameters** from the shortcut menu.

   or

   Double-click the components or connectors that you want to modify.

   The **Electrical Parameters** dialog box opens.

   When several connectors are selected in the **Electrical Parameters** dialog box, you can use the **Common Pins** tree to change values for a selected parameter across all listed connectors.
6. Click **Pins** under **Display For** in the **Electrical Parameters** dialog box.
7. Expand the **Common** tree.
8. 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.

   Note: Values for most columns will read as `<As Is>`. This means that values differ for the same pin name across connectors.
9. At the prompt, type the new value and press ENTER. The values for the selected parameters are updated across all listed connectors.

To Create a Parameter File from Object Parameters

1. Click **Tools** > **Parameters** > **Objects**. The **Select By Type** and **SELECT** dialog boxes open simultaneously.
2. Select the object or objects for which you want to create files and click **OK** in the **SELECT** dialog box.
3. Click **Modify** in the **Select By Type** dialog box.

   or

   Select the wires, cables, components, or connectors to modify and right-click to select **Parameters** from the shortcut menu.

or

Double-click the wires, cables, components, or connectors that you want to modify.

The **Electrical Parameters** dialog box opens.

4. Highlight an object in the object tree.
5. Click **File** > **Write** in the **Electrical Parameters** dialog box.
6. Enter a filename for the new parameter file to write the parameters of the selected diagram object to a file.

To Read a Parameter File Into a Connector or Component

1. Click **Tools** > **Parameters** > **Objects**. The **Select By Type** and **SELECT** dialog boxes open simultaneously.
2. Select the object or objects to receive the parameters and click **OK** in the **Select** menu.
3. Click **Modify** in the **Select By Type** dialog box.

   or

   Select the components or connectors to modify and right-click to select **Parameters** from the shortcut menu.

   or

   Double-click the components or connectors that you want to modify.

   The **Electrical Parameters** dialog box opens.
4. Select the first target object in the tree.
5. Click **File** > **Read** in the **Electrical Parameters** dialog box.
6. Use the **Open** dialog box to select the target parameter file to import.
7. Click **Apply** and then **OK** in the **Electrical Parameters** dialog box.
8. Click **Close** in the **Select By Type** dialog box.

Component Parameters

`OBJECT_TYPE` (Required)	The object type (`COMPONENT` OR `CONNECTOR`). Format: `OBJ_TYPE` text string
`REF_DES` (Required)	The reference designator name. When placing a component symbol in a diagram, you must supply a unique `REF_DES` name. The system saves this name with the symbol instance and uses it in wire lists, and the like.

	Format: REF_DES text string
NUM_OF_PINS (Required)	The number of logical pins. The number of visible pins is determined by number of nodes in the symbol definition. Format: NUM_OF_PINS integer
GENDER	Required If object type is connector (Male or Female.).
MODEL_NAME	The name of the reference solid model in Pro/CABLING—the physical model that the component symbol represents. This helps Pro/CABLING to designate components automatically. Format: MODEL_NAME text string
DESCRIPTION	The description of the component. Format: DESCRIPTION text string.
DEF_GROUPING	The default grouping for entry ports in the connector, ROUND, WIRE or FLAT. Default value ROUND
DEF_INTERNAL_LEN	The default internal length value for all entry ports in connector, default value 0.0
TABLE_AUTO_ASSIGN	Optional, True, False, defaults to True. You can also set this parameter at the component level so that the parameter is applicable to all the pins of the component. Example: If you set the SIGNAL_VALUE_AUTO_ASSIGN parameter to FALSE for a terminator table, you cannot set the corresponding parameter for that pin.
User Defined	Establishes a user-defined parameter. Format: <parameter name> text string or value Example: SUPPLIER KGY_Supply_Co

Connector Parameters

REF_DES (Required)	The reference designator name. When placing a connector or component symbol in a diagram, you must supply a unique REF_DES name. The system

	saves this name with the symbol instance and uses it in wire lists, and the like. Format: `REF_DES` text string
`MODEL_NAME`	The name of the reference solid model in Pro/CABLING—the physical model that the component symbol represents. This helps Pro/CABLING to designate components or connectors automatically. Format: `MODEL_NAME` text string
`DEF_GROUPING`	The default grouping for entry ports in the connector, `ROUND`, `WIRE` or `FLAT`. Default value `ROUND`
`DEF_INTERNAL_LEN`	The default internal length value for all entry ports in connector, default value `0.0`
`GENDER`	Required if object type is connector (`Male` or `Female`.).
`NUM_OF_PINS` (Required)	The number of logical pins. The number of visible pins is determined by number of nodes in the symbol definition. Format: `NUM_OF_PINS` integer
`OBJECT_TYPE` (Required)	The object type (`COMPONENT` OR `CONNECTOR`). Format: `OBJ_TYPE` text string
`TABLE_AUTO_ASSIGN`	Optional, `True`, `False`, defaults to `True`. You can also set this parameter at the component level so that the parameter is applicable to all the pins of the component. Example: If you set the `SIGNAL_VALUE_AUTO_ASSIGN` parameter to `FALSE` for a terminator table, you cannot set the corresponding parameter for that pin.

Component Pin Parameters

`SIGNAL_NAME`	Signal name for the pin. Optional and can be entered in the ASCII file.
`SIGNAL_VALUE`	Signal value for the pin. Optional and can be entered in the ASCII file.
`ENTRY_PORT`	If the value is the name of a coordinate system

	on the designated part in 3D, Pro/CABLING can identify the coordinate system as the correct entry port when using Logical Ref. If you enter this parameter in Pro/DIAGRAM, it is automatically included in the 3D assembly connector parameters when using Logical Ref/Update for the connector. (This parameter can also be entered into the 3D connector parameters manually.) This pin-to-coordinate system association can be used when routing cable conductors, to highlight the target entry port for each selected conductor as you route it. Required only for cable autorouting if you use more than one entry port coordinate system per connector.
`TERM_NAME`	Specifies the terminator name associated with a particular pin. Format: `TERM_NAME string`
`TERM_AUTO_ASSIGN`	Specifies the automatic assignment of terminator information from a table. The value `TRUE` (default) or `FALSE` determines if the terminator information is automatically overwritten by executing a terminator table. Example: `TERM_NAME 6 TERM6327Y FALSE`
`TABLE_AUTO_ASSIGN`	Optional, `True,False`, defaults to `True`. You can also set this parameter at the component level so that the parameter is applicable to all the pins of the component. Example: If you set the `SIGNAL_VALUE_AUTO_ASSIGN` parameter to `FALSE` for a terminator table, you cannot set the corresponding parameter for that pin.
`SHIELD`	
`PIN_PLUG`	
`INTERNAL_LEN`	The internal length for specific pin, overrides `def_internal_len`.
`GROUPING`	The grouping for specific pin, overrides `def_grouping`. (`ENTRY_PORT` is required to use this parameter.)
User Defined	Establishes a user-defined parameter. Format

	<parameter name> text string or value

Wire Spool Parameters

The following is a list of spool parameters for wires. Parameters that have no relation to a diagram, but that pass values which are used in Pro/CABLING are marked.

Note: For text values, you cannot use spaces in text strings.

`NAME`	The name of the spool file (text_string). Format: `NAME` text_string
`TYPE`	The type of spool used. Wires are of type `WIRE`. Format: `TYPE` text_string
`COLOR`	The color of the wire. `UNDEFINED` is the default. Format: `COLOR` text_string
`COLOR_CODE`	The color code of the wire (text_string. Format: `COLOR_CODE` text_string
`DENSITY`	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.
`INSUL_TYPE`	Insulation type (text_string).
`MASS_UNITS` (Pro/CABLING)	The units of mass for a wire. Used to determine mass properties of 3D wires that reference the diagram. Format: `MASS_UNITS` text_string Example: `MASS_UNITS KG`
`MIN_BEND_RADIUS` (Pro/CABLING)	The minimum bend radius allowed for the wire in harness part length units. Example: `MIN_BEND_RADIUS 0.3`
`THICKNESS` (Pro/CABLING)	The diameter of the wire. Example: `THICKNESS 0.1`
`UNITS` (Pro/CABLING)	Specifies the units used for measurement of wires. Used by Pro/CABLING when referencing a diagram, in conjunction with the `DENSITY` parameter, to determine the mass of a wire. The default is the units used in a Pro/CABLING assembly. Example: `UNITS inch`

WIRE_CONSTRUCTION	Wire construction. (text_string) Example: WIRE_CONSTRUCTION SINGLE_STRAND
WIRE_GAUGE	Wire gauge (text_string). Example: WIRE_GAUGE 18AWG
User Defined	Establishes a user-defined parameter, which can be accessed like other spool parameters. Format: <parameter_name> <value> Example: MANUFACTURER X_Corp

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.

NAME	The name of the spool file. Format: NAME text_string
TYPE (read only)	Determines a cable spool or wire spool. Cables are of type PREFAB.
NUM_CONDUCTORS	The number of conductors present in a cable. The default value is zero.
DENSITY	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.
INSUL_TYPE	Insulation type. (Text string) NONE is the default.
SHIELD_TYPE (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 DEFAULT, in which case the name of the cable shows in the report table. Any other value is interpreted as plain

	text. For example, &cable_name has no special meaning.
CABLE_SHIELD_REPORT_NAME	Use this name for the cable symbol if the cable is shielded. The default value is SHIELD.
CABLE_NODE_REPORT_NAME	Use this name for the nodes of a cable symbol. The default value is "-".
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.

Individual Wire and Cable Parameters

The following table lists the available parameters for individual wires and cables. These parameters are established when the wire or cable is created, but are not passed from the spool.

NAME	Indicates the name of the wire or cable. This value is read-only in the parameter field and is created automatically upon creation of a wire or cable. Click **Edit** > **Value** to change the name of the wire or cable. The value of this parameter updates automatically in the file. Format: NAME read-only_string Example: NAME CABLE0001
SPOOL	Indicates the spool used for a wire or cable. This value is read-only and is created automatically upon creation of a wire or cable. Select the wire or cable and click **Edit**

	> **Properties** to change the spool of the wire or cable. The value of this parameter updates automatically. Format: SPOOL read-only_string.
MAX_ALLOWED_LENGTH (Pro/CABLING)	Sets the maximum allowed length of a wire or cable. Format: MAX_ALLOWED_LENGTH value Example: MAX_ALLOWED_LENGTH 12
MIN_ALLOWED_DISTANCE (Pro/CABLING)	Routes the wire offset from all others at a location by the specified distance. This physically affects the display of the wires. Format: MIN_ALLOWED_DISTANCE value Example: MIN_ALLOWED_DISTANCE 2.5
TARGET_LENGTH (Pro/CABLING)	Sets the ideal length for a wire or cable. Format: TARGET_LENGTH value Example: TARGET_LENGTH 6.2
User Defined	Establishes a user-defined parameter that can be accessed like other wire or cable parameters. Value may be <parameter_name> or text_string. Value: <parameter_name> text_string Example: MANUFACTURER X_Corp.

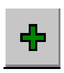
Note:

- Only required parameters are applied to an individual wire or cable when you create it.
- If a wire is separated from a cable, it does not inherit any parameters from the cable. This wire is treated as a newly created wire and has only the default required parameters, NAME and SPOOL.
- Wire and cable parameters can be used to create labels and notes by adding the prefix cbl_ for a cable or wire_ for a wire to the wire or cable parameter.

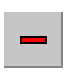
Using Model Parameters

To Add Diagram Parameters

1. Click **Tools** > **Parameters** > **Diagram**. The **Parameters** dialog box opens.
2. Click  or click **Parameters** > **Add Parameter** to create a new parameter.
3. In the **Parameters** dialog box, type the **Name** and **Value**.
4. Specify the **Type** by selecting one of the following:
    - **Integer**—Adds a parameter in the form of an integer.
    - **Real Number**—Adds a parameter in the form of a real number.
    - **String**—Adds a parameter in the form of a string.
    - **Yes No**—Adds a parameter with a value of `yes` or `no`.

    Note: The **Type** cannot be modified after being assigned a value.
5. Click **Designate** if you want to designate the parameter.
6. Click  or click **Parameters** > **Add Parameter** to add another parameter.
7. Click **OK**.

To Modify or Delete Existing Diagram Parameters

1. Click **Tools** > **Parameters** > **Diagram**. The **Parameters** dialog box opens.
2. Select an existing parameter that you want to delete.
3. Click  or **Tools** > **Parameters** > **Delete Parameter**.
4. Click **OK**.

To Show All Current Diagram Parameters and Values

1. Click **Tools** > **Parameters** > **Diagram**. The **Parameters** dialog box opens.
2. Click **Show** > **Info** in the **Parameters** dialog box. An INFORMATION WINDOW opens that lists all the parameters and their values.

Using Notes in Pro/DIAGRAM

About Notes in Pro/DIAGRAM

You can add text or parametric drawing notes to the diagram.

To Create Parametric Notes

1. Click **Insert** > **Note**. The **NOTE TYPES** menu appears.
2. Click **On Item**.
3. Click **Make Note**.
4. Select the wire to which you want to attach the note using the graphical selection methods, or the Diagram Item or Wire filter. The wire is highlighted.
5. At the prompt, type the note contents and press ENTER.
6. Click **Done/Return** to exit the note commands. The note is entered on the wire.

Note: When a note is created **On Item** and attached to a wire segment, it can include any of the parameters valid for wire labels.

Showing Diagram Parameters in Notes

You can use **Insert** > **Note** > **On Item** to display parameter values for selected diagram entities.

For example, to display the color of a cable conductor as defined in the cable spool, use the `&cond_<parameter>` form as `&cond_color`. If conductor2 has the parameter COLOR = BLACK, a note attached to conductor2 with the parameter `&cond_color` displays `BLACK` in the note.

If you update the conductor parameter value for that individual cable, the note is updated. If there is no value for that cable conductor parameter in the individual cable, Pro/DIAGRAM uses the conductor parameter value in the cable spool.

For example, if you use the note contents string:

```
&cbl_name,COND&cond_name,&color
```

Bold shows the values the `&` variables will supply, not bold shows text you included in the label. So, your cable label displays:

```
CABLE02,COND1,WHITE
```

Showing Diagram Parameters for Cables and Cable Conductors

To show diagram parameters, attach the following parametric note parameters to wires in a diagram cable:

- `&spool_name`—Shows the name of the parent spool.
- `&spool_<parameter>`—Shows the value of a top-level spool parameter, for example, Color.
- `&cbl_name`—Shows the cable name to which the wire belongs.
- `&cbl_<parameter>`—Shows a specific cable parameter, for example target length, maximum allowed length, user-defined, and so on.

- &cond_name—Shows the name of the spool conductor that has been assigned to that wire.
- &cond_<spool parameter>—Shows a specific spool conductor parameter for that wire's assigned conductor, for example, wire construction, color, user-defined, and so on.

Note: You can set Pro/DIAGRAM to show the cable name and conductor number for a wire assigned to a cable by default. If you set the drawing configuration option cond_name_from_cable to YES, the wire label shows the cable name and conductor name instead of the wire name for any wires added to a cable and assigned a conductor number.

For Cable Symbols

You can enter &cbl_name, &cbl_<parameter>, &spool_name, and &spool_<parameter name> for note to display the cable name, spool name, or any cable or spool parameter.

Showing Diagram Parameters for Wire Breakpoints

Parametric wire notes can be attached to wiring breakpoints. Use the configuration option def_wire_break_label to create a default wire breakpoint note. When a variable is entered for this option, all subsequent breakpoints are automatically created with a note that contains the defined variable.

In addition to all existing wiring parameter symbols, such as &wire_name, the following parametric symbols can be included in the note:

- &wire_opp_sheet—Displays the sheet number where the wire routing resumes.
- &wire_opp_sym—Displays the reference designator of the symbol where the wire terminates.
- &wire_opp_node—Displays the terminating node name.
- &wire_opp_item—Displays the reference designator of terminating symbol or name of terminating rail.

Defining Label Contents

Use the setup option pc_ident_label to define a pattern of parameters used as contents of a label for the note on a parametric connector.

For example, if you set the value of pc_ident_label in the setup file to &first-&second-&third, add the parameters first, second, and third to the parametric connector with the values of Parametrics, Designer, and Refdes, respectively, the label for the parametric connector becomes Parametrics - Designer - Refdes.

To Set Up Label Contents by Pattern

1. Click **Tools** > **Parameters** > **Pattern**. The **PARAM PATTERN** menu appears.

2. Click **Add** to specify a target parameter name that contains pattern information for all the specified objects. The **MOD PAT TYPE** menu appears.
3. Click **Wires/Cables** to modify parameters of wires and cables or **Ref Objects** to modify parameters of components and connectors. You are prompted to specify the name of the target parameter that you want to set.
4. Type a target parameter name to be set that contains pattern information for all the specified objects and press ENTER. You are prompted to specify the name of the parameter that contains the pattern.
5. Type a source parameter name that contains the pattern, to override the default pattern, if needed. You are prompted to specify a default pattern.
6. Type a default pattern, for example, `&<param_name1>-&<param_name2>`.

   Note: The separator can be any valid character.
7. Click **Execute All**. The system loops through all the specified objects, evaluates the patterns, and sets the target parameter to the value obtained from the pattern.

Use this procedure for both the `ref_des` pattern and for the wire name or rail name pattern.

Reference Zones and Parametric Notes

Reference zones are designated areas of a diagram drawing in which views of multiple-view components (MVC) can be cross-referenced using parametric notes. These notes are used primarily in the construction of ladder diagrams.

A reference zone consists of either horizontal or vertical division lines, each of which is assigned an index or line number. You first determine the physical size of the reference zone and then specify the starting index and delta.

Using MVC Cross References in a Note

Cross-reference notes are used with multiple view components (MVC) and connectors. They identify the physical position of the views by showing the index of the division line they lie closest to within the reference zone. Cross-reference notes can be evaluated to identify the physical position of all views of an MVC or just those views with a specified view name.

Enter the following into the body of the note:

```
&zone_index:<ref_des>:<view_name>
```

where `<ref_des>` is the reference designator of the MVC and `<view_name>` is the name of the view whose position is to be evaluated. The value of this note is the index of the division line in the reference zone closest to the view. If the view is not in a reference zone, an empty value is returned. If the view is later deleted, the value `***` is returned.

If `<view_name>` is omitted, that is, `&zone_index:<ref_des>` is used, then the position of all views of the MVC are evaluated.

Cross-reference notes can be used in the rung labels of a ladder diagram to report on the position of other views of MVCs. In this case, it is recommended that `&zone_index` is used, without the `ref_des` or `view_name` fields.

You can omit the view name from the cross-reference note if the note is attached to the required view. If the view name is omitted and the note is not attached to any symbol, then the cross-reference note returns the index of the division line for all placed views with each index value separated by a space.

Tip: Notes Attached to Nodes

Notes attached to diagram symbol nodes can refer to node parameters (such as, `&signal_name`) and to the node name (`&node_name`). Other symbol notes can refer to top symbol parameters (such as `&ref_des`). The types of text that can go in a node note vary.

Manipulating Diagram Objects

About Cutting and Pasting Diagram Objects

Select an object that you want to cut using the Diagram Item filter or an appropriate filter, or select the object graphically using any selection method, and use **Edit** > **Cut**.

If you select a conductor to cut, the conductor and the terminal components on either end of the conductor are highlighted to identify the pins that are to be disconnected. When you execute the cut, only the selected conductors are removed from the drawing.

You can paste the selected objects at another location by selecting the location on the drawing and placing them on the drawing using **Edit** > **Paste**.

To Insert a Sheet

Click **Insert** > **Sheet** to insert a blank drawing sheet. The sheet is inserted after the last sheet.

To View a Sheet

1. Click **View** > **Go to Sheet** or double-click the sheet number tag at the bottom of the current drawing sheet. The **Go to Sheet** dialog box opens.
2. Type the sheet number or click **Previous**, **Next**, or **Go To**, depending on the current position of the sheet.
3. Click **Close**.

To Move a Sheet

1. Click **Edit** > **Move Sheet** to move the sheet to the required position. The **Move Sheet** dialog box opens.

2. Under **Insert sheet after**, sheets for the current drawing are displayed. Select a sheet after which you want to place the selected sheet. Select **Insert at beginning** to place the sheet at the beginning of the drawing.

3. Click **OK**.

To Delete a Sheet

1. Click **Edit** > **Remove** > **Sheet**.

2. At the prompt, type the sheet numbers to delete and press ENTER.

   Note: Sheet numbers must be separated by commas.

To Move Objects

1. Select a component, connection, note, or symbol using the graphical selection methods or use any of the following filters:

   - Diagram Item
   - Component
   - Connector
   - Table
   - Note
   - Wire

   The selected objects are highlighted.

   Note: Using a Connector filter you can select a Parametric, single view, multi-view, or Inline type of connector. If you use a Component filter, you can select a component group and single view or multi-view type of components.

2. Drag the item to its new position.

3. Left-Click to place the item at the new position.

Alternatively, you can select the objects you want to move and use **Edit** > **Move Special** to change the x- and y-coordinates of the object.

Note: When moving components, connectors, and wires, Pro/DIAGRAM retains all the established links. Use **View** > **Repaint** to regenerate the diagram view after moving some items.

To Move Objects to Another Sheet

To move components, connectors, and conductors from one sheet to another:

1. Select the items that you want to move to another sheet using the appropriate filter or any graphical selection method. The selected objects are highlighted.

2. Click **Edit** > **Move Item to Sheet**.

Note: If the diagram has only one sheet, Pro/DIAGRAM creates a second sheet and places the items on the new sheet after you click **Move Item to Sheet**.

3. At the prompt, type the number of the sheet to which you want to move the item. You can enter a new sheet number to insert a sheet and move the items to the new sheet.

Note: All wires attached to the symbol nodes break at an arbitrary point on the first sheet and continue on the destination sheet, with break-off points displayed as "X". You can later modify the wire shape and the break-off point location.

To Rotate Objects in 90-Degree Increments

To rotate Components or Connectors:

1. Select the items that you want to rotate using the graphical selection methods, the Diagram Item, Component, or Connector filter. The selected objects are highlighted.
2. Click **Edit** > **Properties**. The **EDIT CONN** and **CONN VIEW** menus appear. the **SELECT** dialog box also opens.
3. Click **Rotate 90** in the **CONN VIEW** menu. Pro/DIAGRAM rotates each object counterclockwise by 90 degrees.
4. Click **Done** in the **EDIT CONN** menu.

To rotate Symbols:

1. Select the symbol that you want to rotate, right-click, and select **Properties** from the shortcut menu. The **Custom Drawing Symbol** dialog box opens.
2. Under **Properties**, switch the symbol instance angle between 0, 90, 180, and 270 degrees. The selected angle appears in the **Angle** box.
3. Click **OK**.

To Edit Labels or Reference Designators

1. Click **Edit** > **Value** to edit variable text labels or reference designators.
2. Select the object label or reference designator you want to edit. You are prompted to enter the value for editing.
3. At the prompt, type a new value for the label or reference designator and press ENTER. The text value is changed.Click **Edit** > **Value** to edit variable text labels or reference designators. Click **Edit** > **Value** to edit variable text labels or reference designators.

To Delete Diagram Objects

1. Select an object that you want to delete. Select multiple objects by using a selection rectangle around the objects that you want to delete or simultaneously hold down the SHIFT key and select the objects. You can also use an appropriate

filter to select multiple objects for the selected filter. The selected objects are highlighted.

2. Click **Edit** > **Delete**. The selected objects are deleted.

   Note: If you delete components that have wires routed to them, the **Confirm Delete** dialog box opens and you are prompted to take action on the connected wires. You can delete the attached wires or leave the wires as unattached.

3. If you have selected multiple items, clear the **Apply to All Selected** check box and click **Apply** in the **Confirm Delete** dialog box.

   or

   If you have selected a single item for deletion and right-click to select **Delete** from the shortcut menu.

4. To restore the deleted diagram objects, click **Edit** > **Undelete** or right-click and select **Undelete** from the shortcut menu.

   Note: Deleted objects are restored only if you perform an **Undelete** operation immediately after a **Delete** operation.

Copying Diagram Objects

Copying Objects

Using **Edit** > **Copy**, you can copy drawing items, not only from within the same diagram, but also from one diagram to another. Items that you copy from one diagram to another retain the same reference designator. However, if the target diagram has an item with a reference designator that is the same as the copied item, Pro/DIAGRAM appends the suffix `_cN`, where `N` is a number, to the reference designator of the copied item in the target diagram.

The following guidelines can be helpful as you use the **Copy** option:

- If you copy a cable or wire, but the spool for the cable or wire does not exist in the target diagram drawing, Pro/DIAGRAM copies the spool along with the cable or wire.

- If you copy a cable or wire, but a spool for the cable or diagram has the same name as an existing spool in the target diagram, Pro/DIAGRAM compares their parameters.

  If the parameters for the two spools are different, it copies the spool from the source diagram and gives this spool a nonconflicting name by appending the suffix `_cN`, where `N` is a number, to the spool name. The copied wire or cable references this new spool. It also copies relevant symbol definitions, if they do not already exist in the target diagram.

- When you copy items from one diagram to another, Pro/DIAGRAM does not copy layer information from any source items to the target diagram.

Note that, because no two items in a diagram can have the same reference designator, Pro/DIAGRAM appends the suffix `_cN`, where `N` is a number, to the reference designator of each copied symbol in the diagram. You can change this reference designator later.

To Copy Items

1. Select the object or objects that you want to copy using an appropriate filter for selection of items or any graphical selection method.
2. Click **Edit** > **Copy**. Use the sheet number box on the menu bar to switch sheets, if necessary.
3. Click **Edit** > **Paste**. The **Diagram Clipboard** window opens showing the copied object.
4. In the **Diagram Clipboard** window, click on a point in or around the copied object where you want it attached to the pointer. A small yellow box marks the pick point.
5. In the diagram, click a point where you want the attach point placed. The copied object is added to the diagram at the selected point.

To Copy Items From One Diagram To Another

1. Click **File** > **Open** to open the diagram from which to copy an object or objects to another diagram.
2. Select the items that you want to copy to another sheet using the graphical selection methods or the Diagram Item filter. The selected objects are highlighted.
3. Click **Edit** > **Copy**. Use the sheet number box on the menu bar to switch sheets, if necessary.
4. Click **Edit** > **Paste**. The **Diagram Clipboard** window opens showing the copied object. The **GET POINT** menu appears with the following options:
   - **select Pnt**—Select a point.
   - **Vertex**—Select an entity end point.
   - **On Entity**—Select a point on an entity and correct the parameter.
   - **Rel Coords**—Type the offset from the ruler reference point.
   - **Abs Coords**—Type the coordinates.
   - **Done**—Accept this entity.
   - **Quit**—Reject this entity.
5. In the **Diagram Clipboard** window, select a point in or around the copied object or objects where you want it attached to the pointer. A yellow box marks the selected point.

6. Click **Window** > <**Diagram Name**> to navigate to the diagram where you want to place the copied object or objects, if the copied object or objects belong to a different diagram.

7. In the selected diagram, select a point where you want to place the attached point and click **OK** in the **SELECT** dialog box. The copied object or objects are added to the diagram at the selected point.

Note: Click **Edit** > **Transform** for more advanced copy options, including:

- **Translate**—When pasting, defines a directional vector and number of copies.
- **Rotate**—When pasting, places a specified number of copies around a center point at specified angle points.

Adding and Editing Components

About Components

Components are schematic representations of electromechanical parts or subassemblies. Each component symbol uniquely identifies or represents a physical model, unless both symbols are part of the same multiple-view component (MVC). Two or more occurrences of the same component symbol represent different instances of that component. They must have different reference designators and can have different attributes.

A component has a graphical symbol representation and a set of defining parameters. Each component symbol must contain a set of predefined attach points called nodes, which define wire terminators and pins. You can define your own component symbols or retrieve them from the Pro/ENGINEER electrical symbols library.

Nodes are displayed as green circles in Pro/DIAGRAM. The radius of the circle is controlled by the `node_radius` diagram setup option.

Components vs. Connectors

A component is a type of symbol with the parameter `OBJ_TYPE = COMPONENT` associated with it. Components use reference designators as unique identifiers and include properties for pins.

Connectors are a sub type of component with the `OBJ_TYPE` value of `CONNECTOR`. A connector can use two additional parameters; `GENDER` (`MALE` or `FEMALE`) at the component level and `ENTRY_PORT` at the pin level. These parameters are used specifically to pass information to Pro/CABLING. The gender is a design reference for the intended solid model and is required for connectors. The entry port is the name of a coordinate system on the solid model that specifies the beginning or end of a connection. In Pro/CABLING, use the `ENTRY_PORT` parameter to designate pin-to-pin wire and cable connections for manual cabling or autorouting.

Automatic Entry Port Designation

If ENTRY_PORT in Pro/DIAGRAM has a value the same as a valid coordinate system in the 3D connector in Pro/CABLING, Pro/CABLING automatically designates the coordinate system as an entry port, adding the ENTRY_PORT parameter to the connector and using default values of ROUND and 0.0 for the grouping and internal length values. This saves you having to enter the values manually after importing a logical reference into Pro/CABLING. The Internal length applies to all wires routed to pins represented by that entry port, not to individual pins.

Note: The ENTRY_PORT parameter can be used with both component and connector.

To Add a New Single View Component

1. Click **Insert** > **Component** > **Single View**. The **DGM SYM TYPE** and the **GET SYMBOL** menus appear. When you add a component, you place an instance of a predefined symbol into the diagram.
2. Click one of the following options from the **DGM SYM TYPE** menu:
   - **Free**—Places the view as a regular component freely on the diagram.
   - **Insert**—Inserts the view into a wire. When selected, only components with two pins or less are valid components for placement. Insertion causes the wire to be broken into two wires.
   - **Att Wire**—Places the view as a through splice. When selected, only components with one pin are valid components for placement.
   - **Replace**—Replaces a selected component.
3. Use the **Get Symbol** options to select the component symbol. If the symbol does not have all the appropriate parameters, you are prompted to specify if they must be corrected. Enter [Y] to correct them and place the symbol.
4. If the symbol height is defined as **Variable**, you are prompted to enter the symbol height.
5. If the symbol definition includes variable text, you are prompted to enter the text.
6. Select a location on the diagram for the symbol origin (specified when defining the symbol).
7. The component symbol appears in the diagram. The **Electrical Parameters** dialog box opens. Enter and modify parameters for the component.
8. When you are finished, click **OK**. The **Adjust Inst** menu appears with the following options to adjust the symbol placement:
   - **Relocate**—Selects a drag point on the symbol (not necessarily the same as the symbol origin). Specify its new location. Middle-click to abort.
   - **Move Origin**—Selects the new symbol origin. The symbol moves immediately so that the new origin is in place of the old one. The symbol origin determines its position when you move it later using Move.

- **Rotate 90**—Rotates the symbol around its origin 90 degrees counterclockwise.
- **Resize**—Enter new value for symbol height. This option is only available if the symbol is defined as `Variable`.
- **Done**—Completes the component position.

9. Proceed to create other components, or click **Quit** from the **GET SYMBOL** menu.

Using Components in Multiple Views

About Multiple View Components

Use multiple view components (MVCs) when you must show the same component, or parts of the same component, more than once in a diagram. The MVC (or connector) can appear repeatedly in the diagram, but is a single object in the Bill of Materials. It has one reference designator and one set of parameters that apply to all of its views.

After you have added a MVC or connector, you can add an instance of this component or connector to a diagram. The instance is a representation of the MVC. Only one view can be added at a time. You can also insert the view in an existing wire or a cable.

When you add instances of the MVC definition, you can place all the symbols defined within the view, or you can place selected symbols from the view to show only details associated with them. Each item within the MVC definition can be placed separately and moved, rotated or positioned as necessary.

Note:

- Symbols used to define a MVC can also be used individually as regular components that are not associated with a multiple view component.
- If a symbol is used both individually and as a MVC, any parameters that are set for the individual symbol do not affect the multiple view component.
- You must first create individual symbols and then define a MVC by including the individual symbols.

  For example, if a MVC consists of a contact and a coil, both the contact and the coil must first be created individually before they can be brought together to define a MVC.

To Create a New Multiple View Definition

1. Click **Format** > **Multi-View Component Gallery** or **Multi-View Connector Gallery**. The **MULTI VW DEF** menu appears.
2. Click **Define** to create a multiple view definition.
3. Enter a name for it in the information line when prompted and press ENTER. The **MULTI REDEF** and **GET SYMBOL** menus appear.

4. Select an appropriate command from the **MULTI REDEF** menu.

   Note:

   - The parameter `Model_Name` for a multiple view component is defined in the definition of the MVC and is retained even when a view of an instance of the MVC is placed.
   - Parameters that apply to MVCs have no effect on the original symbols.

5. Click **Add View** to add a symbol to the MVC definition. You can select a symbol by name, select an instance of an existing symbol, or retrieve a symbol from disk. Repeat this step until you have added all the symbols required to define your MVC.

   Note: You can add the same symbol multiple times.

6. Click **Delete View** to remove any unwanted symbols from your MVC definition.

   By default, the pins on the symbols you add to your MVC definition assume the names defined in the symbol definitions. Conflicts are automatically resolved using extensions `_c0`, `_c1`, and so on.

7. Click **Pin Names** to re-define the pin names for the MVC. You are given each symbol's name, its pin names, and the corresponding multiple view component pin name. Modify any MVC pin name as needed.

8. Click **Done** from the **MULTI REDEF** menu. A new MVC is defined with the name you specified using the symbols you added.

9. To save this MVC to disk, click **Write** from the **MULTI VW DEF** menu. Select the new MVC definition that you have specified and click **OK** in the **Select MVC Definitions** dialog box. The MVC definition is saved with a `.mvc` extension.

To Add Multiple Views

1. Click **Insert** > **Component (or Connector)** > **Multi-View**. The **Select MVC Definitions** dialog box opens.

2. Select the required definition from the dialog box and click **OK**. You are prompted to enter a reference designator.

3. Enter a reference designator and press ENTER. The **VIEW NAMES** menu appears showing the unplaced views in the selected MVC.

4. Select the symbol for the view that you want to place. The **GET POINT** menu appears.

5. Choose the appropriate option and click to place the symbol at a required location on the diagram.

   Note: If the selected view contains one or more nodes, the **DGM SYM TYPE** menu appears. Use this menu to place the views in a wire or a cable.

6. Click **Free**. The **GET POINT** menu appears.

7. Place the symbol using the placement methods available on the **GET POINT** menu.

   If you use the same symbol more than once in the MVC definition, the corresponding views in the MVC have incremented node names, for example _c1, _c2, and so on.

8. Repeat step 4 until you have placed all the required views. Middle-click to complete the task.

   Note: All views of a MVC must exist on the same layer of a drawing. However, views can exist on different sheets.

To Delete an MVC Definition

1. Click **Format** > **Multi-View Connector Gallery** or **Multi-View Component Gallery**. The **MULTI VW DEF** menu appears.
2. Click **Delete** from the **MULTI VW DEF** menu. The **Select MVC Definitions** dialog box opens.
3. Select the MVC you want to delete. You are prompted to confirm the deletion.
4. Click `Yes` at the prompt.

To Delete a Placed Multiple View Component or Connector from the MVC Definition

1. Click **Format** > **Multi-View Component Gallery** or **Multi-View Connector Gallery**. The **MULTI VW DEF** menu appears.
2. Click **Redefine**. The **Select MVC Definitions** dialog box opens.
3. Select the required multiple view component instance that you want to remove from the diagram sheet. The **MULTI REDEF** menu appears.
4. Click **Delete View** to remove a view from the MVC definition. The **VIEW NAMES** menu appears.
5. Select the view to be deleted and click **Done** in the **MULTI REDEF** menu.

   Alternatively, you can also select a view to be deleted from the diagram sheet and right-click to select **Delete** from the shortcut menu.

Note:

- Deleting all views of a MVC from a diagram sheet does not delete its definition.
- If wires are attached to the view, you are prompted to delete the wires or leave them unattached.
- You cannot delete a symbol definition if an MVC is using it.

To Show the Multiple View Component Name

1. Click **Format** > **Multiple-View Component Gallery**. The **MULTI VW DEF** menu appears.
2. Click **Show Name** to display the name of the multiple-view component (MVC) to which a symbol belongs and then click the symbol in question.

To Modify a Multiple View Component or Connector

1. Click **Format** > **Multi-View Component Gallery** or **Multi-View Connector Gallery**. The **MULTI VW DEF** menu appears.
2. Click **Redefine**.
3. Select the multiple-view component (MVC) or connector that you want to modify. The **MULTI REDEF** menu appears.
4. Use this menu to add or delete symbols to and from your MVC or connector. You can also modify the MVCs or connector's parameters and pin names.
5. Click **Done** when you finish modifying the MVC.

To Modify Multiple View Definition Pin Names

1. Click **Format** > **Multi-View Component Gallery** or **Multi-View Connector Gallery**. The **MULTI VW DEF** menu appears.
2. Click **MULTI VW DEF** > **Redefine**. The **Select MVC Definitions** dialog box opens.
3. Select the symbol for the view that you want to redefine. The **MULTI REDEF** and **GET SYMBOL** menus appear.
4. Click **Pin Names** from the **MULTI REDEF** menu. The text editor opens.
5. Modify the value of the items in the pin name column to give a different value to a pin in your MVC.

Using Connectors

About Connectors

A connector is a specific type of component symbol with the parameter and value `OBJ_TYPE = CONNECTOR`. A connector can use two additional parameters; `GENDER` (`MALE` or `FEMALE`) at the component level and `ENTRY_PORT` at the pin level. Connectors may have wires and cables routed between the nodes or pins on the symbol.

There are two types of connectors in Pro/DIAGRAM:

- **Fixed**—The connector appears as a drawn, fixed-shape symbol, either created in session or retrieved from a library. Fixed connectors are defined, stored, and added like components and differ only in parameters.
- **Parametric**—The parametric connector appears as a standard symbol, its shape determined by the specified connector attributes, such as gender and number of pins. When the connector attributes are modified, the appearance of the symbol updates accordingly.

To Add a Fixed Connector

To add a fixed connector, you place a predefined symbol into the diagram:

1. Click **Insert** > **Connector** > **Single View**. The **DGM SYM TYPE** menu and **GET SYMBOL** menus appear.
2. Retrieve a previously defined connector symbol. If the symbol does not have all the appropriate connector parameters, you are prompted to correct them. Click `Yes` at the prompt to correct the selected symbol and place the symbol in the drawing.
3. If the symbol is defined as `Variable`, enter the symbol height.
4. If the symbol definition includes variable text, enter the text as prompted.
5. Select a location on the diagram. This is where the symbol origin (specified when defining the symbol) is placed. The connector symbol appears in the diagram. The **Electrical Parameters** dialog box opens.
6. Enter the appropriate parameter values into the dialog box, or use **File** > **Read** to load parameters from a file. You must have values for all required parameters.
7. Complete the change with the **ADJUST INST** menu options.
8. Create other fixed connectors, or click **Quit** from the **GET SYMBOL** menu.

To Move Components and Connectors

1. Select the symbol that you want to move. The selected symbol is highlighted in red.
2. Click the symbol again to attach it to the pointer.
3. Drag the symbol to the required location and click to indicate the new location. The symbol is placed such that its origin is at the selected point. All wires attached to the symbol nodes follow. The shape of the wires is updated parametrically with the new vertical and horizontal segments created automatically, if needed.
4. To continue to modify the location of this symbol, select the connector and select a new location.

Alternatively, you can select the objects that you want to move and use **Edit** > **Move Special** to change the x- and y-coordinates of the object.

To Mark a Pin Name as Lowercase

If you need to have pin names of the same letter, differentiated only by case, use the less than symbol (<) before the pin name to mean that the pin name letter is lowercase. For example:

`<A` means `a`

`<ABC` means `abc`

The name is not displayed in lowercase, but you can have two components or connectors in the same diagram named `<A` and `A`.

To Show or Hide Node Names

1. Select a connector from the drawing using the graphical selection methods, the Diagram Item, Component, or Connector filter. The selected connector is highlighted.
2. Click **Edit** > **Properties**. The **EDIT CONN** and **CONN VIEW** menus appear.
3. Click **Param Display**. The **PARAM DISP** menu appears.
4. Click **Show** or **Erase**.

   Note: Clicking **Erase** hides the node names and does not erase them permanently.

Tip: Switching Node Names

You can switch node names for fixed connectors and components using the same method you would use to switch the node sequence for parametric connectors. Only **Default** and **Switch** are available in the **Edit** > **Properties** > **EDIT CONN** > **Pin Sequence** menu.

You can switch two node names as long as the nodes belong to the same connector, and you can also switch node names among different views of a multiple view component.

- If you switch a node name in one view of a MVC with a node name from another view of the same MVC and delete the original view from the symbol definition, Pro/DIAGRAM renames the node in the remaining node to its original name, as defined in its symbol definition.
- If you rename one of the nodes, the node names remain switched.

Parametric Connectors

About Parametric Inline Connector

Pro/DIAGRAM supports inline parametric connectors that consist of one male and one female connector with different reference designators, each with an identical number of pins, mated together.

When you connect two parametric connectors to create an inline connector, perform the following tasks:

- Renames the nodes of the second selected connector to match those of the first selected connector.
- Rearranges the visible pins of the second selected connector to line up with those of the first selected connector.

Multiple parametric connector symbols can be displayed as partial views of the connector. The connector views can also be arbitrarily split and merged between any two pins and the new views can be placed on different sheets.

To Add a Parametric Connector

1. Click **Insert** > **Connector** > **Parametric**. The **PARAM CONN** menu appears.
2. Click **Male** or **Female**.
3. Click a location for the symbol in the diagram. You are prompted for the number of pins.
4. Enter the number of pins on the prompt line and press ENTER. The **Electrical Parameters** dialog box opens.
5. Enter appropriate parameter values into the dialog box, or use **File** > **Read** to import them from a file. The connector symbol appears in the diagram.
6. Complete the connector using the options in the **EDIT CONN** and **CONN OPTIONS** menus.

Note:

- The default origin of the symbol is under pin 1. If you change the pin gap, the pin location moves, but the origin point remains at its original location.
- At the prompt, you can enter up to 999 pins for the **Num Vis Pins** for a parametric connector.

To Create a Parametric Inline Connector

1. Click **Insert** > **Connector** > **Inline**.
2. The **INLINE CONN** menu appears with two options:
    - **Select Wires**—Lets you click several wires to which to attach the inline connector. The selected wires must be parallel to each other. If a wire in a cable is selected, all other wires in the cable must be selected as well. If you create an Inline connector in this manner, each selected wire splits into two wires and places the inline connector at the point where the wires are separated.
    - **Sel Location**—Lets you create an inline connector at a specified location on your drawing instead of on selected wires. This is the default option. After choosing this option, select the location for the inline connector.

3. Enter the number of visible pins for this connector. The **Electrical Parameters** dialog box opens, showing lines for both connectors
4. Enter the appropriate reference designator and parameter values for each connector into the dialog box and click **OK**. The inline connector appears in the diagram.
5. Use the **EDIT CONN** and **CONN OPTIONS** menus to complete the creation of the inline connector.

Note: To create an inline connector from two parametric connectors of opposite gender, use the **Connect** option.

To Modify the View of a Parametric Connector

1. Select a parametric connector to modify using the graphical selection methods, the Diagram Item, Component, or Connector filter. The selected connector is highlighted.
2. Click **Edit** > **Properties**.

   or

   Select the parametric connector to modify and right-click to select **Properties** from the shortcut menu. The **EDIT CONN** and **CONN VIEW** menus appear.
3. Use the **EDIT CONN** and **CONN VIEW** menus to modify the connector views.

Note: You can also use this procedure to modify the gender, visible pins, and pin spacing of a parametric connector.

To Edit the Pin Sequence of a Parametric Connector

1. Select a parametric connector to modify.
2. Click **Edit** > **Properties**.

   or

   Select the parametric connector to modify and right-click to select **Properties** from the shortcut menu. The **EDIT CONN** and **CONN VIEW** menus appear.
3. Click **Pin Sequence** from the **EDIT CONN** menu.
4. Select one of the following commands at the bottom of the **PIN SEQUENCE** menu before you select the editing method:
   - **Fix Wires**—Does not change the location of wires as you modify the pin sequence.
   - **Adjust Wires**—Reconnects wires according to the new pin sequence.
5. Select one of the following commands from the **PIN SEQUENCE** menu to edit the pin sequence.
   - **Default**—Restores the default pin sequence.

- **Enter Seq**—Displays the current pin sequence in a message window. (Number names cannot exceed the value of the `NUM_OF_PINS` parameter.)
- **Select**—Clicks a node to change its number.
- **Switch**—Selects two nodes to switch numbers.
- **Reverse**—Reverses the current numbering sequence for the connector.

6. When finished, click **Done** from the **EDIT CONN** menu.

To Modify the Name of a Reference Designator

1. Click **Edit** > **Value**.
2. Select a reference designator to modify.
3. At the prompt, type a new value for the reference designator.
4. Press ENTER.

Note: Alternatively, click **Tools** > **Parameters** > **Objects** to modify the parameter value of the selected object or objects in the **Select by Type** dialog box.

To Modify the Pin Gap in Parametric Connectors

1. Select a parametric connector to modify using the graphical selection methods or the Connector filter. The selected parametric connector is highlighted.
2. Click **Edit** > **Properties**.

   or

   Select the parametric connector to modify and right-click to select **Properties** from the shortcut menu. The **EDIT CONN** and **CONN VIEW** menus appear.
3. Click **Mod Pin Gaps** on the **CONN VIEW** menu. The **GAPS** menu appears. You can define or modify the distance between selected pins during or after creation of a parametric connector.
4. Click **Add** to increase the distance between pins in any parametric connector and click between any two pins (or between an end pin and the end of the connector symbol.) The distance is increased according to the current pin spacing.

   or

   Click **Remove** to remove gaps between pins.

To Move Parametric Connector Text

1. Select the connector containing the text that you want to move.
2. Click **Edit** > **Properties**. The **EDIT CONN**, **CONN OPTIONS**, and **CONN VIEW** menus appear.
3. Click **Move Text** from the **CONN VIEW** menu.

4. Select the node names or the reference designator text that you want to move. If you click a node name, Pro/DIAGRAM automatically selects all node names for the connector view.
5. Select a new location where you want to place the node names or the reference designator text. Middle-click to cancel the operation.

Note: Pro/DIAGRAM labels new pins that are added to a parametric connector using the same offset of the labels for existing pins. If two parametric connectors are joined to form an inline connector, the labels for the second parametric connector revert to their default positions.

To Modify Parametric Connector Text Height

1. Select a parametric connector containing the text that you want to change.
2. Click **Format** > **Text Style**. The **Text Style** dialog box opens
3. Enter a new value in the **Height** box.
4. Click **Apply** to implement the change and then **OK** to close the **Text Style** dialog box.

To Specify the Default Parametric Connector Node Label

1. Click **File** > **Properties**. The **Options** dialog box opens.
2. Use the diagram-specific setup option `pc_node_label` to set the default node label of parametric connectors. The value can be any text, a pin parameter, or any combination, including a user-defined pin parameter. The default node label is `&node_name`, which corresponds to the pin name for the node. Other valid parameter names include:
   - `&signal_name`
   - `&signal_value`
   - `&entry_port`
   - `&user_defined`

You must first define a user-defined pin parameter for the connector before its value can be shown in the parametric connector node label. If the parameter does not exist, then it shows in the note as parameter text, for example, `&address`.

To Edit the Parametric Connector Node Name

1. Click **Edit** > **Value**.
2. Select a node or a node name that you want to modify.
3. At the prompt, type the new node name.
4. Press ENTER.

Note: Alphanumeric names are valid.

Grouping Component Symbols

About Grouping Connector and Component Symbols

There are groups of symbols that are usually used repeatedly in many different diagrams and multiple times in a single diagram. These groups typically consist of a component symbol and several connector symbols. You can define a group of single view components, connectors, and detail symbols that can move, scale, rotate, and mirror all together.

An example of a symbol group is a cooling fan component, whose symbol is grouped with the power and control connectors that always appear with it.

All symbol parameters, text, and geometry are taken from the individual symbol definitions. All symbols that are included in the component group must have the option for free placement in their definition.

Note: You cannot use a multiple-view connector (MVC) to represent the group because every item belonging to this group is either a connector or a component and must have its own parameters and reference designators. For a multiple-view component or connector, the reference designators and the parameters are defined across all the views of a MVC definition.

To Create a Component Group Definition

1. Click **Format** > **Component Group Gallery**. The **CMP GRP DEF** menu appears.
2. Click **Define**. At the prompt, type a component group name. The **CMPGRP EDIT** menu and the **CMPGRP_EDIT_<NAME>** window opens.
3. Click **Insert** > **Component Group View** from the **CMPGRP_EDIT_<NAME>** window. The **GET SYMBOL** menu appears.
4. Click **Retrieve** to add a detail symbol, connector, or component from disk. You can also use the **Name** and **Pick Inst** commands to select symbols available in the current component group definition.
5. When you have placed all the symbols for the group in the window, the **ADJUST INST** menu appears.
6. Use the commands in the **ADJUST INST** menu to relocate, resize, or rotate the component by 90 degrees and click **Done**.
7. Click **Quit** in the **GET SYMBOL menu**.
8. Click **Set Origin** from the **CMPGRP EDIT** menu to change the origin of the component group definition. The **GET POINT** menu appears.
9. Use the commands in the **GET POINT** menu to select a point and set the origin.
10. Click **Done** from the **CMPGRP EDIT** menu.

To Save a Group Definition File

A group definition must exist for this command to be active.

1. Click **Format** > **Component Group Gallery**. The **CMP GRP DEF** appears.
2. Click **Write**. The **Select Comp Group Definitions** dialog box opens.
3. Select the definition to save. You are prompted for a path.
4. Enter a path and press ENTER. The file is saved as `<groupname>.cpg`.

   Note: If you do not specify a path, Pro/ENGINEER saves the component group definition file in the startup directory.

To Modify a Component Group Definition

1. Click **Format** > **Component Group Gallery**. The **CMP GRP DEF** menu appears.
2. Click **Redefine**. The **GET CMPGRPDEF** menu appears and the **Select Comp Group Definitions** dialog box opens at the same time.
3. Select the component group to redefine. It can be a symbol, component, or a connector and click **OK**. The **CMPGRP_EDIT_<NAME>** window opens and the **CMPGRP EDIT** menu appears at the same time.
4. If you want to add a new member to the component group, click **Insert** > **Component Group View** from the **CMPGRP_EDIT_<NAME>** window. The **GET SYMBOL** menu appears.
5. Click **Retrieve** to select the symbol, connector, or component you want to edit. You can also use the **Name** and **Pick Inst** commands to select existing symbols in the diagram from the **Select Comp Group Definitions** dialog box.
6. When you have placed all the symbols for the group in the window click **Quit**. The **ADJUST INST** menu appears.
7. Use the commands in the **ADJUST INST** menu to relocate, resize, or rotate the component by 90 degrees and click **Done**.
8. Click **Quit** in the **GET SYMBOL menu**.
9. If you want to change the origin of the component group, click **Set Origin** in the **CMPGRP EDIT** menu to change the origin of the component group definition. The **GET POINT** menu appears.
10. Use the commands in the **GET POINT** menu to select a point and set the origin.

    Note: You cannot change the location of an individual member after it is placed on the diagram sheet. If you move (drag) an individual member of the group, the other members of the group also move. This is because the physical location of the members in a group is defined at the group definition level.
11. Click **Done** from the **CMPGRP EDIT** menu.

Note: You can add symbols that are not components or connector object types to the component group, but you must add them as plain symbol instances with no parameters.

To Create a Component Group Instance

1. Click **Insert** > **Component Group**. The **Select Comp Group Definitions** dialog box opens.
2. Select the component group to redefine. You are prompted to name the instance.
3. Type a name for the new instance of the component group. The **VIEW NAMES** menu appears showing unplaced views in the group.
4. Select views to place from the **VIEW NAMES** menu. The **Electrical Parameters** dialog box opens if you are placing a component or connector view.
5. Enter a **REF_DES** name and a **MODEL_NAME** for each component in the group.
6. Modify the parameters as required and click **OK**.

   Note:

   - If you are placing a plain (detail) symbol, no parameters are required, so the **Parameters** dialog box does not open.
   - If you use a plain (detail) symbol, you are unable to route wires to it, even if it has nodes.
7. Select a symbol location in the main diagram window. The view of the instance of the object is placed at that location with the new reference designator name. As you select additional views to add, they are automatically placed because their relative location is defined in the component group definition. You can place some or all of the views in a component group instance when you create them.
8. To display additional views of an existing instance where all views are not placed, click **Insert** > **Component Group View** and select an existing member of the component group. The **VIEW NAMES** menu appears showing a list of additional views in the group.
9. Select the additional views to place from the **VIEW NAMES** menu and place them in the diagram.

Note: You can select an instance in the diagram about which to obtain information by clicking **Info** > **Component Group**.

- You can use the commands on the **Format** menu to manipulate component group symbol instances.
- To resize a component group instance, the individual subsymbols must all be of variable height.

Creating Spools

About Spools

Each wire and cable in the design must be created from a predefined spool. Each spool has a unique name and a unique set of parameters and values that are passed to the wire or cable when the wire or cable is created. The spool is saved within the diagram file. It can also optionally be written out to a text file (`.spl`) that can be referenced in new diagram and cabling designs.

Spools have a certain number of required parameters. You can also include standard optional parameters and custom defined informational parameters in the spool definition.

Wire Spools

Wire spools require values for `Name` and `Type`. The name value is the unique identifier, the type value is `WIRE`, to differentiate it from a cable spool. Other common but optional parameters defined in the wire spool are `color` and `wire_gauge`. Some parameters are automatically generated, for example `min_bend_radius`, to provide values for calculations in Pro/Harness Manufacturing.

Cable Spools

Cable spools have the same required parameters as wire spools, plus the required parameter `num_conductors`, which defines the number of insulated conductors included in the cables that will be created from the spool. When you create a cable spool, you edit the new cable spool file using the **Electrical Parameters** dialog box to include an integer for the number of conductors and a `<define conductor-enddef>` section for each conductor.

Note: Although you can legally create cable spools without the define conductor sections, you should set up the conductor definitions if you intend to use the diagram as a logical reference for autorouting in Pro/Cabling.

Active Spool

When you first start to route a wire or create a cable, you are prompted to select a spool from which to create the wire or cable. The selected spool becomes the active spool. The spool name is displayed in the lower right corner of the sheet. Any further wires or cables you add are created from their respective active spool. You may change the active spool to a different spool at any time.

Note: The information defined in a spool is important for any Pro/REPORT tables and wire/cable labels you create.

You can write a spool file in ASCII format to be retrieved for use in other drawings.

The configuration file option, `pro_spool_dir`, lets you specify the directory from which spools can be read. If this configuration file option is not set, the current working directory is used.

To Create a Wire or Cable Spool

1. Click **Format** > **Spools** > **Create**.

   The **SPOOL TYPE** menu appears with the following options:

   - **Wire**—Create a wire spool.
   - **Cable**—Create a multi-conductor cable spool.

   Click the type of spool you want to create. You are prompted to specify a name for the spool.

2. At the prompt, type a name for the spool. The **Electrical Parameters** dialog box opens.
3. Click **View** > **Columns** and use the **Model Tree Columns** dialog box to add more parameters to the spool, if required.
4. Select the **NUM_CONDUCTORS** parameter in the **Electrical Parameters** dialog box and specify an appropriate integer value in the **Value** box. This parameter decides the maximum number of conductors that a cable can have.

   Using the **Electrical Parameters** dialog box, you can add parameters to wires or cables of a connector or to the conductors of a cable spool.

5. When finished, if you want to save the spool data, click **File** > **Write** and then **OK** to close the **Electrical Parameters** dialog box.

To Modify Spool Parameters

1. Click **Format** > **Spools** > **Edit**. The **Select Spools** dialog box opens with a list of all the spools used in the diagram.
2. Select the name of a spool to edit or hold down the CTRL key and select multiple spools from the list. Optionally, use the **Spool Filter** commands to filter the list by type or number of conductors.
3. Click **OK**. The **Electrical Parameters** dialog box opens.
4. Edit the required parameters.
5. Save the diagram to save changes to the spool in the current diagram.
6. Use the **File** > **Write** option in the **Electrical Parameters** dialog box to modify the spool definition for use in other diagrams.

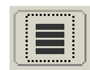
Note: Alternatively, click **Tools** > **Parameters** > **Objects**. In the **Select By Type** dialog box select **Spools** as the object type. Click **Modify** and use the **Electrical Parameters** dialog box to modify the spool definition.

To Rename Spools

1. Click **Format** > **Spools** > **Rename**. The **Select Spools** dialog box opens.

2. Click a spool name from the spool list. You can use the pull-down menus over the list to filter the list by spool type or number of conductors.
3. Click **OK** in the **Select Spools** dialog box. You are prompted for a new spool name.
4. At the prompt, type a new spool name and press ENTER. The spool is renamed.
5. Click **Done/Return** in the **SPOOLS** menu.

To Save and Retrieve Spools

1. Click **Format** > **Spools** > **Write**. The **Select Spools** dialog box opens.
2. Click a spool name or names from the spool list. Click  to select all the spools from the list or hold down the CTRL key and select multiple spools from the list. You can use the pull-down menus over the list to filter the list by spool type or number of conductors.
3. Click **OK** in the **Select Spools** dialog box to save an existing spool to disk. If the spool is written, it is confirmed at the prompt. If multiple spools are written, a list of all the spools written to the output files is displayed in the INFORMATION WINDOW that confirms that the spools have been written to the default directory.

To read in a previously written spool (`.spl`) file:

1. Click **Format** > **Spools** > **Read**. The **Open** dialog box opens.
2. Select a spool file name (`.spl`). You can hold down the CTRL key and select multiple spools from the directory browser.
3. Click **Open** in the **Open** dialog box.
4. At the prompt, to read an existing spool you are prompted to overwrite the spool. If the spool is successfully read, it is confirmed at the prompt.

Adding Wires

About Adding Wires

To add a wire to the design you must have at least one wire spool created in the database and selected as the active spool. If no wire spool is selected you are prompted to select one. The active spool name is displayed in the lower right corner of the sheet. Be sure it is the spool you want to use to create the wire.

You can use wires to link nodes on one sheet or across multiple sheets. Wire breaks across sheets are marked with an X. You can also end wires in space.

When you complete a wire it is assigned a unique default wire name that you can modify later if necessary. By default the wire name is displayed in the wire label. You may edit the wire label to read other properties of the wire, for example spool, color or strand type. You can also set configuration defaults to adjust the position and occurrence of the label on the wire.

Before you start it is useful to have **Snap to Grid** checked in the **Environment** dialog box and the grid set to the width between the pins on your connectors so that the pins lie on the grid.

Tip: Orthogonal and Nonorthogonal Sketching

Pro/DIAGRAM lets you sketch wires and other items at any angle. By default, wires and other items sketched in Pro/DIAGRAM appear in horizontal and vertical orientations only. To change this clear the **Snap to XY Axes** option in the **Environment** dialog box.

To Create a Wire Path

Before you start it is useful to have **Snap to Grid** checked in the **Environment** dialog box and the grid set to the width between the pins on your connectors.

1. Click **Insert** > **Wire**. If you are already working with an active wire spool, the **CREATE WIRE** menu appears.

   If no wire spool is active you are first asked to select or create one.

2. Click **Sketch Path**.
3. Select a node. A red line follows the pointer. You can also select a point on a rail, or a point in space.
4. Left-click to change the direction.

   Middle-click to break the wire and continue it on another sheet.

   Right-click to terminate the wire at the last left click in space.

5. To complete the wire, click the destination point. Pro/DIAGRAM gives the wire a default name (`WIREnnnn`) that you can modify using the **Options** dialog box.

To Change the Active Spool

When nothing is selected:

1. Click **Format** > **Default Wire Spool** or **Default Cable Spool**. The **Select Spools** dialog box opens.
2. Select a spool or click **New** to define a new spool.

When inserting a wire:

1. Click **Insert** > **Wire**. The **CREATE WIRE** menu appears.
2. Click **Change Spool**. The **Select Spools** dialog box opens.
3. Select a spool or click **New** to define a new spool.

The selected spool becomes current. All wires added to the diagram use the parameters of the selected spool until you choose **Change Spool** again.

To Make New Wires Follow an Existing Wire

When adding wires, use this procedure to create a pattern of connections based on one selected wire. Once you have selected the pattern wire, you only need to click the from-to nodes of new wires.

1. Click **Insert** > **Wire**. The **CREATE WIRE** menu appears.
2. Click **Follow Wire**.
3. Select a wire, highway, or rail to follow. The wire highlights in red.
4. Select the start and end nodes for the new wire. The new wire is created by offsetting from the selected wire.

To Change the Wire Path

1. Click **Insert** > **Jog**. You are prompted to select the wire, highway, or rail at the position where you want to add a jog.
2. Select the position for the new jog. Pro/DIAGRAM assumes that you are starting to sketch from the end of the original segment that is closer to the position used to select the segment.

   You can create one or more jogs by selecting successive points. A jog is added at a position wherever you click. If you place additional jogs, it is like resketching that one piece of wire, with the end of the sketched section connected with a rubberbanding segment to the other end of the originally selected piece.
3. Middle-click to stop placing the jog.

   Note: To delete a jog, use **Edit** > **Remove** > **Jogs**.

To Change From-To Direction

1. Click **Format** > **Conductor Direction**.
2. Select the wire you want to change. To select all wires in the cable, select the cable symbol, or one of the connectors. The **COND DIRECTION** menu appears.
3. If you select a cable symbol, directional arrows appear showing you the to direction for each conductor.

   If you select a connector, use the **COND DIRECTION** menu to specify all wires going to or from the selected connector.

   If you select a single wire, a message, `Direction of wire XXX has been flipped` appears at the prompt.

You can change the from-to direction of an entire cable or individual wires within the cable.

To change the from-to direction of a wire or cable using a table:

1. Select a wire or cable in the report table. A message, `Direction of wire XXX has been flipped` appears at the prompt.

2. Click **Table** > **Repeat Region** > **Update Tables** to update the table with the changed wire direction.

To Add a Break to a Wire

1. Click **Insert** > **Break**. The **SELECT** dialog box opens.
2. Select two points on a wire. A break symbol appears at the two points. The wire segment between the two selected points is deleted. If a note is present on the wire segment being deleted, the note is also deleted. Wire labels are added to wire segments so that all wire segments are labeled.
3. Click **OK** in the **SELECT** dialog box.

Note: To create leading spaces in default wire-break labels, you must enter two sets of quotation marks before the spaces, such as, `sht&wire_opp_sheet` else, Pro/DIAGRAM does not accept the syntax as input, and does not create the leading spaces in the label.

To Rename Wires

1. Select the wire label and click **Edit** > **Value**.

   or

   Select the wire label to modify and right-click to select **Edit Value** from the shortcut menu.
2. At the prompt, type a new wire label name and press ENTER.

Note: You cannot modify cable conductor names if the setup option `cond_name_from_cable` is set to `Yes`.

To Change the Spool Assigned to a Wire or Cable

1. Select the wire or cable for which you want to change the spools.
2. Click **Edit** > **Properties**.

   or

   Select the wires or cables and right-click to select **Change Spool** from the shortcut menu. The **Select Spools** dialog box opens.
3. Select a spool from the list or click **New** to define a new spool.
4. Click **OK** to update the spool parameters of the wire while not affecting the non-spool parameters of the wire.

Moving Wires

About Moving Wires

You can relocate single selected wires or multiple selected wires. When you move wires, any cable symbols associated with the wires will stretch lengthwise to include the new wire location. However, cable symbols do not change location with wires. It is best to delete them and replace them if you need to move wires that have cable symbols attached to them.

To Move a Single Wire

1. Select a line, segment, or jog. The object is highlighted.
2. Using the cursor, drag the object to a new location.
3. Left click to release it.

To Move Multiple Wires Simultaneously

You can also select individual run segments as well as highway and rail end points. All segments that are selected together will translate together and retain their shape and relative position.

1. Click **Edit** > **Transform** > **Translate**. The **SELECT** dialog box opens.
2. Select runs*, run segments**, and highway or rail endpoints in the diagram.

   Alternatively, click **Edit** > **Find**. Use the **Search Tool** dialog box to select **Wire/Cable** in the **Look for** box and click Find Now. All the wires and cables associated with the diagram are listed. Select the wires you want to move, click **Apply** and then **OK**.
3. Select multiple wires and then click **OK** in the **SELECT** dialog box. The **GET VECTOR** and **GET POINT** menus appear.
4. Define the translation vector. Click one of the following commands from the **GET VECTOR** menu:

   Horiz—Enters an offset value along the x-axis.

   Vert—Enters an offset value along the y-axis.

   Ang/Length—Enters the translation vector in polar coordinates.

   From-To—(default) Selects the end points of the translation vector.
5. Select the first point.
6. Use the **GET POINT** options to complete the move to the final destination point.

   The highway segment and wires move simultaneously, and the wires at the other end of the highway are updated to attach to the new position, as if they had been moved individually.

Note:

*Runs—Wires, cable conductors, rails, and highways.

**Run Segment—A contiguous piece of a wire, cable conductor, rail, or highway that exists between any two components, connectors, cable symbols, rails, highway ends, breaks, or a combination of these. For example, a connector and a break, two breaks, or a rail and a highway end.

Note: The way in which Pro/DIAGRAM moves wires and highways may differ depending on the **Snap to XY Axes** setting in the **Environment** dialog box, which may affect the creation of your diagram. You might have to switch the **Snap to XY Axes** setting on and off to achieve the required shape of the wire or highway you are moving.

To Reroute Wires

1. Click **Edit** > **Attachment**.
2. Select the required wire segment. The wire is unattached from the nearest pin, and activated at the cursor.
3. Sketch the selected wire segment to go to any other termination point. Middle-click to change the sheet. Left-click to terminate unattached.

To Reroute Wires to a New Symbol

Use this procedure to replace one routed connector with another in the diagram. This process also applies to component symbols.

1. Click **Insert** > **Connector** > **Single View**. The **DGM SYM TYPE** menu appears.
2. Click **Replace** and select a component or connector to be replaced.

   You can select a part of a parametric connector, an MVC, or a component group. In case of an MVC, only the selected view is replaced. You cannot select a through splice. You can select a butt splice, because these cannot be distinguished from other components and connectors after they are created.
3. Use the **GET SYMBOL** menu suboptions to identify the replacement symbol.

   Pro/DIAGRAM checks if there are any wires in the original symbol instance that do not have corresponding nodes of the same name, in the new symbol definition. Any such orphaned wires are highlighted in blue. Decide whether you want to delete them, or route them as unattached.
4. The new symbol is attached to the pointer at its origin. Click a point in the design to place it. The **Electrical Parameters** dialog box opens.
5. Update the parameters of the new component or connector, the **REF_DES** and **MODEL_NAME** are already filled in with values of the original.

   Note: If the original component or connector instance has two or more placed views, then the **MODEL_NAME** and **REF_DES** parameters are left blank in the dialog box.

6. Click **OK** in the **Electrical Parameters** dialog box when finished,
    - The old symbol and any wires that were highlighted in blue are deleted.
    - The new symbol is placed.
    - The remaining wires are rerouted to the nodes with the same name on the new symbol.

    Note: If the original symbol was the unique placed view of its component or connector, then the entire component or connector is deleted else, only one view is removed.

To Automatically Reroute From One Symbol to Another

1. Click **Edit** > **Transfer All Connections**.
2. Select the view, the entire component or connector (including parametric connectors and MVCs) or pins whose attached wires you want to reroute.
3. Select the destination view or the component, connector, or pins to which you want to reroute the wires.

    After selecting the destination component, wires in the first component are rerouted to pins with the same name on the second object. Wires terminating to a pin which has no corresponding name in the target object are not rerouted to the selected target.

Editing Wire Labels

About Editing Wire Labels

The wire label's default contents and position on the wire path are determined by the values set for the drawing setup options `def_wire_label` and `label_default_position` respectively.

You can also edit individual wire labels and attach additional wire labels as notes.

To Edit a Wire Label

1. Select a wire label and click **Edit** > **Properties**. The selected wire label is highlighted in red.

    or

    Select the wire label, right-click, and select **Properties** from the shortcut menu.

    or

    Double-click the wire label that you want to modify.

    The **Enter Text** dialog box opens. You can include system-defined parameters such as the spool name or parameters to be displayed adjacent to the wire name.

2. Edit selected notes in the window or click **Editor** to edit the notes in the Pro/DIAGRAM text editor.

   Note: The standard Pro/ENGINEER text editing tools are also available to edit the wire labels.

3. Click **Save** and then **OK**.

To Move and Reattach Wire Labels

1. Click **Edit** > **Attachment** and select the wire label to move.

   or

   Select the wire label, right-click, and select **Edit Attachment** from the shortcut menu.

   You can move it anywhere along the wire segment or attach a wire label to another segment of the same wire.

2. Select a new location where you want to place the label.

To Refresh Labels after Parameter Edit

1. Click **View** > **Update Labels**. The **REDO LABELS** menu appears.
2. In the **REDO LABELS** menu, check the objects you want to include in the refresh. When **Orientation** is checked, you can choose to **Keep Positions** or **Reset Positions**.
3. Choose note types.
4. If you want to change the position, click **Reset Positions** and then choose a selection method.
5. Select the sheets to refresh.
6. Click **Done/Return**.

To Create an Additional Wire Label

1. Click **Insert** > **Note**. The **NOTE TYPES** menu appears.
2. Click **On Item** and **Make Note** from the **NOTE TYPES** compound menu.
3. Select a wire segment to attach the note to.
4. At the prompt, type the note text. It can contain regular text, special symbols, blank spaces, wire parameters, and so on. To finish entering note text, press ENTER twice.

To Set Up Wire Label Defaults

1. Click **File** > **Properties** to set up the wire label contents and position. The **Options** dialog box opens.

2. Use the following setup options:

label_default_position	Position of labels on wire lines. Can be START, MIDDLE, END or any combination separated by commas.
label_ends_gap	Distance between nodes and labels. Numeric value
label_parallel	If YES, the wire label is always parallel to the wire, if NO, labels are always parallel to the orientation of the sheet.
def_wire_label	Sets the default string to display in the wire label.
default_wire_label_style	Sets the default wire label style, which is a user-defined text style.
default_wire_line_style	Sets the default wire line style, which is a user-defined text style or a system line style.
wire_default_increment	Defines the increment numerical value between two consecutively created wires. Used with wire default-prefix and wire-default suffix.
wire_default_prefix	Defines the text prefix of default wire names.
wire_default_suffix	Defines the start number or letter of the default wirename. If wire_default_suffix is 1, and wire_default_increment is 2, consecutive wires are named wire1, wire3, wire5, and so on. If wire_default_suffix is A, and wire_default_increment is 2, consecutive wires are named wireA, wireC, wireE, and so on. (Cannot contain underscores, dashes, or a mix of numerical and alphabetical characters.)
wire_notes_above	If Yes, the wire notes are set above the wire. If No, they intersect the wire. You can also enter a real number for exact offset from the wire.

Parameters for Wire Labels

You can include the following parameters in a wire label:

- `&wire_name`—Displays the name of the wire to which the note is attached. Upon creation, wires are assigned default names based on the diagram options. Use **Edit** > **Value** to modify the parameter value.

- `&spool_name`—Displays the name of the spool referenced by the wire. Modification of the referenced spool name is not allowed.

- `&spool_<parameter>`—Displays the value of a parameter in the wire's spool. Replace `<parameter>` with any spool parameter, for example, `&spool_color`. You cannot use **Edit** > **Value** to edit the value of a parameter. Edit the spool parameter file to modify the parameter value.

- `&wire_<parameter>`—Displays the value of a parameter in the wire. Replace `<parameter>` with any user-defined wire parameter, for example, `&wire_circuitid`. You cannot use **Edit** > **Value** to edit the value of a parameter. Edit the wire parameter file to modify the parameter value.

  The default value for `def_wire_label` is `&wire_name`.

  Note: If you change the `def_wire_label` parameter, only new wires are affected. You must change the existing wire labels using **Edit** > **Properties** to modify a single label, or use **View** > **Update Labels** to reset all labels according to the `def_wire_label` parameter.

Alternatively, use the configuration options to determine the wire label position and orientation.

Routing Across Pages

About Wire Breaks

Wire breaks let you break a wire or cable into multiple segments. This lets you sketch a wire across a number of sheets, or within the same sheet for visual clarity. Once a break point has been created, you can continue to sketch the wire at a different location, even on other sheets of the drawing. The wire break symbol is a large X.

To Switch Sheets While Routing

1. Sketch the wire from one node. Middle-click to create a wire break. You are prompted to enter a sheet number or create a new sheet.

2. Select a sheet on which to place the next piece of wire. The default sheet, when creating wires, is the current sheet.

   When creating cables, the default sheet, if one of the cable conductors has already been sketched, is the sheet containing the other end of the cable. Otherwise, the default is the current sheet.

3. Press ESC to continue sketching the wire without the break.
4. The point at which the wire continues on the new sheet is a large X. The break point X appears at the location where the wire has been terminated on the first sheet. Click on the wire to select a point at which you want the break to continue.
5. Continue sketching the wire.

To Add a Break to a Wire

1. Click **Insert** > **Break**. The **SELECT** dialog box opens.
2. Select two points on a wire. A break symbol appears at the two points. The wire segment between the two selected points is deleted. If a note is present on the wire segment being deleted, the note is also deleted. Wire labels are added to wire segments so that all wire segments are labeled.
3. Click **OK** in the **SELECT** dialog box.

Note: To create leading spaces in default wire-break labels, you must enter two sets of quotation marks before the spaces, such as, `sht&wire_opp_sheet` else, Pro/DIAGRAM does not accept the syntax as input, and does not create the leading spaces in the label.

Tip: Use Grid Settings to Control Jog Points

As you use **Make Jog**, you can set the **Snap to Grid** and **Snap to XY Axes** area checkboxes in the **Tools** > **Environment** dialog box to effect jog point placement:

- **Snap to Grid** checked and **Snap to XY Axes** cleared**—**the jog point jumps to the nearest grid intersections.
- **Snap to Grid** checked and **Snap to XY Axes** checked – if the final jog location is within a small offset of horizontal/vertical in relation to the destination point, it snaps to the horizontal/vertical position. If not, it snaps to the grid.
- **Snap to Grid** cleared and **Snap to XY Axes** checked—When the jog moves, you can sketch only horizontal or vertical segments from the previous placement point. The final segment can be a diagonal line.
- If you click **Snap to Grid** cleared and **Snap to XY Axes** cleared in the **Environment** menu, the jog point follows the cursor at any angle.

To Remove a Break from a Wire

1. Select one of the wire break symbols or the wire itself and click **Edit** > **Remove** > **Breaks**.

   or

   Select the wire that contains breaks, right-click, and select **Remove Breaks** from the shortcut menu.

   This removes all wire breaks.

2. Click **OK** in the **SELECT** dialog box. The wires whose breaks were removed are merged.

   Note: Wire breaks can be removed only if both wire break symbols appear on the same sheet. To remove breaks over a number of sheets, use the **Switch Sheet** menu option to move all wire segments and their attached components to the same sheet first.

Tip: The Wire Break Symbol Setup Option

To control the display of wire break symbols, you set the diagram setup option `break_point_size` to the default setting.

The value assigned to `break_point_size` describes one side of a square that encloses the break point in drawing units. The current default value for the break point size is ~ 0.4 inches in drawing units.

Note: Keep this value as the default value for the break point size in the diagram setup menu.

When you change the value of `break_point_size`, all break points in a diagram change to that size when you repaint the screen with any **View** command.

If you change the size of the break point, the notes associated with the breakpoint do not change.

Display of the green dot associated with the break point is controlled by the setup option that controls node radius.

- If you enter a zero or a very small number for that size value, then the break points do not display.
- If you enter a negative number for `break_point_size`, the value is automatically converted into the value `DEFAULT` and interpreted that way.

Note: If break points are not displayed, any notes associated with the breakpoints still display.

Creating Cables

About Creating and Using Cables

A cable is a specific number of conductors that will be bound together for manufacture. The cable itself has certain parameters, and each conductor within the cable has its own parameters. For example, minimum bend radius is a property of the cable, but each conductor in the cable can have a color or thickness value.

You can add wires to the design and then assign them to a cable; or, you can create the cable and add the wires to it as you route them, on the fly.

The Cable Spool

In Pro/DIAGRAM cable properties are set up and stored in spools similar to wire spools. Cable spools have an additional required parameter for the number of conductors the cable has. When you define a cable spool, you use the **Electrical Parameters** dialog box to define parameters describing the properties of each conductor.

Assigning Wires to Conductors

When you add a cable to the design, you first activate a parent spool based on the number of conductors you need in the cable.

To add a cable to the design:

1. Select wires to add to the cable, then,
2. Assign each wire you added to one of the predefined conductor numbers in the cable.

When the cable is complete, the wires reference their associated conductors in the cable spool for parameters and values, the wire spool properties no longer apply.

To Add a Cable to the Diagram

Use this procedure to add a cable and assign wires to it. If cable spools have not yet been defined in the diagram model, you must first create a cable spool.

1. Click **Insert** > **Cable**. The **Select Spools** dialog box opens.
2. Click the cable spool from the **Cable Spools** list. To create a new spool click **New**. The **ADD WIRES** menu appears.
3. Click **Pick Wire**.
4. From the diagram, select the wires to include in the new cable. You can click **Sketch Wire** or any of the other wire creation commands to create them on the fly.

   The number of wires added to the cable must be less than or equal to the number of conductors in the cable spool. When the maximum number of wires, based on the spool's `NUM_CONDUCTORS` parameter value, has been added to the cable, you receive a warning message.
5. When you have highlighted the wires you want to add, click **ADD WIRES** > **Assign Cond**. Follow the prompts to assign each selected wire to a cable conductor number. When wires become cable conductor numbers, they ignore the wire spool properties and parameters, and assume the properties listed for the conductor number in the cable spool.

   Wires selected for the cable that have not been assigned a cable conductor number are removed from the cable.
6. Click **Done** from the **ADD WIRES** menu.

On the wire label, the individual wire spool names are replaced with the cable spool name. Optionally you may add a cable symbol to the diagram.

To Change the Spool Assigned to a Wire or Cable

1. Select the wire or cable for which you want to change the spools.
2. Click **Edit** > **Properties**.

   or

   Select the wires or cables and right-click to select **Change Spool** from the shortcut menu. The **Select Spools** dialog box opens.
3. Select a spool from the list or click **New** to define a new spool.
4. Click **OK** to update the spool parameters of the wire while not affecting the non-spool parameters of the wire.

To Remove Wires from Cables

1. Click **Edit** > **Cable Contents** > **Remove Wires**. The **SELECT** dialog box opens.
2. Click the wires you want to remove and click **OK** in the dialog box.

   Note: The wires must be reassigned to wire spools. If wires in the cable are to be reassigned to different wire spools, only select those that will be assigned to one spool.
3. The **Select Spools** dialog box opens.
4. Click the wire spool to assign to the selected wires and click **OK**. The wires are removed from the cable.

Repeat the routine for wires that must be assigned to different spools. When all wires are removed from a cable, the cable and symbol are removed from the design.

To Modify a Cable Name

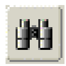
1. Select the Note filter.
2. Select the required cable name.
3. Click **Edit** > **Value**.

   or

   Select the required cable name, right-click, and select **Edit Value** from the shortcut menu.
4. At the prompt, type a new name for the cable and press ENTER. As in modifying wire names, Pro/DIAGRAM preserves any other text in the altered label.

To Delete Diagram Cables

You can delete a cable and its conductors, or delete the cable and reassign the conductors as wires.

1. Click  to select multiple cables to delete. A list of wires and cables in the diagram appears.
2. Select the name of the cable to delete. The selected cables with the wires and cable symbols are highlighted.
3. Click **Edit** > **Delete** to remove the cable and conductors.

   or

   Select all the cable conductors using the **Cable Conductor** filter, right-click and select **Delete** from the shortcut menu. This deletes the cables as well as the conductors.
4. To restore the deleted cables, click **Edit** > **Undelete** or right-click and select **Undelete** from the shortcut menu.

   Note: Deleted objects are restored only if you perform an **Undelete** operation immediately after a **Delete** operation.

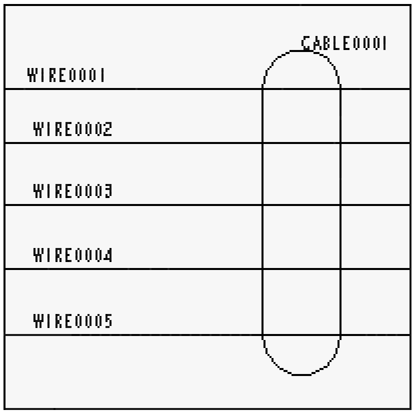
Using Cable Symbols

About Cable Symbols

A cable symbol denotes that all wires passing through it are conductors of a single cable. A note on the symbol includes the cable name. The symbol can stretch or shrink to accommodate moved conductors.

Each wire in a cable is required to pass through every cable symbol for the cable. A wire cannot belong to more than one cable. To place a cable symbol on a sheet, all wires in the cable must have at least one segment on the sheet.

All wires must go through a cable symbol in parallel. If some wires are not parallel, Pro/DIAGRAM gives you the option to let it automatically create jogs so that each wire can be included in the cable symbol. The following figure displays a cable symbol applied to five conductors.



Note: Use the **Electrical Parameters** dialog box to modify the cable symbol parameters.

Assigning Names to Cable Symbols

When a table or a neutral wirelist format output lists a cable symbol to which a wire is routed, the name assigned to the cable symbol is the exact value of the following parameters. No callouts are processed.

- `from_cable_report_name` or `to_cable_report_name` parameter of the wire attached to the cable symbol
- `from_to_cable_report_name` parameter of the wire attached to the cable symbol if the wire does not have parameters mentioned above
- `cable_jacket_report_name or cable_shield_report_name` from the cable spool parameter list if the wire does not have the `from_cable_report_name`, `to_cable_report_name`, or `from_to_cable_report_name` parameters
- The diagram setup options, `cable_jacket_report_name` or `cable_shield_report_name` when the parameters mentioned above do not exist

  Note: The cable name is assigned to the cable symbol name when the value of the setup option is `DEFAULT`.

To Add a Cable Symbol to the Diagram

1. Click **Insert** > **Cable Symbol**. The **SELECT** dialog box opens.
2. Select a wire segment in the cable to place the cable symbol. You can add a symbol to wires that have already been assigned to a cable.
3. Click **OK**.

If other wires in the cable have parallel segments, the cable symbol is created. Else, you are prompted to confirm the automatic creation of jogs (to create enough parallel segments) before the symbol is created.

Note: You can insert cable symbols only for cable conductors.

Relocating and Resizing Cable Symbols

Cable conductors move when you move the symbols that they pass through. To relocate or resize the cable symbols, select the required cable symbol and use the **Edit** > **Properties** menu. Symbols that you can resize are highlighted in red. You can also drag the cable symbols to relocate them.

To Modify the Width of a Cable Symbol

1. Select the cable symbol to resize.
2. Click **Edit** > **Properties**.

   or

   Select the cable symbol to resize, right-click, and select **Properties** from the shortcut menu.

   The **ADJUST INST** menu appears.
3. Click **Resize**.
4. Enter a value, for instance, height. Pro/DIAGRAM displays the current value as the default and rescales the symbol.
5. Click **Done**.

To Delete a Cable Symbol

1. Select the cable symbol or symbols you want to delete.
2. Click **Edit** > **Delete**.

   or

   Select the cable symbols, right-click, and select **Delete** from the shortcut menu. The selected cable symbol or symbols are deleted.
3. To restore the deleted cable symbols, click **Edit** > **Undelete** or right-click and select **Undelete** from the shortcut menu.

   Note: Deleted objects are restored only if you perform an **Undelete** operation immediately after a **Delete** operation.

Tip: Reports Using Custom Cable Parameters

You can create user-defined parameters for specific cable conductors within a cable. These parameters can then be displayed in the report tables. For example, when you specify the end preparation for an individual wire based on the terminator that is applied to it, you can capture the parameter information by creating user-defined parameters.

Use **Tools** > **Logical Reference** to pass the values between the diagram and the 3D harness.

Tip: Representing Connections to Shielding

Wires and cable conductors can be routed up to the arc ends of cable symbols to indicate an electrical connection to the shielding. If a wire is routed to a cable symbol end, a node shows at the midpoint of the arc at that end of the symbol, and the wire is attached to the node. One or both ends of the cable symbol can be routed to and have a node. The node display is controlled with the `node_radius` **Drawing** setup option.

The symbol does not have any parameters that give it a name. However, the setup option `cable_shield_report_name` is used to specify a name to show in report tables as the From-To for wires routed from and to it.

New optional cable spool parameters and wire parameters can be used to override the diagram-wide names specified in the setup option.

Adding Splices

About Splices

You can splice wires together using the splice function. You can create two kinds of splices:

- Butt splices—the wire is split into two separate wires and a one-pin component (the butt splice) is placed at the point of separation.
- Through splices—the one-pin component (the through splice) is placed on the wire without splitting the wire. Additional wires can then be routed to the splice.

  Note: Before you create a splice, you must create a symbol with a single node.

When you create or move a splice, note that:

- A butt splice splits the wire it is placed on into two separate wires and does not move when the wire it is placed on is moved.
- A through splice does not split the wire it is placed on and moves with it.

  Note: A component or connector that is being placed can also be inserted into an existing wire as a splice.

To Create a Butt Splice

1. Click **Insert** > **Splice** > **Butt**. The **SELECT** dialog box opens.
2. Select the wire to which you want to splice a segment and click **OK** in the **SELECT** dialog box. The splice is created at the point you select on the wire. The **GET SYMBOL** menu appears.
3. Use the **GET SYMBOL** menu to select the single node symbol to place at the splice. Use one of the following options:
   - **Name**—Select from a list of in-session diagram symbols by name.
   - **Pick Inst**—Select a symbol currently on the diagram.

- o **Retrieve**—Browse to a directory containing the symbol.

4. When you select the symbol it is placed at the splice point. The wire is split and the second half of the wire is given a new name. The component created is a regular diagram component.

   You can now route additional wires to the butt splice.

To Create a Through Splice

1. Click **Insert** > **Splice** > **Through**. The **SELECT** dialog box opens.
2. Select the wire to which you want to splice a segment and click **OK** in the **SELECT** dialog box. The splice is created at the point you select on the wire. The **GET SYMBOL** menu appears.
3. Use the **GET SYMBOL** menu to select the single node symbol to represent the splice. Use one of the following options:
   - o **Name**—Select from a list of in-session diagram symbols by name.
   - o **Pick Inst**—Select a symbol currently on the diagram.
   - o **Retrieve**—Browse to a directory containing the symbol.
4. When you select the symbol it is placed at the splice point. The component created is a regular diagram component.

   You can now route additional wires to the through splice.

Using Power and Ground Rails

About Power and Ground Rails

Diagram rails are entities used to represent power or ground circuits. Rails are created in a manner similar to highways.

Rails can be attached to components, connectors, other rails, or can be left unattached. Multiple wires can be attached to rails.

Unlike highways, rails can contain parameters. The only required parameter is `NAME`. All other parameters are user-defined and can be modified using the same method as the one used to modify wire parameters. Like highways, rails can be broken into several views.

Rail Display Setup Options

Click **File** > **Properties**. Several drawing setup options that control rail display follow:

`def_rail_label`	Sets the default contents of the rail label.
`default_rail_label_style`	Sets the default rail label style, which can be any valid named text style that is user-

	defined.
`default_rail_line_style`	Sets the default rail line style, which is a user-defined text style or a system line style.
`rail_default_increment`	Defines the increment numerical value between two consecutively created rails. Used with rail default-prefix and rail-default suffix.
`rail_default_prefix`	Defines the text prefix of default rail names.
`rail_default_suffix`	Defines the start number or letter of the default railname. If `rail_default_suffix` is `1`, and `rail_default_increment` is `2`, consecutive rails are named `rail1, rail3, rail5`, and so on. If `rail_default_suffix` is `A`, and `rail_default_increment` is `2`, consecutive rails are named `railA, railC, railE` and so on. (Cannot contain underscores, dashes, or a mix of numerical and alphabetical characters.)
`rail_node_radius`	Sets the size of the nodes displayed at attachment points between two rails or between a wire and a rail for printing and display.
`rail_notes_above`	Places the rail notes a specified real number above the highway. The default is `0.1`. Enter `NO` to center the label on the rail.

To Create a Rail

1. Click **Insert** > **Rail**. The **CREATE RAIL** menu appears.
2. Click **Sketch Path** to draw the rail. Left-click to sketch a path. Middle-click to create a break in the rail. Right- click to complete the sketch at the last left click point.

   or

   Click **Follow Path** to draw the rail parallel to the path of the object selected.

   or

   Click **Use Wire** to replace a segment of wire with a rail, through which the wire is routed.

3. When the rail is compete, click **Done/Return**.

Assigning Rails to a Default Layer

To assign rails to a default layer, set the configuration file option:

```
def_layer layer_dgm_rail <layer_name>
```

where `<layer_name>` is the name of the default layer for rails.

Note: The default layer can also be set using the **SetDefLayer** option in the **Layers** menu.

Using Highways

About Highways

Highways let you simplify a schematic visually by showing the path of multiple wires as a single line. When you include a wire in a highway, it affects only the graphic display of its path.

To create highways you can:

- Sketch the highway directly on the drawing, or
- Use an existing wire path.

To add wires to a highway you can:

- Route a wire to a highway as the wire is being created, or,
- Reroute an existing wire to a highway.

To Add a Highway

1. Click **Insert** > **Highway**. The **CREATE HWAY** menu appears.
2. Click **Sketch** to draw the rail. Left-click to sketch the path. Middle-click to create a break in the rail. Right-click to complete the sketch at the last left click point.

   OR

   Click **Follow Path** to draw the highway parallel to the path of the object selected.

   OR

   Click **Use Wire** to replace a segment of wire with a highway, through which the wire is routed.
3. When the rail is compete, click **Done/Return**.

To Add Routed Wires to an Existing Highway

1. Click **Edit** > **Highway Wires** > **Add Wires**.

2. Select the highway to which you want to add wires.
3. Select a point on each wire where you want to begin re-routing. You can now route from this point to the destination highway.
4. Select a point on each wire where you want it to exit from the highway. Select this point and re-route the wire to the highway.

   Wire labels are added so that all portions have a wire label. These labels can be deleted later, if required.

Note: The **Add Wires** option remains active until another option is selected. This allows you to continue adding wires to an existing highway.

To Add a Wire to a Highway While Routing

1. Sketch a wire. While sketching the wire, select a point on the highway where you want the wire to enter.
2. Pro/DIAGRAM informs you that a wire has entered the highway and prompts you to select an exit point.
3. Select a point on the highway where you want the wire to exit and continue sketching the wire.

To Remove Wires from a Highway

1. Click **Edit** > **Highway Wires** > **Remove Wires**.
2. Select the highway from which you want to remove wires.
3. Select the wire or wires you want to remove. Use the **Search Tool** dialog box to choose the wires or multiple wires, you want to remove.
4. Click **Apply** and then **OK** in the **Search Tool** dialog box.

Note: When a wire is removed from a highway, the wire's path remains the same as the highway it was removed from. You can move the wire away from the highway by selecting the appropriate wire to move.

To Modify the Highway Entry or Exit Point

1. Click **Edit** > **Attachment**.
2. Click  or **Edit** > **Find**. Use the **Search Tool** dialog box to select the wire whose entry or exit point you want to move. Select the wire segment that is directly attached to the highway.
3. Route the wire to a new attachment point.

To Modify or Delete Highway Names

1. Select the Note filter.

2. Select the highway name to modify and click **Edit** > **Value**. At the prompt, type a new name for the highway and press ENTER.

   or

   Select the highway name, right-click, and select **Edit Value** from the shortcut menu.

3. Select the highway name you want to remove and click **Edit** > **Delete** to delete the highway name.

   or

   Select the highway name, right-click, and select **Delete** from the shortcut menu.

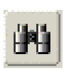
Note: Use the setup configuration option `def_highway_label`.

To Reroute a Highway

1. Click **Edit** > **Attachment**.
2. Select the end segment to reroute and reroute the highway.

To Delete a Highway

A highway can be deleted like a wire in a diagram. When a highway is deleted, all notes attached to the highway are deleted as well.

1. Click  to select a single highway or multiple highways to delete. A list of highways in the diagram appears.
2. Select the highway or highways you want to delete and click **OK**. Ensure that the selected highways appear on the same sheet.

   Highways and wires can be deleted simultaneously.

3. If the selected highway has any wires attached to it, the **DEL OPTIONS** menu appears with the following options:
   - **Delete**—Removes the highway and all attached wires and cables.
   - **Explode**—Deletes the highway, but leaves all attached wires and cables in place. Select and drag to separate the remaining wires and cables.
4. To restore the deleted highway, click **Edit** > **Undelete** or right-click and select **Undelete** from the shortcut menu.

   Note: Deleted objects are restored only if you perform an **Undelete** operation immediately after a **Delete** operation.

Alternatively, you can also select the highway or highways to delete and right click to select **Delete** from the shortcut menu.

To Set Up the Highway Label

To set up the highway label contents and position:

1. Click **File** > **Properties**. The **Options** dialog box opens.
2. Use the following setup options:

`def_highway_label`	Defines the highway label contents. This sets the highway label name. The default is no label. Enter `&hway_name` as a value to show the name; with the assigned prefix, suffix, and increment.
`highway_notes_above`	Places the highway notes a specified integer value above the highway. The default is `.01`. Enter `NO` to center the label on the highway.
`highway_default_suffix`	Defines the start number or letter of the highway name that increments as you add each new highway. The value must be either numerical or alphabetical, and cannot contain underscores, dashes, or a mix of numerical and alphabetical characters. The default value is `0001`.
`highway_default_prefix`	Defines the prefix of highway names. The default value is `HIGHWAY`.
`highway_default_increment`	Defines the increment from `name_suffix` between two consecutively created highways. This value must be numerical. The default value is `1`.
`default_highway_label_style`	Defines the default highway label style. This is a user-defined text style.
`default_highway_line_style`	Any valid line style can be specified including geometry, hidden, leader, cut plane, phantom, and centerline

Using Ladder Diagrams

About Ladder Diagrams

You can create ladder diagrams using the reference zone and cross-reference note functions. When creating a ladder diagram, you are prompted to select an existing reference zone. Rails are created along each side of the reference zone. Wires are created between the rails, along each of the division lines. Rails and wires can also be created manually.

There are four steps to creating a ladder diagram:

1. Create a reference zone.
2. Use the ladder diagram functionality to create the rails and wires that represent the ladder diagram.
3. Create, or retrieve, and place the appropriate single and multiple-view components on the wires in your ladder diagram.
4. Create and place the appropriate rung labels for your ladder diagram. Each rung (division line) of a ladder diagram can be labeled with a rung label. Rung labels can contain cross reference notes which, for each view on the rung, identify the physical position of other views of multiple view components.

After you create a reference zone, you can add and remove division lines from the reference zone. You can also modify the following attributes:

- The spacing between the division lines
- The numbering scheme used for the division lines
- The reference zone's boundary
- A division line's index
- The position of the index labels

To Create a Reference Zone

1. Click **Format** > **Reference Zones**. The **REF ZONES** menu appears.
2. Click **Create** to create a reference zone. You are asked to specify a rectangular area for your reference zone by selecting corner points. If this rectangular area overlaps an existing reference zone, the reference zone you create is truncated. The **ZONE DIR** menu appears.
3. Click **Horizontal** in the **ZONE DIR** menu if you want the division lines in this reference zone to be displayed horizontally. Click **Vertical** if you want the division lines to be displayed vertically.
4. Enter the new grid spacing of the division lines, a starting index, and a delta (index difference between the lines). The starting index can be a negative

number. Horizontal division lines are used with a default grid spacing of 0.25, a starting index of 10, and a delta index of 10. The **ZONE SETUP** menu appears.

5. Select an appropriate command from the **ZONE SETUP** menu.

   - **Mod Boundary**—Click a new area of your drawing to use as a reference zone. Division lines which no longer fit in the area you have specified are deleted. New division lines are not created if the new area is larger than the old area.
   - **Spacing**—Enter a new spacing for the division lines in your reference zone. This value should be entered in drawing units.
   - **Renumber**—Specify a new starting index and a new delta. The reference zone is modified accordingly. The starting index can be a negative number.
   - **Insert**—Add a division line to your reference zone. When selected, you are prompted to select a point in your reference zone where you want the division line to be placed and to enter an index for this division line. A new division line is placed parallel to existing division lines at the location you specified. This option does not effect the current numbering of the division lines.
   - **Remove**—Deletes an existing division line and its index. This option does not effect the current numbering of the division lines.
   - **Mod Index**— Lets you modify the index of one division line. When selected, you are asked to choose the division line whose index you want to modify. You are then prompted to enter a new index. This option does not effect the current numbering of the division lines.
   - **Label Pos**—Change the position of all the labels in the reference zone. When selected, the **Offset Side** menu appears, allowing you to indicate the side of the reference zone where you want the index labels to appear and the index label's distance from the edge of the zone.

   If the reference zone has vertical division lines, the **Offset Side** menu has the following options:

   - **Top**—Place all index labels on top of the reference zone.
   - **Bottom**—Place all index labels on the bottom of the reference zone.

   If the reference zone has horizontal division lines, the **Offset Side** menu has the following options:

   - **Left**—Place all index labels to the left of the reference zone.
   - **Right**—Place all index labels to the right of the reference zone.

6. Click **Done/Return** first in the **ZONE DIR** menu and then in the **REF ZONES** menu.

Note: Check the **Zone Labels** and **Zone Grids** options in the **ENVIRONMENT** dialog box to see the reference zone.

To Create a Ladder Diagram

A reference zone must exist before you can create a ladder diagram. Also, a spool must be active. If none is active you are prompted to select one.

1. Create a reference zone. Use this reference zone to create the rails and wires to make up the ladder diagram.
2. Click **Insert** > **Ladder**. The **CR LADDER** menu appears.
3. Click **Make Ladder**. The **Select Spools** dialog box opens.
4. Select the required spool and click **OK**.
5. Select an existing reference zone that you have created. Wires are created over the horizontal division lines in your reference zone and rails are created over the vertical boundary lines in your reference zone, depending on the type of division lines you have selected for the reference zone.

   Note: Check the **Zone Labels** and **Zone Grids** options in the **ENVIRONMENT** dialog box to see the reference zone.
6. If rung label templates are available, you can choose a rung label template to use as the default rung label for your ladder diagram.
7. Click **Done/Return** in the **CR LADDER** menu.

To Create a Rung Label

1. Click **Format** > **Rung Labels** to create, delete, or edit rung labels. The **RUNG LABEL** menu appears.
2. Click **Template**. The **RUNG TEMPL** menu appears. If you chose to create or modify a template or create a new rung label, you are prompted to specify a name for the rung label template. The **MOD RUNG LBL** menu appears. This menu lets you edit each type of note that appears in the rung label and specify the views of a multiple view component. You must consider the views of a multiple view component during evaluation of the label.
3. Select an appropriate command from the **MOD RUNG LBL** menu and click **Done/Return** in the **RUNG TEMPL** menu.
4. If the rung labels in your ladder diagram need to be updated, click **Update** on the **RUNG LABEL** menu.

Reference Zone Display Options

Use the following two **Environment** dialog box options (**Tools** > **Environment**) to control the display of reference zones in ladder diagrams:

- **Ref Zone Labels**—The labels of reference zone division lines are displayed and printed.
- **Ref Zone Lines**—The grid lines of the reference zone are displayed and printed.

Using Terminator Tables

About Terminator Tables in Diagrams

ProCmdDgmTermTable@CMD,auto_param_table@DLG

ProCmdDgmTermTable@CMD,auto_param_table@DLG

ProCmdDgmTermTable@CMD,auto_param_table@DLG

ProCmdDgmTermTable@CMD,auto_param_table@DLG

ProCmdDgmTermTable@CMD,auto_param_table@DLG,ProCmdDgmElecParams@CMD

ProCmdInfoDgmBom@CMD,ProCmdInfoDgmBomLayer@CMD

ProCmdInfoDgmBom@CMD,ProCmdInfoDgmBomLayer@CMD

ProCmdInfoDgmWirelist@CMD,ProCmdInfoDgmWlLayer@CMD$$$

1. Click **Info**.
2. Click **Wire List** to generate a list for all layers.

   or

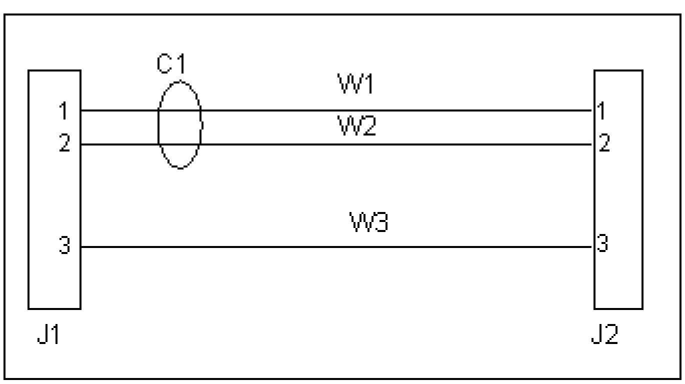
   Click **Wire List for Layer** to generate a list of wires for a selected layer and select a layer from the **Layer Sel** menu.

   The wire list is generated and displayed in the INFORMATION WINDOW. The wire list is written to a file, `dgm_wirelist.inf.#`, where `#` is the version number in your current working directory.
3. Click **Close** to close the INFORMATION WINDOW.

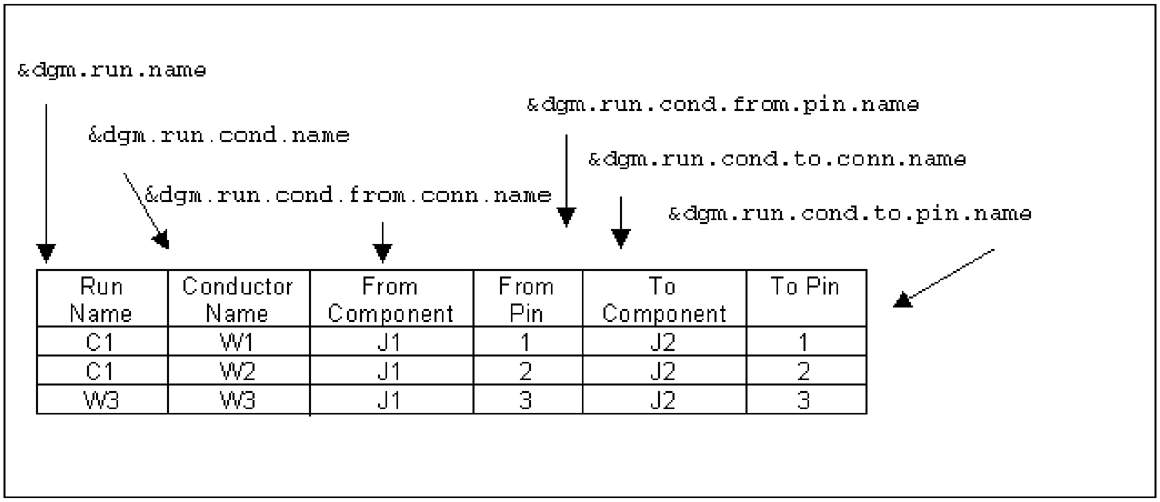
Pro/REPORT Table Reports

Using REPORT Tables in Pro/DIAGRAM

Include customized wire lists in your diagrams using the Pro/REPORT table functionality. The following example shows how to enter repeat regions in a table to produce a from-to wirelist for a simple connection:



Use the table editor to enter repeat regions as shown. The table shows the output of each repeat region:



Run Name	Conductor Name	From Component	From Pin	To Component	To Pin
C1	W1	J1	1	J2	1
C1	W2	J1	2	J2	2
W3	W3	J1	3	J2	3

A cable listed in the report table shows From-To information only if all cable conductors at one or both ends of the cable go to the same ref_des.

- If the conductors go to different connectors, the report table entry for that end is "-", to show that the value has no meaning.
- If the conductors go to the same connector, the connector information is shown appropriately.

For more information on report tables see the *Pro/ENGINEER Drawing Guide*.

To Modify Wire Names with Pro/REPORT

The name of a wire or cable can be modified through a Pro/REPORT table using the following parameters:

- `&dgm.run.name`
- `&dgm.run.cond.name`
- `&dgm.spool.run.name`
- `&dgm.spool.run.cond.name`.

Pro/REPORT Parameters for Pro/DIAGRAM

There are four different groups of diagram parameters:

- Parameters that begin with `&dgm.run` are used to create wire lists.
- Parameters that begin with `&dgm.spool` are used to create tables listing information by spool.
- Parameters that begin with `&dgm.sym` are used to create BOM tables.
- Parameters that begin with `&dgm.layer` isolate items on selected layers.

Note: If you mix two parameters from different groups in the same repeat region, two sets of reports are created in the table, one located below the other.

A list of system parameters that pertain to Pro/DIAGRAM appear in the following table. Parameters specific to Diagram models also appear in this table. If you want to enter report symbols into a table after defining a repeat region, click **Edit** > **Properties** > **Enter Text** > **Report Sym** command or double-click on the table cells.

BOM and Family Table Parameters	
`&rpt.index`	Displays the number assigned to each record in a repeat region.
`&rpt.level`	Shows the recursive depth of an item.
`&rpt.qty`	Displays the quantity of an item.
Layer Parameters	
`dgm.layer.name`	The layer name.
`dgm.layer.sym`	Duplicates the heirarchy under `dgm.sym`. Only evaluates to symbols on the current layer.
`dgm.layer.run`	Duplicates the heirarchy under `dgm.run`. Only evaluates to runs on the current layer.
Diagram Parameters	
`&dgm.run.cond.color`	Lists conductor color defined in the spool parameter file for wires belonging to cables. Lists color defined in the spool parameter file for individual wires.
`&dgm.run.cond.from/to.conn.name`	Lists the reference designators of the component/connecto

	rs or the name of the rail that wires (free wires or wires belonging to cables) are run to/from.
`&dgm.run.cond.from/to.conn.<User_Defined>`	Lists specified parameters for the component/connectors or rails that wires (free wires or wires belonging to cables) are run to/from. Any parameter of the connector or rail can be entered manually.
`&dgm.run.cond.from/to.pin.name`	Lists the pin names of the connectors that the wires (except for free wires) are run to/from.
`&dgm.run.cond.from/to.pin.entry_port`	Lists the entry port of each pin that conductors are run from or to.
`&dgm.run.cond.from/to.pin.signal.name`	Lists the pin signal names of the connectors that the wires (free wires or wires belonging to cables) are run to/from.
`&dgm.run.cond.from/to.pin.signal.value`	Lists the pin signal values of the connectors that the wires (free wires or wires belonging to cables) are run to/from.
`&dgm.run.cond.from/to.pin.<User_Defined>`	Lists any user-defined parameter that you enter for each pin that conductors are run

	from or to.
&dgm.run.cond.from/to.sheet	Lists the sheet number on which the wires (free wires or wires belonging to cables) start or terminate.
&dgm.run.cond.name	Lists the assigned conductor name for wires (free wires or wires belonging to cables) in a diagram.
&dgm.run.cond.User Defined	Lists user-defined conductor parameter described in the spool parameter file for wires belonging to cables. Lists user-defined parameter described in the spool parameter file for individual wires.
&dgm.run.from/to.conn.name	Lists the reference designators of the connector or the name of the rail that cables and free wires are connected from/to.
&dgm.run.from/to.conn.<User_Defined>	Lists any parameters defined in the connector or rail that you enter manually.
&dgm. run. name	Lists any wire or cable in the diagram.
&dgm.run.param.<User_Defined>	Lists any parameter that you enter manually as a user defined parameter for diagram wires or cables.
&dgm.run.spool.name	Lists the wire and cable spools name

	parameter run.
`&dgm.run.spool.<User_Defined>`	Lists any user defined parameter for wire and cable run.
`&dgm.spool.name`	Lists the spool name.
`&dgm.spool.<User_Defined>`	Lists any spool parameter that you enter manually as a user defined parameter.
`&dgm.spool.run.cond.color`	Lists conductor color parameter for wires in cable and wire spool color for independent wires.
`&dgm.spool.run.cond.from/to.conn.name`	Lists the name of each connector or rail that every conductor is run from/to.
`&dgm.spool.run.cond.from/to.conn.<User_Defined>`	Lists the specified parameter for each connector or rail that every conductor is run from/to that you enter manually.
`&dgm.spool.run.cond.from/to.pin.name`	Lists the pin names that conductors are run from/to for every spool.
`&dgm.spool.run.cond.from/to.pin.signal.name`	Lists the pin signal names that conductors are run from/to for every spool.
`&dgm.spool.run.cond.from/to.pin.<User_Defined>`	Lists the specified user-defined parameter for each pin that conductors are run from or to for

	every spool.
&dgm.spool.run.cond.from/to.pin_entry_port	Lists the entry port of each pin that conductors are run from or to for every spool.
&dgm.spool.run.cond.from/to.pin.signal.value	Lists the pin signal values that conductors are run from/to for every spool.
&dgm.spool.run.cond.from/to.sheet	Lists the sheet number on which the wire starts or terminates.
&dgm.spool.run.cond.name	Lists the name of the wire in the diagram for each spool.
&dgm.spool.run.cond.<User_Defined>	Lists the wire and cable specified parameters by spool.
&dgm.spool.run.from/to.conn.name	Lists the connector or rail names that conductors are run from/to by spool.
&dgm.spool.run.from/to.conn.<User_Defined>	Lists the connectors or rails parameters that conductors are run from.
&dgm.spool.run.name	Lists the spool names for each wire or cable in the model.
&dgm.sym.node.name	Lists the name of the given symbol node.
&dgm.sym.node.sheet	Lists the sheet number on which a given node is located.
&dgm.sym.node.<User_Defined>	Lists a pin parameter such as signal_name

	for each connector.
&dgm.sym.<User_Defined>	Lists a parameter for a component or connector symbol.
&dgm.sym.ref_des	Lists the reference designator name for a component or connector.

An item's parameters can also be used in a Pro/REPORT table as a user-defined parameter.

The `MODEL_NAME` parameter of a connector can be used as a user-defined parameter in a Pro/REPORT table as &dgm.run.from.conn.model_UserDefined and entered as model_name. This is possible even though a system-defined Pro/REPORT parameter called `&dgm.run.from.conn.model_name` does not exist.

When creating a report, it is important to consider the structuring of the symbols in the report. Pro/REPORT provides the definition of each symbol based upon the item or items to the left of the description.

The table below lists the names of all sketched wires and the name of the associated spool.

&dgm.run.name	&dgm.run.spool.name
W21	Spool32
W33	Spool18W

Pro/REPORT provides information based upon a hierarchy of specifications.

In the example above, the parameter `&dgm.run.name` gives the name of all the runs (cables and individual wires) in the harness. The spool name, `&dgm.run.spool.name`, gives the name of the appropriate spool for each run in the harness.

The next table illustrates the slight differences between similar parameters.

&dgm.spool.name	&dgm.spool.run.name
Spool32	W21
Spool18W	W33
Spool12	

In this case, the repeat region parameter `&dgm.spool.name` lists all spools in the diagram, whether or not that spool has an associated run. Because Spool12 has no associated wire, it appears in the second table.

Report Parameters for Connections

The following is a complete list of report parameters that work for conductor names, connectors names, pin names, and the sheet to which the conductor is routed.

```
&dgm.run.from/to.conn.name
&dgm.run.cond.from/to.conn.name
&dgm.spool.run.from/to.conn.name
&dgm.spool.run.cond.from/to.conn.name
&dgm.run.cond.from/to.pin.name
&dgm.spool.run.cond.from/to.pin.name
&dgm.run.cond.from/to.sheet
&dgm.spool.run.cond.from/to.sheet
```

Diagram and the pos_loc callout

In Pro/DIAGRAM the &pos_loc callout recognizes the following types.

View	<view name>
parent_note	<view name>
ref_obj	<reference designator>
routed_from	<diagram connection name>
routed_to	<diagram connection name>

For example, to show the location of a view named main_view, type the following in the note text box:

```
Main view is located in &pos_loc:view:main_view
```

Accessing Conductor Parameters in Pro/REPORT

You can access conductor parameters using the following Pro/REPORT parameters:

- dgm.run.cond.<User Defined>—This parameter searches first in the conductor of the individual cable for the specified parameter. If the parameter is not found in the conductor, it searches in the spool conductor parameters for that parameter.
- dgm.run.cond.spool.<User Defined>—This parameter searches in the spool conductor parameters for that parameter.

Glossary

Glossary of Terms

Term	Definition
Components	Components are schematic drawings of electromechanical parts or sub-assemblies. A component has a graphical symbol drawing and a set of defining parameters, such as name, type, and reference designator. Each component also contains a set of predefined attach points called nodes. You can retrieve components either from user-defined libraries, or, if you have a Pro/LIBRARYACCESS license, from the Pro/ENGINEER library of electrical symbols.
Connectors	Pro/DIAGRAM connectors are schematic drawings of physical connectors. The display of the connector can be either through a fixed shape symbol (fixed connector) or through a parametric definition where a standard connector shape is determined from specified parameters (parametric connector). Parameters include the number of pins to show, pin sequence, gender, partial or full display, size, and orientation.
Pins (Nodes)	Valid wire attach points on components and connectors are called nodes or pins. A pin must have a unique ID within the component/connector symbol. A pin can contain parameters that define which conductors are connected to it. Pins display as green dots in the symbol.
Wires	Wires are used to connect nodes on connectors and components. Wires may be routed between different sheets or as unattached. Wire paths

	update automatically when the components and connectors are moved. You can also modify the wire paths by moving wire segments or vertices.
Cables	Cables are multiple wires, intended to be routed together. Cables can contain any number of wires. Cables can be distinguished from wires with a symbol that surrounds the wires that the cable contains.
Conductors	Conductors are schematic representations of physical conductors. They can be individual wires, or wires that are added to a cable. Conductors in cables are the individual wires of varying properties that are added to the cable in the spool definition.
Spools	All wires and cables are added to the diagram from predefined wire or cable spools. Properties defined in the spool such as thickness, color, min bend radius and so on are passed to the wire or cable when it is added to the design. The properties may then be modified for each cable or wire as necessary. Cable spools have values for the external appearance and overall minimum bend radius, as well as values associated with each of the assigned 'conductors' within the cable.
Highways	Highways are paths in a Pro/DIAGRAM drawing in which any number of wires or cables can be routed. This simplifies the final drawing by allowing multiple wires to be represented graphically by a single path. This option is for display purposes only. It has no physical significance.
Rails	Rails are used to represent ground or power busses or conductive strips. You can route rails between

	components, connectors, or other rails, or leave the ends unattached.
Splices	The Splice functionality lets you splice wires together. Two types of splices can be created: a butt splice and a through splice. When a butt splice is placed on a wire, the wire is split into two separate wires with different reference designations. When a through splice is placed on a wire, the wire is not split. Once a splice has been placed, additional wires can be routed to the splice.

Index