Loop Drawings Module

Overview

You use the Loop Drawings module to retrieve predefined instrumentation data and generate appropriate loop drawings. Each component in the loop drawing is assigned the necessary drawing blocks that represent its functionality and wiring routing. The drawings display information about the loops and their instruments, wiring routing, line data, DCS data, document references, approvals, revisions, and general information.

The Loop Drawings module generates loop drawings by means of a fast build process. By configuring typical blocks for key instrument types, the loop components can be automatically initialized with their corresponding graphical elements. The Loop Drawings module enables you to create loop drawings that include data retrieved from the database and if required, from an external file.

You can generate loop drawings one of the following ways:

- Using the Enhanced Report Utility (for details, see Enhanced Report Utility: An Overview).

- Using the built-in standard SmartLoop drawing generator (for details, see Generating SmartLoop Drawings).

- Using an external CAD application (SmartPlant Instrumentation supports Intergraph SmartSketch, Autodesk AutoCAD (2000 or later), or Bentley Systems MicroStation for Windows). For details, see Generating a Loop Drawing Using an External CAD Engine.

Starting the Loop Drawings Module

The following procedure explains how to start the Loop Drawings module where you configure and generate drawings that display your instrumentation loops.

 Note

- Before starting this module, verify that the Domain Administrator has granted you the appropriate access rights for the tasks that you are about to perform.

➢ **To start the Loop Drawings module**

1. Do one of the following:

    - On the main toolbar, click .

    - On the **Modules** menu, click **Loops Drawings**.

 Tip

- You can maintain several modules operative simultaneously. The number of concurrently operative modules depends on your computer resources.

Loop Drawing Generation Methods

Overview

SmartPlant Instrumentation allows you to assign a loop number to one of the following generation methods: CAD, SmartLoop, Enhanced SmartLoop, and manual. Methods are marked by designated symbols attached to loop numbers in the **Loops** folder of the **Domain Explorer**, and in the **Loop Explorer**, which is only accessible from the Loop Drawings module. By default, loop numbers are not assigned to any generation method.

You can either assign each loop number to a specific method or apply the same method to all loop numbers when generating loop drawings in batch mode. For detailed information on each method and the associated symbol, see Setting a Loop Drawing Generation Method Symbol.

Loop drawing generation is traditionally done using a CAD application such as SmartSketch, AutoCAD, or MicroStation. You must specify the CAD application paths and other CAD file locations in the **Preferences** dialog box under the Loop Drawings module options.

In addition to using a CAD method, you can also use one of the two built-in utilities that enable you to generate, view, modify, and annotate loop drawings as needed.

The utilities are:

- The Enhanced Report Utility, used to generate enhanced SmartLoop drawings and reports.

- The SmartLoop Generator, used to generate standard SmartLoop drawings.

 Notes

- We strongly recommend using the Enhanced Report Utility over the SmartLoop Generator for creating your loop drawings because it uses better technology and it includes a wider set of features than the SmartLoop Generator. Furthermore, there is ongoing software development and support for Enhanced Report Utility. The SmartLoop Generator is intended for users who have already used this method of loop drawing generation in earlier versions of the software.

- The Enhanced Report Utility has an advantage over CAD as a loop drawing generation method in that it requires no extra work in creating CAD drawing blocks. The report comes directly from the database and does not necessarily require additional modifications.

Setting a Loop Drawing Generation Method Symbol

You can annotate loops in the **Domain Explorer** (or in the **Loop Explorer**, which is only accessible from the Loop Drawings module) with an appropriate generation symbol for the intended method of generation.

The available annotation symbols are:

- C — Loop drawing to be generated by a CAD application (SmartSketch, AutoCAD, or MicroStation).

- M — Loop drawing to be generated manually, without using SmartPlant Instrumentation options. When generating loop drawings, the software always skips loops assigned to the Manual method.

- For loop drawings where the selected generation method is Enhanced SmartLoop, the following symbols are available:

  - E — Per Loop

  - ES — Per Signal

  - EC — By Custom Symbol

- For loop drawings where the selected generation method is SmartLoop, the following symbols are available:

  - SC — By Category

  - SL — By Location

  - LS — By Location with Setting

Notes

- The loop generation symbol is a reference only, and does not place any restrictions on the drawing generation method. However, the selected SmartLoop symbols affect the way the SmartLoop drawing is displayed. See SmartLoop Display Settings for further explanation.

- If you define a setting, other than the default, for a loop drawing, the drawing generation will be according to the setting, regardless of the loop generation symbol you selected.

- If you assign a loop with the symbol C, M, or no symbol, and you do not attach any setting to the loop, drawing generation is according to the parameter you define in the INTOOLS.INI file, under the SmartLoop section, as follows:

  ```
  DefaultMethod = 0 – By Category
  DefaultMethod = 1 – By Location
  ```

➢ **To select the loop drawing generation method symbol**

1. Do one of the following:

   - Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.

   - On the Loop Drawings module toolbar, click ⬛ to open the **Loop Explorer**.

2. Under **Entity Name**, select and right-click the desired loop numbers and on the shortcut menu, click **Apply Generation Method**.

3. Click one of the available options to apply the generation method.

   💡 **Tip**

   - To clear the current loop generation method for one or more loops, on the shortcut menu, click **Clear Generation Method**.

SmartLoop Drawings

Overview

The SmartLoop Generator is a built-in SmartPlant Instrumentation feature that allows you to generate loop drawings.
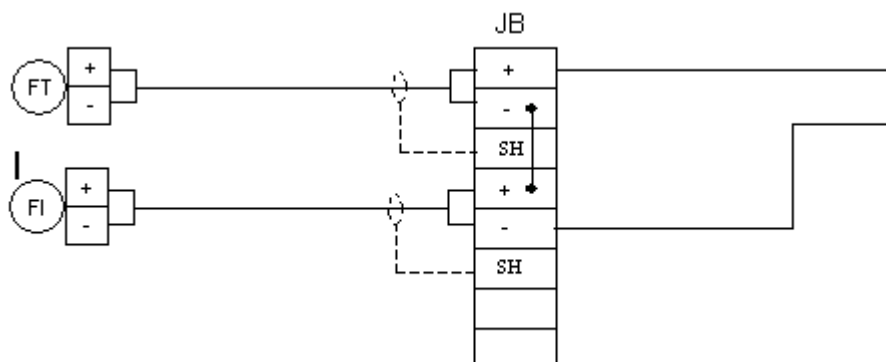
📝 **Note**

- We strongly recommend using Enhanced Report Utility over the SmartLoop Generator for creating your loop drawings because it uses better technology and it includes a wider set of features than the SmartLoop Generator. Furthermore, there is ongoing software development and support for Enhanced Report Utility. The SmartLoop Generator is intended for users who have already used this method of loop drawing generation in earlier versions of the software.

The SmartLoop Generator can generate drawings for most standard loop-configurations enabled by CAD applications. The following loop configurations are supported in addition to the common loop configurations (a series of two wires and terminals connected from a field device to a DCS):
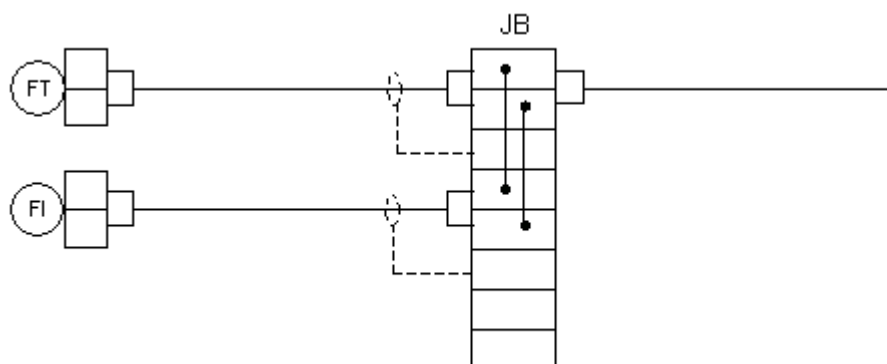
- Transmitters and their indicators connected in series and in parallel

- Temperature element and transmitter connected together

- Instruments with more than one terminal strip

- Manufacturer's terminology for DCS

- Overall shield wires

- Jumpers

SmartLoop Drawing Examples

### Flow Transmitters with Associated Field Loop Indicators in Series (including Jumpers)

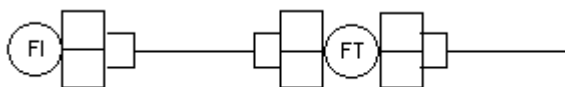### Flow Transmitters with Associated Field Loop Indicators in Parallel (including Jumpers)

Note

- In the above examples, the FT and FI do not need to be connected to consecutive terminals.
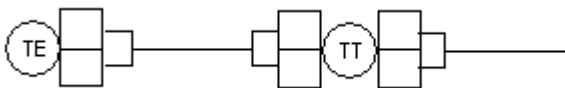
Flow Transmitters with Associated Field Loop Indicators in Parallel (alternative configuration)

JB
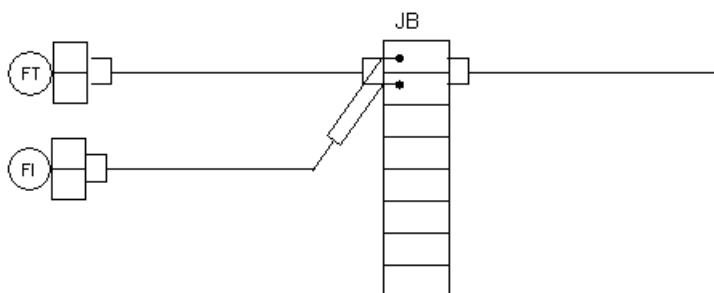
Flow Indicator and Transmitter Connected Together

Temperature Element and Transmitter Connected Together

Instruments with more than one Terminal Strip

Device Panel

Analog signal

Digital signal

Power supply 220V
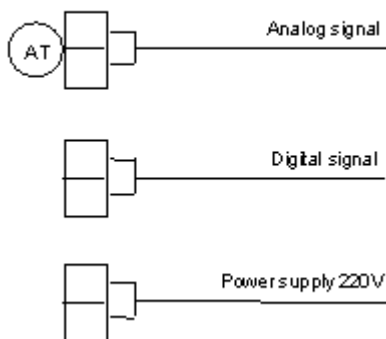
DCS Support for Manufacturer's Terminology

In the DCS definition, the headers can be customized as required. Therefore the labels can match any required manufacturer terminology. This affects the labels in the respective edit windows (panel, strip, terminal, cable, and so forth.) and in the actual SmartLoop drawings.

Generating SmartLoop Drawings

Use this procedure to generate SmartLoop drawings for single or multiple loops.

➢ **To generate SmartLoop drawings**

1. Do one of the following:

    *   Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.

    *   On the Loop Drawings module toolbar, click 🖫 to open the **Loop Explorer**.

2. Do one of the following:

    *   In the tree view pane, right-click a desired loop.

    *   In the **Entities** pane, select and right-click several loops.

3. On the shortcut menu, point to **Reports**, and click **Generate Loop Drawings**.

4. In the **Generate Loop Drawings** dialog box, from the **Generation method** list, select one of the following options:

    *   SmartLoop — To generate SmartLoop loop drawings for all the selected loops, regardless of the generation method assigned to the loops.

    *   As previously applied — To generate loop drawings according to the associated methods. In this case, the software generates SmartLoop drawings only for those loops that are assigned to SmartLoop drawing generation methods.

    🔆 **Tips**

    *   If from the **Generation method** list you selected SmartLoop, the software always applies the By Category method to all the loops. This means that if you need to generate a batch of SmartLoop drawings using the By Location or By Location with Setting method, you must first assign each loop manually to the By Location or By Location with Setting method.

    *   If among the loops that you selected there are loops assigned to the Manual method (symbol M in the **Explorer**), the software skips these loops when generating drawings in batch mode.

5. Specify automatic save options if required.

6. Click **OK**, and at the preview prompt, do one of the following:

- Click **Yes** to open the print preview of the generated drawing. Choose this option if you want to modify or annotate the drawing.

- Click **No** to start printing the drawing without displaying its print preview.

Notes

- When generating loop drawings in batch mode, SmartLoop displays only the first selected loop.

- When generating a SmartLoop drawing, a SmartLoop drawing overwrites any existing drawing that has not been generated by the SmartLoop generator for the selected loop or loops. However, a SmartLoop drawing does not overwrite an associated external CAD drawing.

Activities in the SmartLoop Drawing Print Preview

The SmartLoop feature enables you to generate loop drawings for one or more that you selected in the **Domain Explorer** (or in the **Loop Explorer**, which is only accessible from the Loop Drawings module). When the drawing is generated and the print preview window is open, you can perform several loop drawing actions. In the print preview of a loop drawing, you can perform the following activities:

- Define the loop drawing display settings.
- Modify the loop data.
- Annotate the drawing.
- Print the drawing.
- Move between drawings and their multiple pages.
- Save the drawing in a DXF file.

Sample Loop Drawing (After Generation)

SmartLoop Drawing Display Settings

SmartLoop Drawing Generation Methods

You can assign a SmartLoop drawing generation method to one or more loops you select in the **Domain Explorer** (or in the **Loop Explorer**, which is only accessible from the Loop Drawings module).

The following methods are available:

- By Category — This is the default SmartLoop drawing generation method. This method always applies when generating multiple SmartLoop drawings in batch mode. These are four default section headings of a loop drawing generated using the By Category method.

    - **FIELD** — device panel category

    - **FIELD JUNCTION** — junction box category

    - **TERMINATION PANEL** — marshaling rack category

    - **CONTROL SYSTEM** — DCS or PLC category

    If you do not want to let the software group the panels using the default sections, in the **Loop Diagram Settings** dialog box, you can set the number of panels and the panel category to be displayed in each section, modify the section headings and the number of sections.

- By Location

- By Location with Setting

Note

- If you need to generate SmartLoop drawings for a batch of loops by selecting the SmartLoop method in the **Generate Loop Drawings** dialog box, the software always applies the By Category method. This means that if you need to generate a batch of SmartLoop drawings using the By Location or By Location with Setting method, you must first assign each loop manually to the By Location or By Location with Setting method in the **Explorer**.

➢ To assign or change a SmartLoop drawing generation method

1. Do one of the following:

    - Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.

    - On the Loop Drawings module toolbar, click to open the **Loop Explorer**.

2. Do one of the following:

   - In the tree view pane, right-click a desired loop.

   - In the **Entities** pane, select and right-click several loops.

3. On the shortcut menu, point to **Apply Generation Method** >**SmartLoop**, and then, click the desired SmartLoop drawing generation method.

Display Settings

You can set the SmartLoop drawings display settings to automatic or manual mode. The default is set to automatic mode. So when you select **DEFAULT** from the **Loop Setting** list you actually select automatic mode. Any other configuration receives its own name and is selected in the same manner as the default does. Select the automatic mode to automatically calculate the optimum layout for a drawing. The calculation is based on the number of points in the loop and the selected paper size. The page layout changes automatically in accordance with any loop modifications that you perform.

Automatic Settings
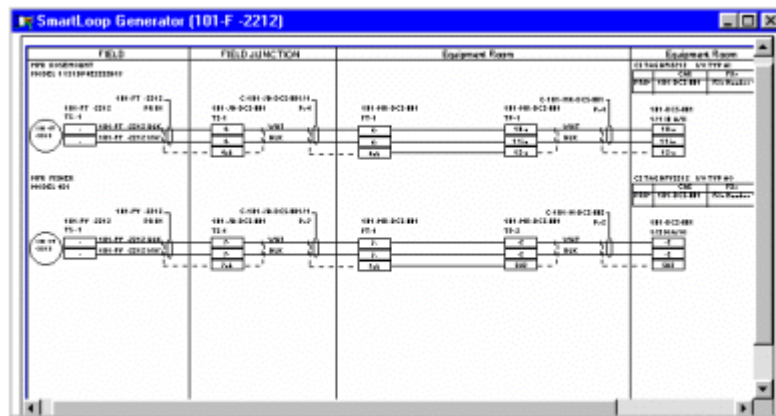
You can set the SmartLoop drawings display settings to automatic or manual mode. The default is set to automatic mode. When you select DEFAULT from the **Loop Setting** list in the **Loop Diagram Settings** dialog box, you actually select the automatic mode.

These are the actions SmartPlant Instrumentation performs when you select automatic settings:

- The number of sections available (maximum of 8).

- The number of panels displayed for each section.

- The number of wiring rows displayed vertically according to the vertical size of the paper (set in your default printer settings), up to a maximum of 12; the number of terminals displayed for all the rows in the largest terminal strip.

- The height of each row according to the number of terminals with active signals.

When the automatic settings are operative, you can change the section headers if required (for details, see Section Headers); SmartLoop sets any other parameter automatically.

Manual Settings

You can manually set the display for loop drawings. This is performed in the **Loop Diagram Settings** dialog box where you can manually select the number of elements in each section and set their arrangement on the drawing layer. Apply the new display settings by clicking **Attach**. You can generate a loop drawing for any of the loops in a domain according to your manually configured settings. However, when you generate a loop with your manually predefined settings, the drawing layout size is not optimized automatically. For example, increasing the number of strips and panels does not automatically change the layout size and therefore may cause the drawing to be overloaded with the loop elements.

Notes

- When you generate a SmartLoop drawing and access its settings, a default name is suggested for the loop settings derived from the name of the selected loop. Click **Attach** to apply the settings to the loop drawing, otherwise the default setting is used.

- If a manual setting has parameters that are identical to the current settings, (not including the header names) the current setting name is automatically suggested.

Caution

- Changing a particular existing loop setting that is already associated with loops changes all the related loop layouts.

Manual Settings - Section Headers

The section headers that appear in the drawings are determined by the following factors, regardless of whether automatic or manual display settings are in use.

1. Header names are retrieved from the text entry for the appropriate section header number in the **Loop Diagram Settings** dialog box (per loop setting).

2. If there is no text entry for the section in the **Loop Diagram Settings** dialog box, the section header pane is left empty.

Manual Settings - Number of Panels per Row in a Section

The number of panels (Device Panels, Junction Boxes, DCSs, and so forth.) you can display in sequence in a drawing row depends on the selected printer paper size. The following table displays the common options:

Page Size	Max. No. of Panels per Row
Letter (8.5" x 11"), A4	5
Ledger (11" x 17"), A3	8

If the number of panels in a loop exceeds the maximum, the extra panels are displayed in a new row, along with an appropriate reference to the previous row. A maximum of 12 rows can be displayed in a single loop drawing. Loop drawing elements can be arranged in up to eight screen sections each capable of containing one or more panels.

In the following example, there are six panels of four panel types arranged in four screen sections:

- Section 1 (entitled **Field**) contains two Device Panels.

- Section 2 (entitled **Field Junction**) contains a Junction Box.

- Section 3 (entitled **Equipment Room**) contains a Marshaling Rack with two strips. This panel is regarded twice due to its strip connections, even though in this example the strips belong to the same panel.

- Section 4 (entitled **Control System**) contains a DCS.

 Note

- The headers for sections 1, 2, and 4 are retrieved from the **Loop Diagram Settings** dialog box. The header for section 3 is retrieved from the location of the Marshaling Rack / Cabinet.

For this example, the SmartLoop drawing was set to print using paper size **Letter** (that is, a maximum of five panels are allowed per row) and the maximum number of panels in the third section (Equipment Room) was set to one. This setting causes the loop to break on the third section and continue on the next row, but enables two panels to be included in the first section (FIELD). This situation is illustrated in the screen shot below. Defining a maximum of two panels in the third section would require that the maximum number of panels in the first section be reduced to one. Therefore the total number of panels for all the sections does not exceed the nominal value (five for this example).
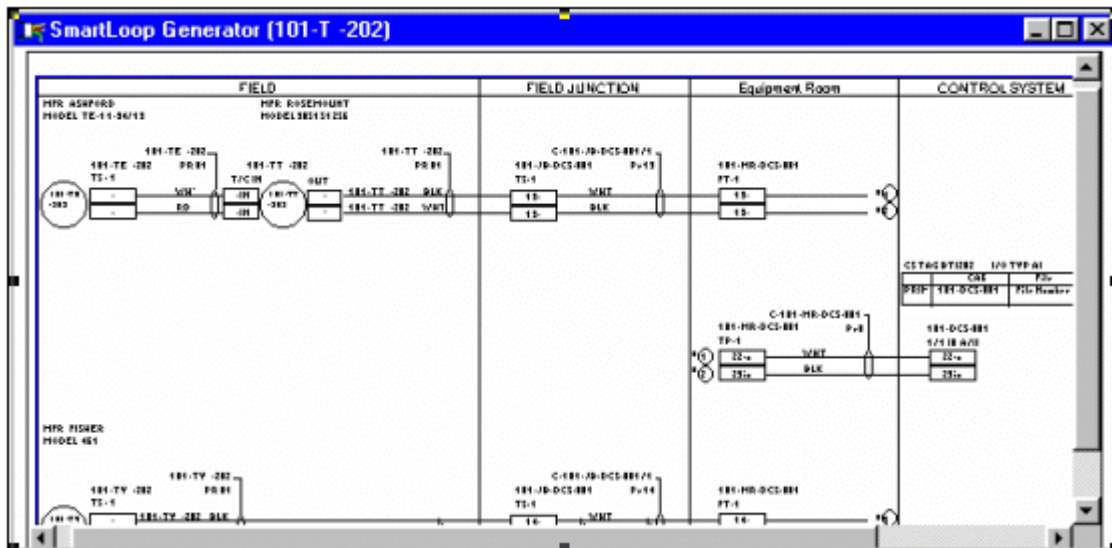
Manual Settings - Panel Arrangement

You can set the panel arrangement on the drawing layer for a particular loop drawing. The panels are arranged according to the following categories.

- Panel Location: the panels are arranged according to their actual location. The panel location is defined in the panel edit dialog box (when you create a new panel or edit an existing one).

- Panel Category: the panels are arranged according to six categories. A particular section may contain panels in one or more categories. The following categories are available:

  - **DP**: Device Panel (Field Device)

  - **JB**: Junction Box

  - **MR**: Marshaling Rack

  - **Cab**: Cabinet

  - **DCS**: Distributed Control System

  - **PLC**: Programmable Logic Controller

Note

- **Panel Location** has a higher priority than **Panel Category**. Only if no panel location is specified, or if none of the selected panel locations matches the panel location specified in the **Loop Diagram Settings** dialog box, the panels are arranged according to **Panel Category**.

Defining a New Loop Diagram Setting

This procedure explains how to define and save a new set of settings to use in the loop drawing generation process. However, this matches the manual settings category which can always be set to automatic (the default selection).

➢ **To define a new loop diagram setting**

1. Open the print preview of the required SmartLoop drawing.

2. To open the **Loop Diagram Settings** dialog box, do one of the following:

    - On the **Design** menu, click **Settings**.

    - Click 🔲

3. If required, click **Attach** to attach the setting to the currently selected SmartLoop drawing and to enable the fields and buttons in the window.

4. Click **New** to create a new loop setting and to make the **Loop Setting** field available for editing with the default entry NEW SETTING.

5. Type a name for the new loop setting.

6. In the **Section Header** data fields, type the required section headers to be used where no locations are defined for any of the panels in the section.

7. In the **Num. of Panels** data field, type the maximum number of panels a section can contain.

8. In the **Panel Position Criteria** section, do one of the following:

    - From the **Panel Location** list, select the required location. All the panels whose location settings match your selection are positioned in the specified section.

    - Leave the **Panel Location** field empty and select the appropriate **Panel Category** option buttons to assign a panel category for each section header. All the panels whose location settings match the selected category are displayed in the specified section.

📝 **Note**

    - You cannot select the same panel category for more than one section.

9. Click **Save**.

📝 **Note**

    - The drawing is automatically refreshed with the new settings.

Editing a Loop Diagram Setting

This procedure explains how to modify the display settings that you generate your loop drawings with. These settings allow you to control the panel arrangement, number of panels in a drawing row, and the text displayed in the section headers. The work with the display settings is performed using sets of settings that you assign to each and every one of your loop drawings. Those that you leave unattached are defined as loops with automatic settings. This procedure relates to a set of manual settings.

➢ **To edit a loop diagram setting**

1. Do one of the following:

    •   On the **Design** menu, click **Settings**.

    •   Click

2. From the **Loop Setting** list, select the required set of loop settings.

3. Click **Edit** to edit the selected loop setting.

4. Edit the loop setting parameters as required.

5. Click **Save**.

6. Click **Close**.

Attaching Display Settings to Loop Drawings

This procedure explains how to attach manually configured display settings to your loop drawings.

➢ **To attach a display setting to a loop drawing**

1. Open the print preview of the required SmartLoop drawing.

2. To open the **Loop Diagram Settings** dialog box, do one of the following:

    •   On the **Design** menu, click **Settings**.

    •   Click

3. From the **Loop Setting** list, select the required loop setting.

4. Click **Attach** to attach the setting to the currently selected SmartLoop drawing.

5. Click **Save**.

6. Click **Close**.

Deleting a Display Setting

This procedure explains how delete an existing set of display settings.

➢ **To delete a display settings**

1. Open the print preview of the required SmartLoop drawing.

2. To open the **Loop Diagram Settings** dialog box, do one of the following:

    *   On the **Design** menu, click **Settings**.

    *   Click

3. From the **Loop Setting** list, select the required loop setting.

4. Click **Delete**.

5. Click **Yes** at each prompt to confirm the deletion.

6. Click **Close**.

Modifying SmartLoop Drawings

Overview

This option allows you to modify the loop drawing data from within the SmartLoop print preview. The changes can be made both for a single and a batch of loops. After making the required changes, click elsewhere in the **SmartLoop Generator** window to save the changes to the database and regenerate the loop. The items whose data you can modify are listed here as follows.

- Component (Device Panel or another signal source)

- Cable, cable set, and wire

- Panel, strip, and terminal

- Connections in a selected panel

➢ **To modify a displayed loop drawing**

1. In the **SmartLoop Generator** window, move the pointer over the object whose data you want to modify. The pointer changes its shape.

2. Double click the object to open the appropriate edit dialog box where you modify its properties as required.
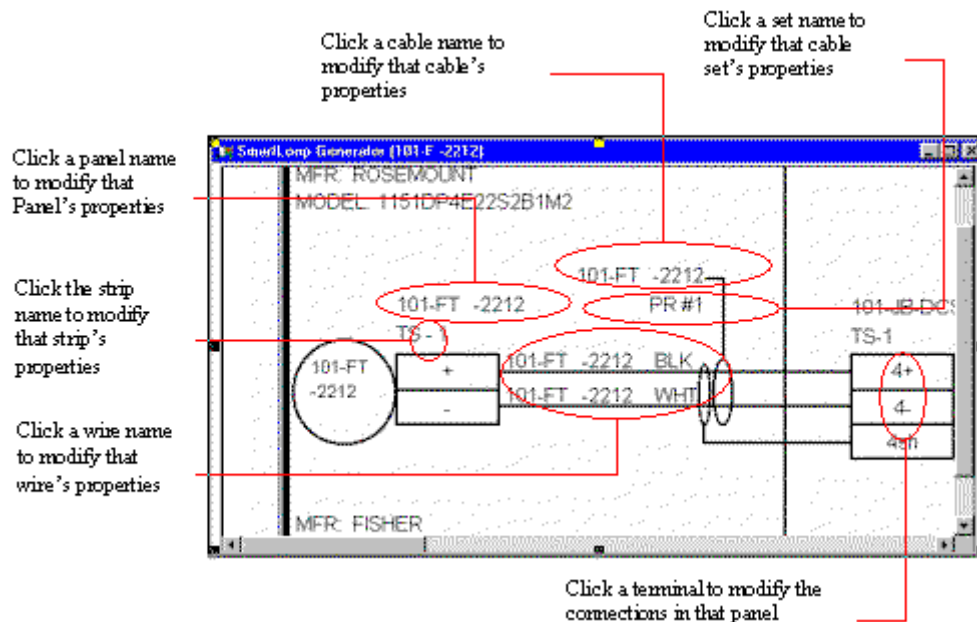
The following screen shot describes DCS associated items accessible to edit by clicking.

This information describes the CS Tag name, the CS Tag Type, and the number of points in the loop

Click a DCS panel name to modify the properties of that DCS

Click an I/O card name to modify its properties

Equipment Room

CS TAG: BFI2212　　I/O TYPE: AI　　CH: 4

	CAB	File	Pos
PRIM.	101-DCS-001	File Number 1	1

C-101-MR-DCS-001

I01　　　　　Pr4

101-DCS-001

I/1 (8 A/I)

WHT

BLK

10+v
11in
12-v

SmartLoop Generator (101-F -2212)

Additional Loop Drawing Attributes

SmartLoop drawings are generated with a title block and data concerning the wiring connections of the displayed objects. You can however, add other attributes of a selected item to the drawing, for example, manufacturer, model, and name. After you add an attribute, you can view and edit its text properties including its retrieving macro, size, and alignment of the text.

Note

- You can enable the use of the model and manufacturer additional attributes only in the **Preferences** dialog box.

If the drawing has a specific display setting, the additional attributes you define are saved with the setting, and appear in the other loops associated with that setting. Likewise, if you delete an attribute from a particular loop drawing, it is concurrently deleted from all other drawings associated with that setting.

➢ **To add an attribute to a loop drawing items**

1. In the **SmartLoop Generator** window, move the pointer over the object for which you want to add an attribute. The cursor changes its shape.

2. Right click the object to open its shortcut menu.

3. Select **Add Attributes** to open the **Insert Attribute** window.

4. In the **Macro Attributes** pane, highlight the required attribute.

5. From the list of macros for the selected attribute that appear in the lower data window, select the required macro, and drag it to the loop drawing.

6. Click **Close** to close the **Insert Attribute** window.

7. Click the attribute you just added to open its edit dialog box.

8. Set the required values and close the dialog box.

9. Move the attribute by dragging it to the required location.

Modifying Loop Drawing Attributes

This procedure explains how to modify the properties of an attribute relating to a certain object. For example, after you add the manufacturer name of a certain panel, you can modify its font size and text alignment.

➢ **To edit the properties of an attribute added to a loop drawing**

1. In the **SmartLoop Generator** window, move the pointer over the object that you want to add an attribute to. The cursor changes its shape.

2. Right click the object to open its shortcut menu.

3. Select **Edit Attribute Properties** to open the **Attribute Properties** window.

4. Click the **Font Size** and **Alignment** lists and change the font size and text alignment as required.

5. Click **OK**.

Deleting Loop Drawing Attributes

This procedure explains how to delete attributes that you added to a loop drawing relating to a certain object. The deletion is performed using the right-clicked shortcut menu.

➢ **To delete an object attribute**

1. In the **SmartLoop Generator** window, move the pointer over the object that you want to add an attribute to. The cursor changes its shape.

2. Right click the object to open its shortcut menu.

3. Select **Delete Attribute** to open the deletion confirmation prompt.

4. Click **Yes** to delete the attribute.

Redlining

You can annotate SmartLoop drawings using basic shapes and text in red. Use the Redlining toolbar to perform the redlining activities. These annotations are added within the loop drawing print preview and include the following options.

- Redlining in a SmartLoop drawing

- Setting redlining line properties

- Duplicating redlining items

- Editing redlining text

- Deleting redlining items

- Setting redlining color

➢ To display the Redlining toolbar
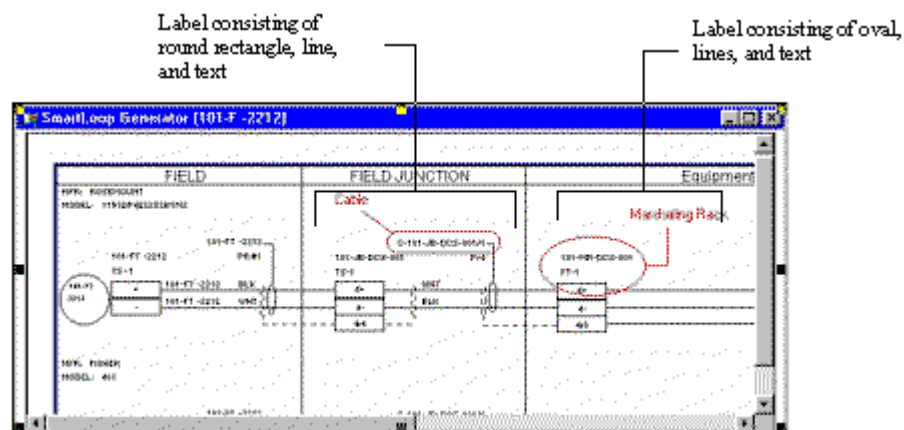
1. Open the print preview of the required SmartLoop drawing.

2. Do one of the following to display the Redlining toolbar:

   - On the **Design** menu, click **Redlining Menu**.

   - Click 🖌.

🖹 **Note**

- When you enable the redlining options, the left-click function for editing entities in the SmartLoop drawing is disabled.

The following screen shot shows an annotated SmartLoop drawing.

Redlining in a SmartLoop Drawing

After you access the Redlining toolbar, you can start annotating your drawing. This topic describes how to use the redlining options.

Moving a closed shape (circle, rectangle, triangle)

Click inside the shape and drag it to the required location.

Resizing a closed shape

Move the mouse pointer to one of the corners or sides of the shape until a double arrowhead appears, then drag the shape border to the required dimensions.

Drawing a line

➢ **To a draw line**

1. Select the line tool (⟍).

2. Place the line tool on the drawing layer as required and click once.

3. Move the line tool to draw the required line and click again.

4. Repeat steps 2 and 3 to draw as many lines as you need.

5. To stop drawing lines, reselect the pointer (⟈).

Moving a line

➢ **To move a line**

1. Select the pointer tool (⟈).

2. Double-click a line to select it.

3. Drag one end of the line to the new location.

4. Double-click the new line end to redraw the line.

5. If required, repeat steps 1 to 3 to move the other end of the line.

Adding text to the drawing

➢ **To add text to the drawing**

1. Select the text tool (**A**).

2. Click where you intend to place the text, to open the **Text Properties** window.

3. In the **Text** field, enter the required text.

4. From the lists, select the required font size and alignment.

Note

- To add a carriage return, hold down **Ctrl** and press **Enter**.

5. Click **Preview** to preview the text in the **Text** field.

6. Click **OK**.

Setting Redlining Line Properties

This procedure explains how to change the width or style of the redlining annotations that you add to SmartLoop drawings.

➢ **To set redlining line properties**

1. Select the pointer tool ( ).

2. Select one or more redlining objects as follows:

   - Double-click an object to select it.

   - Click and then drag the mouse pointer over several adjacent objects that you want to select.

3. On the **Design** menu, point to **Redlining**, point to **Selected Objects**, and click **Line Properties**.

4. In the **Line Properties** dialog box, change the line properties as required:

   - Under **Width**, click **+** or **-** to change the line width.

   - Under **Style**, click **+** or **-** to select the required line style.

 Note

   - You can change a line style for the narrowest line only.

5. Click **OK**.

Duplicating Redlining Items

You can duplicate redlining items that you have created. You duplicate items usually to save time when

➢ **To duplicate a redlining item**

1. Select the pointer tool (   ).

2. Select one or more objects as follows:

    •   Double-click an object to select it

    •   Click   and then drag the mouse pointer over several adjacent objects that you want to select.

3. On the **Design** menu, point to **Redlining**, point to **Selected Objects**, and click **Duplicate** to duplicate the selected objects.

4. Drag the duplicated items to the required location.

Editing Redlining Text

This procedure explains how to edit a text string that you have added as annotation to your SmartLoop drawing.

➢ **To edit redlining text**

1. Select the pointer tool ( ).

2. Select one or more objects as follows:

   - Double-click an object to select it

   - Click and then drag the mouse pointer over several adjacent objects that you want to select.

3. On the **Design** menu, point to **Redlining**, then point to **Selected Objects**, and select **Edit Text** to open the **Text Properties** dialog box.

4. Edit the text properties as required.

5. Click **OK**.

Deleting Redlining Items

This procedure explains how to remove a redlining item from SmartLoop drawings.
You can remove a single item and a batch of items as required.

➢ **To delete a redlining item**

1. Select the pointer tool ( ).

2. Select one or more objects as follows:

   - Double-click an object to select it

   - Click and then drag the mouse pointer over several adjacent objects
     that you want to select.

3. Do one of the following:

   - On the keyboard, press **Delete**.

   - Right-click the selected object and on the shortcut menu, click **Delete**.

 Tip

- To remove all unsaved redlining, click .

Setting Redlining Color

This procedure explains how to set the color of all redlining objects that you add to your SmartLoop drawing.

Note

- If you change the redlining color, it affects only redlining objects that you subsequently add to SmartLoop drawings. Existing redlining objects retain their original colors.

➢ **To set redlining color**

1. Do one of the following:

   - On the redlining toolbar, click the color icon ( ).

   - On the **Design** menu, point to **Redlining** and click **Color**.

2. In the **Set Color** dialog box, move the sliders to determine the amount of red, green, and blue to obtain the required color.

3. Click **OK**.

Saving SmartLoop Drawings in DXF Format

Saving a Single SmartLoop Report as a DXF File

The .dxf file format is compatible with the CAD applications used by SmartPlant Instrumentation. From the SmartLoop report print preview, you can save a single report as a .dxf file.

➢ **To save a single SmartLoop report as a .dxf file**

1. Open the desired SmartLoop report and do one of the following:

   - On the **View** menu, click **Save As**.

   - Click .

2. In the **Save As** dialog box, select the **DXF file** option button and click **OK**.

3. In the **Save as DXF Options** dialog box do one of the following:

   - Select **Include title block and frame with report** to save the report with its title block (which includes the logo and generic report data) and its frame.

   - Clear **Include title block and frame with report** to exclude the title block and frame when saving the report. (This may be relevant in instances where the CAD package supplies its own frame.)

4. If you are saving batch of reports to open with AutoCAD, proceed to optimize the process automation:

   a) Define the parameters in the **AutoCAD Options** section of the screen as follows:

   💡 **Tip**

   - It is recommended that you first insert a single report to AutoCAD and manually adjust the parameters until the required layout is obtained.

     Enter the coordinates for the insertion point of the report in the AutoCAD drawing. The insertion point is defined by the X- and Y- displacements from the lower-left corner of the AutoCAD drawing layer. The upper-left corner of the report is inserted relatively to that location.

     Enter the appropriate scale factor for sizing the report to fit into the AutoCAD layer.

b) Define the parameters under **User-defined options** as follows:

- Enter the font height and width coefficients for the text characters in the drawing. This is required because SmartPlant Instrumentation calculates the font size independently from the drawing size, whereas AutoCAD scales the font with the drawing as a whole.

- Enter an AutoCAD color to substitute black. All reports are displayed in black-and-white. When viewing the report in AutoCAD with a black background, an alternative foreground color has to be set. The recommended value is 7.

5. Click **OK**.

6. In the **Save DXF File As** dialog box, do the following:

a) Navigate to the required folder.

b) In the **File name** field, enter the required name for the saved DXF file.

c) Click **Save**.

Saving SmartLoop Reports as DXF Files in Batch Mode

The .dxf file format is compatible with the CAD applications used by SmartPlant Instrumentation. From the SmartLoop drawing print preview, you can save a group of drawings as DXF files. In batch mode, the drawings are saved as separate files in a folder that you can access outside SmartPlant Instrumentation so that you can open them using an external CAD application.

➢ **To save a group of reports as .dxf files in batch mode**

1. Open the print preview of a SmartLoop report.

2. On the **Design** menu, click **Save All as DXF**.

3. In the dialog box that opens, select the target folder (available for saving in batch mode only) in one of the following ways:

- Type the path name in the appropriate field.

- Click **Browse** to navigate to and select the required folder.

4. Click **OK**.

Loop Drawing Properties

Updating CAD Drawing Properties: An Overview

In the Loop Drawings module, you can specify a drawing type for every loop drawing that you want to generate using your CAD application.

The following drawing types are available:

- Single page — A CAD loop drawing type used to display all the drawing blocks and the title block on a single drawing page.

- Multi-page — A CAD loop drawing type used to display drawing blocks on more than one page, where all the pages share the same document number, description, and title block.

- Multi-drawing — A CAD loop drawing type used to display blocks on more than one page, where each page has an individual document number, description, and title block.

By default, all drawings belong to the single page drawing type.

All CAD drawings include drawing blocks and general blocks. Drawing blocks are instruments and wiring routings in a loop that make up the drawing. General blocks include the logo and title blocks. Loop drawing properties appear in the title block fields. These fields contain identifying information about the document number, page number, and file name, and also information about revisions, approvals, and references to other documents.

Properly completed identifying information ensures that each drawing is uniquely identifiable. You can maintain your loop drawing properties by keeping the revision, document reference and client / vendor approval information up to date.

For details, see the following topics:

- Modifying CAD Drawing Identifying Information

- Modifying Properties of a Single Page Drawing

- Modifying Properties of a Multi - Page Drawing

- Modifying Properties of a Multi - Drawing

- Maintaining CAD Drawing Revisions

- Maintaining Document References

- Making CAD Drawing Approvals

Modifying CAD Drawing Identifying Information

Use this procedure to set or modify loop drawing identifying information, such as a document number, description, page number, and file name.

➢ **To modify CAD drawing identifying information**

1. Do one of the following:

    • Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.

    • On the Loop Drawings module toolbar, click to open the **Loop Explorer**.

2. In the **Entities** pane, select and right-click loops.

3. On the shortcut menu, point to **Actions** and click **Loop Drawing List**.

4. In the **Loop Drawing List** dialog box, click **Properties**.

5. Modify the data as you require.

 Notes

    • Properties that you can edit are determined by the specified drawing type.

    • The document number does not have to be unique.

Modifying Properties of a Single Page Drawing

Use this procedure to modify properties of a CAD drawing to be generated as a single page drawing.

➢ **To modify properties of a single page drawing**

1. Do one of the following:

   - Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.

   - On the Loop Drawings module toolbar, click to open the **Loop Explorer**.

2. In the tree view pane, right-click a loop.

3. On the shortcut menu, point to **Actions** and click **Loop Drawing List**.

4. In the **Loop Drawing List** dialog box, select the loop, and then click **Properties**.

🔆 Tip

- If the drawing belongs to the single page type, the **Single page** option button under **Drawing type** is selected automatically. You can change the type of the drawing to multi-page or multi-drawing. For more information, see Defining a Drawing as a Multi - Page Drawing or Defining a Drawing as a Multi - Drawing.

5. Modify the values as you require.

Defining a Drawing as a Multi - Drawing

Use this procedure to define a drawing as a multi-drawing.

➢ **To define a drawing as a multi - drawing**

1. Do one of the following:

   - Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.

   - On the Loop Drawings module toolbar, click 🔲 to open the **Loop Explorer**.

2. In the tree view pane, right-click a loop.

3. On the shortcut menu, point to **Actions** and click **Loop Drawing List**.

4. In the **Loop Drawing List** dialog box, select the loop, and then click **Properties**.

5. In the **Loop Drawing Properties** dialog box, under **Drawing type**, click **Multi-drawing**.

6. Click **New** and then enter the values as you require. Repeat this action as many times as you need.

📝 **Notes**

- You cannot change the loop number value.

- In the **Page** column, you can type any alphanumeric character that designates a specific loop drawing page.

- You can specify an individual source path for any drawing page by clicking 📂 to the right of the selected drawing file. A source path is a path which software uses to locate the source file when generating a CAD drawing that belongs to the multi-drawing type. The path that you specify does not overwrite the path setting specified in the **Preferences** dialog box. The software saves all generated loop drawings in a designated folder that you define in the **Preferences** dialog box.

Defining a Drawing as a Multi - Page Drawing

Use this procedure to define a drawing as a multi-page drawing.

➢ **To define a drawing as a multi – page drawing**

1. Do one of the following:

    - Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.

    - On the Loop Drawings module toolbar, click to open the **Loop Explorer**.

2. In the tree view pane, right-click a loop.

3. On the shortcut menu, point to **Actions** and click **Loop Drawing List**.

4. In the **Loop Drawing List** dialog box, select the loop, and then click **Properties**.

5. In the **Loop Drawing Properties** dialog box, under **Drawing type**, click **Multi-page**.

6. Click **New** and then enter the values as you require. Repeat this action as many times as you need.

 Notes

- You cannot change the loop number value.

- In the **Page** column, you can type any alphanumeric character that designates a specific loop drawing page.

- You can specify an individual source path for any drawing page by clicking to the right of the selected drawing file. A source path is a path which software uses to locate the source file when generating a CAD drawing that belongs to the multi-page type. The path that you specify does not overwrite the path setting specified in the **Preferences** dialog box. The software saves all generated loop drawings in a designated folder that you define in the **Preferences** dialog box.

Modifying Properties of a Multi - Drawing

Use this procedure to modify properties of a multi-drawing.

➤ **To modify properties of a multi- drawing**

1. Do one of the following:

   - Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.

   - On the Loop Drawings module toolbar, click to open the **Loop Explorer**.

2. In the tree view pane, right-click a loop assigned to a multi-drawing.

3. On the shortcut menu, point to **Actions** and click **Loop Drawing List**.

4. In the **Loop Drawing List** dialog box, select the loop number row, and then click **Properties**.

5. In the **Loop Drawing Properties** dialog box, modify the values as you need. For details, see Defining a Drawing as a Multi - Drawing.

Modifying Properties of a Multi - Page Drawing

Use this procedure to modify properties of a multi-page drawing.

➢ **To modify properties of a multi-page drawing**

1. Do one of the following:

    -   Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.

    -   On the Loop Drawings module toolbar, click 🔲 to open the **Loop Explorer**.

2. In the tree view pane, right-click a loop assigned to a multi-page drawing.

3. On the shortcut menu, point to **Actions** and click **Loop Drawing List**.

4. In the **Loop Drawing List** dialog box, select the loop number row, and then click **Properties**.

5. In the **Loop Drawing Properties** dialog box, modify the values as you need. For details, see Defining a Drawing as a Multi - Page Drawing.

Assigning a Block to a Drawing Page

Where a block is associated with a multi-page drawing or multi-drawing, you can use this procedure to assign a block to a specific page.

➢ **To assign a block to a specific drawing page**

1. On the Loop Drawings module toolbar, click to open the **Loop Explorer**.

2. In the tree view pane, expand the hierarchy to the tag level, and select a tag.

3. In the **Entities** pane, right-click a block.

4. On the shortcut menu, point to **Actions** and click **Select Drawing Page**.

5. In the **Select Drawing Page** dialog box, select a page number.

6. Double-click the selected number or click **OK**.

Revision Display Order

You can display a number of revisions in the title block of a loop drawing. Each revision is represented by a numbered macro. The revisions are displayed in the order of the macro numbering beginning with the first or the last revision (ascending or descending order). You make the selection for ascending or descending order in the **Preferences** dialog box, **Loop Drawings** tag, **CAD Drawings** tag, **Revision macro order** list. If the total number of revisions is greater than the number of lines available for display in the drawing, and you want to see the latest revisions, you should select to display the revisions in **descending** order so that the latest revision is displayed first. If you display the revisions in ascending order, you would need to re-organize the revision numbering in the CAD application every time you make a revision, in order to display the latest ones.

Maintaining CAD Drawing Revisions

Revisions enable you to keep track of the changes made to the CAD loop drawing during its lifetime. It is important and useful to have a chronological description of the changes, dates of change, and a list of persons who approved them. Use the following procedure to add, edit, or delete revisions.

➢ **To maintain CAD loop drawing revisions**

1. Do one of the following:

   - Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.

   - On the Loop Drawings module toolbar, click to open the **Loop Explorer**.

2. In the **Entities** pane, select and right-click one or more loops.

3. On the shortcut menu, point to **Actions** and click **Loop Drawing List**.

4. In the **Loop Drawing List** dialog box, click **Revisions**.

5. In the **Revisions** dialog box, select one of the revision numbering methods (use **P0, P1, P2**... for preliminary revisions or **0, 1, 2** /**A, B, C** and so forth for normal serial revisions).

 Note

- When you first select a revision numbering method, several options are available to you, including preliminary revisions (designated by P0, P1, P2…). Once you select one of the other revision methods, you cannot return to the preliminary revision method and this option becomes disabled.

6. Click **New** to add new revision data or click **Edit** to update the existing data.

 Tips

- As a time saver and a forget-me-not precaution, take advantage of using a default Revision method.

- SmartPlant Instrumentation automatically adds a new line with the next logical character and date each time you click **New** after you select the initial method.

7. Add or edit the revision data in the appropriate data fields.

Tips

- The **By** column contains the current user initials by default, if previously defined by the System Administrator. You can edit this value as you require.

- You can delete obsolete revisions by clicking **Delete**.

- You can also maintain revisions in batch mode. For more information, see Global Revisions Overview.

Maintaining Document References

This option enables you to maintain document references that you include in title blocks of CAD loop drawings. You can manually add a new document reference, or you can edit or delete an existing document reference.

Title blocks can display the following document references:

- Automatic references to P&ID drawings

- Manual references to loop drawings

- Automatic references to reports generated in the Wiring module

 Notes

- You can view document references to reports generated in the Wiring module only if you have selected the **Automatic wiring report reference** check box in the **CAD Drawings** page of the **Preferences** dialog box for loop drawings. You cannot edit properties of these document references.

- The time of reference creation determines the order of the references in the title block. In the title block, the software displays the references in descending order, according to the time of creation. This means that the reference you create first appears in the title block at the top of the reference list.

- If a tag number in the selected loop has a reference in a P&ID drawing, this reference automatically appears in the **Document References** dialog box. You can create and edit this reference in the Instrument Index module. For details, see Instrument Index Module, Editing Tag Numbers.

➢ **To maintain a document reference**

1. Do one of the following:

    - Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.

    - On the Loop Drawings module toolbar, click to open the **Loop Explorer**.

2. In the **Entities** pane, select and right-click one or more loops.

3. On the shortcut menu, point to **Actions** and click **Loop Drawing List**.

4. In the **Loop Drawing List** dialog box, click **References**.

5. In the **Document References** dialog box, do one of the following:

- Select a row in the data window and click **Properties** to update the existing document reference name and description of a manually inserted reference.

- Click **New** to define properties of a new document reference.

🔅 **Tip**

- You can select an automatic reference to a wiring report and click **Properties** to define a reference description. Also, in the Instrument Index module, you can edit a P&ID drawing reference.

Making CAD Drawing Approvals

This option enables you to make client / vendor approvals for CAD loop drawing.

➢ **To make client / vendor approvals**

1. Do one of the following:

   - Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.

   - On the Loop Drawings module toolbar, click 📇 to open the **Loop Explorer**.

2. In the **Entities** pane, select and right-click one or more loops.

3. On the shortcut menu, point to **Actions** and click **Loop Drawing List**.

4. In the **Loop Drawing List** dialog box, click **Approvals** to open the **Edit Approvals** dialog box.

5. In the **Approvals** dialog box, set data for client / vendor approvals as you require.

Setting the Date Format in CAD Drawings

Use this procedure to set a date format in your CAD loop drawings. You set the date format when customizing your CAD drawing preferences in the **Preferences** dialog box.

➢ To customize the CAD drawing date format

1. On the **File** menu, click **Preferences**.

2. In the tree view, beside **Loop Drawings**, click ⊞ to expand the hierarchy view.

3. Click **General**.

4. In the **Date format** box, enter the date format using the syntax in the following table:

Date Format	Description or Example
Dash (-), slash (/), space, and punctuation marks	The character or space in the formatted data
D	Day with no leading zero (for example, 9)
Dd	Day with leading zero if appropriate (for example, 09)
Ddd	Day abbreviation (Sun, Mon, and so forth)
Dddd	Day name (Sunday, Monday, and so forth)
M	Month with no leading zero
Mm	Month with leading zero if appropriate
Mmm	Month abbreviation (Jan, Feb, and so forth)
Mmmm	Month name (January, February, and so forth)
Yy	Two-digit year (02)
Yyyy	Four-digit year (2002)
mm-dd-yy	Friday-Jan. 30-02
d-mmm-yy	30-Jan-02
dd-mmmm	03-January
mmm-yy	Jan-02
dddd, mmm d,yyyy	Friday, Jan. 30, 2002

Loop Drawing Reports

The following reports are associated with loop drawings that you can generate in SmartPlant Instrumentation:

- CAD Loop Drawing Generation Errors — Available on the shortcut menu in the **Domain Explorer** or **Loop Explorer**. This report lists a summary of errors that occurred during the last loop drawing generation using your CAD application. Data that appears in this report depends on specific loop numbers that you select.

- Macros — Available in the Loop Drawings module, on the **Reports** menu of the main window. Lists all the SmartPlant Instrumentation macros. This report provides you with information about every macro name, description, database name, and function description (if it exists). This report displays macros for all loop numbers existing in your database.

- User-Defined Macro Functions — Available in the Loop Drawings module, on the **Reports** menu of the main window. Lists all the macro functions you created on the basis of SmartPlant Instrumentation standard macro functions. The macro function affects the value which the macro returns after generation. For example, you can specify a user-defined function that removes all the spaces in the retrieved values and displays the remaining characters as upper case. This report displays user-defined macro functions for all loop numbers in your database.

CAD Loop Drawing Generation

Steps in Loop Drawing Generation Using a CAD Application

SmartPlant Instrumentation enables you to generate drawings for loop numbers in your domain using an external CAD application. These are the main steps in generating these loop drawings.

Drawing Preparations

- Creating CAD Drawing Blocks

- Inserting Macros in Drawing Blocks

- AutoCAD Plotter Settings

Defining Blocks in SmartPlant Instrumentation

- Define a Block Type

- Define a Block

- Define Default General Blocks

Setting CAD Application Parameters

For details on preliminary Intergraph SmartSketch, AutoCAD or MicroStation settings, see:

- Installation Guide, Appendixes > Appendix B > CAD Settings for SmartPlant Instrumentation

- SmartPlant Instrumentation and MicroStation

Customizing SmartPlant Instrumentation Macro Definitions (if required)

- Assign User-Defined Macro Names to Macros

- Create User-Defined Macro Functions

- Link External Macros to SmartPlant Instrumentation

Making Block Associations

- Define Assignments Between Blocks and Instrument Types

- Apply associated blocks to all tag number belonging to a specific instrument type

- Associate blocks with Tag Numbers Manually

Performing the Generation Process

- Generate the Drawing Define

Generating a Loop Drawing Using an External CAD Engine

SmartPlant Instrumentation allows you to generate loop drawings using external CAD applications. The same generation principles are valid for any of the listed CAD applications. The following CAD applications are supported:

- Intergraph SmartSketch

- AutoCAD

- MicroStation

In the generation process, SmartSketch (.sym), AutoCAD (.dwg), or MicroStation (.dgn) type drawings are created from the selected loops.

Prerequisites

- Before generating loop drawings, make sure that CAD application paths are set up correctly in the **Preferences** dialog box.

- If you use SmartSketch, prior to starting the generation process, in the **Preferences** dialog box, you must select the check box **Use macro symbol (&) to retrieve data** because in SmartSketch all macros have the '&' prefix. If you clear the check box, the software displays only macro attributes in the drawings.

- If you use AutoCAD, prior to starting the generation process, set the **Generation** dialog box option in the **Loop Drawings** tab of the **Preferences** dialog box to Visible to switch from AutoCAD back to SmartPlant Instrumentation when the generation process halts.

➢ **To generate a loop drawing using an external CAD engine**

1. Do one of the following:

   - Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.

   - On the Loop Drawings module toolbar, click to open the **Loop Explorer**.

2. In the **Entities** pane, select and right-click the desired loop numbers.

   Tips

   - If among the loops that you selected there are loops assigned to the Manual method (symbol M in the **Explorer**), the software skips these loops when generating drawings in batch mode.

3. On the shortcut menu, point to **Reports** and click **Generate Loop Drawings**.

4. In the **Generate Loop Drawings** dialog box, from the **Generation method** list, select CAD.

5. Under **Automatic save options**, select **Save drawing automatically** and do one of the following:

    •   In the **Output drawing path** data field, type the exact folder path to which you want to save the generated drawing.

    •   Click **Browse** to navigate to the desired path for saving the file.

6. Click **CAD Options**, and then, from **Save file in format** list, specify the CAD file format in which to save the generated drawings (if your CAD application supports more than one file format).

7. Select the following check boxes:

    •   **Send to plotter or printer** — Sends the generated drawing to the current CAD plotter.

    •   **Use macro functions** — Applies macro functions associated with the macros.

    •   **Save output data to database** — Enables you to compare drawing data of the current generation to data of a subsequent generation made for the same loop.

8. Under **Generation criteria**, do one of the following:

    •   Click **Per Loop** to generate drawings for all the loops you selected.

    •   Click **Per page** and then type the desired page number to generate only a specific drawing page for the drawings you selected.

    •   Click **Per block** to select a block from the list and generate drawings only for the loops that to which you assigned this block. The blocks displayed in the lists are blocks associated with tag numbers manually (indicated by the red icon in the **Loop Explorer**).

9. Under **Create report about**, select which reports you want to generate:

    •   **Null data** — Displays macro strings in the drawing for which SmartPlant Instrumentation data is unavailable.

    •   **Invalid macros** — Displays macro strings in the drawing which are not defined in SmartPlant Instrumentation.

    💡 **Tip**

    •   You can select to generate both reports if needed.

10. Under **Mark drawing using**, select the options for which you want to display a text string, specified in the adjacent text box:

   - **String for null data** — Allows you to type a string that indicates in the drawing those macros for which SmartPlant Instrumentation data is unavailable.

   - **String for invalid macros** — Allows you to type a string that indicates in the drawing those macros which are not defined in SmartPlant Instrumentation.

11. If you need to use macros from an external source during the current loop drawing generation, under **External data**, select the **Use external macro source** check box, and then click **Define**.

12. Click **OK** to start the loop drawing generation process in your CAD application.

Loop Drawing Generation Using a CAD Application - Basic Principles

The following diagram illustrates a (schematic) flow control loop.
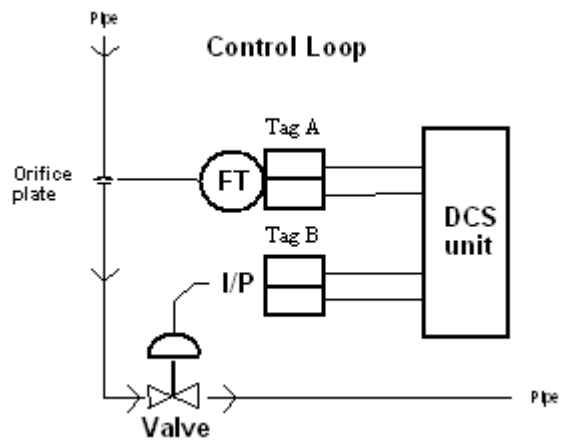
- Tag A is a monitoring system consisting of a flow transmitter and flow element.
- Tag B consists of a control valve with an I/P transducer.

Loop Drawing Generation Using a CAD Application - Blocks

Blocks are the basic loop drawing units. Each block graphically represents the functionality and wiring routing of one or more loop components. Blocks have to be associated with the tag numbers that they are representing. The Loop Drawings module enables you to define a block and associate it with a tag number, either directly or via an instrument type connection. In the Basic Principles example topic, the loop drawing includes five drawing blocks for the following components: monitoring system, control valve, each set of wiring connections to the DCS, and the DCS itself. The wiring routing and control systems are conceptually the same for both tags (but with different data), so the blocks for these items in this example, are identical.

Loop Drawing Generation Using a CAD Application - Macros

A macro enables you to perform a number of automated actions in a predefined order. When using a CAD application, macros are used to retrieve data from the database to a predefined location on the drawing layer. The macros are assigned per block and the text that they retrieve is eventually displayed as part of the block.

Each block can have one or more macros assigned to it. Same macros are used for retrieving data concerning instruments with the same functional type. For example, the function type is common for flow transmitters, flow elements, and I/P transducers. Therefore the same macros are used to retrieve the data for each of these components. The macros required for each of these components are the tag prefix (TAG_PRFX), instrument type (FUNC_ID), tag number (TAG_NUMC), and tag suffix TAG_SUFF).

The diagrams below display a loop drawing that was generated along with its related macros. This example illustrates how the same macros can be used to extract the same or different data for different tag numbers - the Tag Prefix and Tag Number are identical for each of the three components. However, the Instrument type is different, although represented by the same macro.

 Note

- No tag suffix was assigned to any of the components in this example.

Loop Drawing with Macros   **Generated Loop Drawing**
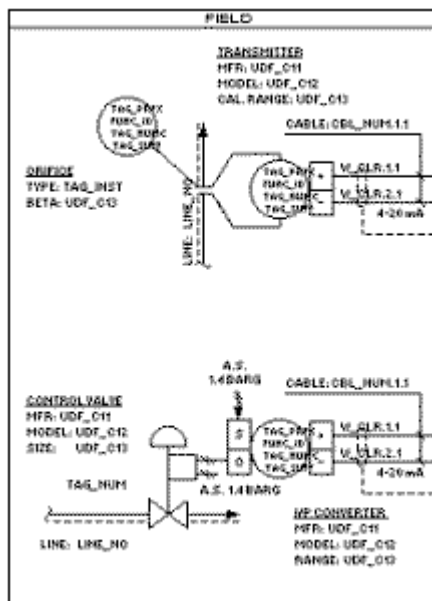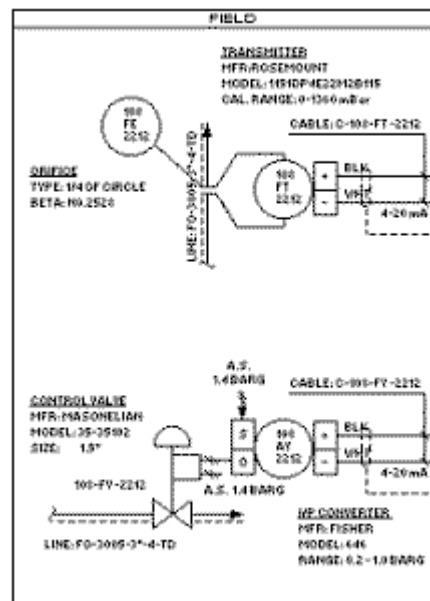
Associating an External CAD Drawing

This option enables you to associate an external CAD drawing with one or more loops you have selected in the **Domain Explorer** (or in the **Loop Explorer**, which is only accessible from the Loop Drawings module). An external CAD drawing is a drawing you have created outside the SmartPlant Instrumentation environment, using SmartSketch, AutoCAD, or MicroStation.

➢ **To associate an external CAD drawing**

1. Do one of the following:

   - Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.

   - On the Loop Drawings module toolbar, click 📇 to open the **Loop Explorer**.

2. In the **Entities** pane, select and right-click one or more loops.

3. On the on the shortcut menu, point to **Actions**, and click **Associate External CAD Drawing**.

4. In the **Select File** dialog box, select the required SmartSketch, AutoCAD or MicroStation file.

   📝 **Note**

   - You can associate the following file types: .dwg, .dxf, .dgn, .cgm.

5. Click **Open** to associate the selected drawing.

   📝 **Note**

   - If there is already an external CAD drawing associated with the current loop or loops, the new CAD drawing overwrites the existing one.

Viewing an External CAD Drawing

This option allows you to view an external CAD drawing that has been associated with one or more loops the **Domain Explorer** (or in the **Loop Explorer**, which is only accessible from the Loop Drawings module). An external CAD drawing is a drawing you have created outside the SmartPlant Instrumentation environment, using SmartSketch, AutoCAD, or MicroStation.

➢ **To view an external CAD drawing**

1. Do one of the following:

   • Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.

   • On the Loop Drawings module toolbar, click to open the **Loop Explorer**.

2. In the **Entities** pane, select and right-click one or more loops.

3. On the on the shortcut menu, point to **Reports** and click **View External CAD Drawing** to open the CAD drawing associated with the selected loop or loops.

Updating Paths for CAD Drawings and Block Files

Use this procedure to update in the SmartPlant Instrumentation database paths for existing drawings and block files. You can update paths globally for block files associated with tag numbers and for generated CAD drawings.

Note

- The paths that you specify do not overwrite the preferences settings specified for new block files and output drawings.

➢ **To update paths for existing CAD drawings and block files**

1. Open the Loop Drawings module.

2. On the **Actions** menu, click **Update Paths for Existing Drawings and Blocks**.

3. Under **Path for block files associated with tag numbers**, do one of the following:

    - Click **Apply to drawings in the domain/project** to apply the new path to all the plants in the current domain, or to the current project when the domain type is Operating owner.

    - Click **Apply to drawings in the current <plant> only** to apply the new path to the current plant only.

4. Type the full path to the folder where you keep your drawing blocks files or click **Browse** to navigate to the desired folder.

5. Under **Path for generated drawings**, do one of the following:

    - Click **Apply to blocks in the domain/project** to apply the new path to all the plants in the current domain, or to the current project when the domain type is Operating owner.

    - Click **Apply to blocks in the current <plant> only** to apply the new path to the current plant only.

6. Type the full path to the folder where you keep your generated drawings or click **Browse** to navigate to the desired folder.

CAD Loop Drawing Generation History

Displaying CAD Drawing Generation History

You can keep track of all the macro data changes in your CAD drawings. You choose this option when generating a loop using an external CAD engine. You can access the loop generation history from the print preview of loop drawings and the relevant reports.

The loop generation history feature compares the retrieval values of macros from different generations. Therefore, the history data is available when you have already generated at least one loop drawing for a selected loop.

When comparing a loop drawing that you have just generated with an older one, you have to set the comparison date range to include both generations. If that date includes more than one loop drawing, select the exact loop drawing to which you want to compare the currently generated one.

The results show a comparison of the current data in the loop drawing with the data stored in the history for the specified date range or generation.

➢ **To display CAD drawing generation history**

1. In the **Generate Loop Drawings** dialog box, in the **CAD Options** tab, select the **Save output data to database** check box.

2. In the **CAD History** tab, select the **Compare outputs** check box if this is not the first generation to make the additional options available.

3. Do one of the following:

   - Click **Set date range** and then, in the **From** and **To** boxes, specify the history date range you need for the comparison.

   - Click **Select specific generation**, and then, from the data window, select a specific generation to which you want to compare the current generation.

4. Under **History data**, select one of the following check boxes or both check boxes:

   - Select **Display in the drawing** to display history data in the generated drawing.

   - Select **Display in the report** to display the **Data Changes Report** before proceeding with the loop generation. The report includes the tag number, block name, macro name and description, old value and the new value.

5. Do one of the following:

    •   Click **Show data from previous generation** to display the old history
        value from the previous generation.

    •   Click **Mark with user-define value** and type a new value in the box below.
        This marks the drawing or the report with the new value instead of the old
        history value from the previous generation.

    💡 **Tips**

    •   In the generated drawing or report, the software displays the old value in
        parentheses.

    •   If you selected **Display in the report**, the software displays the **Data
        Changes Report** before proceeding with the loop generation. After
        printing or viewing the report, click **OK** to return to the **Generate Loop
        Drawings** dialog box.

6. Click **OK** to start the generation process.

Deleting CAD Drawing Generation History

To improve performance when comparing generated data values in CAD drawings, you need to delete the history of obsolete CAD drawing generations. Before deleting the values of previous CAD drawing generations, you select the required loops, and then set the period within which you want to delete the selected history records.

➢ **To delete CAD drawing generation history**

1. Do one of the following:

   - To delete CAD drawing generation history for specific loops using the **Domain Explorer**, press F7, right-click loops that you select, and then, on the shortcut menu, point to **Actions** and click **Delete Generation History Records**.

   - To delete CAD drawing generation history for specific loops using the **Loop Explorer**, on the Loop Drawings module toolbar, click 🔲, right-click loops that you select, and then, on the shortcut menu, point to **Actions** and click **Delete Generation History Records**.

2. In the **From** and **To** boxes, specify the date range of the generation history records that you want to delete.

3. Click **Show Records**.

4. Under **Select records for deletion**, select history records that you want to delete.

   💡 **Tip**

   - Select the **Select all** check box to be able to delete all the displayed records at once.

5. Click **Delete**.

Showing the Last Generated Drawing

This option enables you to display the last loop drawing that was generated in SmartPlant Instrumentation using an external CAD application. The drawing is opened in the default CAD application.

➢ **To show the last generated loop drawing**

- On the **Actions** menu of the Loop Drawings module, click **Show Last Generated Drawing** to open the loop drawing in the appropriate CAD application.

Clearing Temporary Data

When generating a loop drawing the macro results are saved in a SmartPlant Instrumentation table for the next time you want to generate the same loop. The saved data accumulates and at a certain stage might affect the generation performance.

➢ **To clear loop generation temporary data**

1. Do one of the following:

    - Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.

    - On the Loop Drawings module toolbar, click to open the **Loop Explorer**.

2. In the **Entities** pane, select and right-click loops whose temporary generation data table you want to clear.

3. On the shortcut menu, point to **Actions** and click **Clear Temporary Data**.

Generating a Report of CAD Loop Drawing Generation Errors

Use this procedure to create a CAD drawing generation error report for loop numbers whose loop drawings you generated using a CAD application.

➢ **To generate a CAD Loop Drawing Generation Errors report**

1. Do one of the following:

    -   Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.

    -   On the Loop Drawings module toolbar, click to open the **Loop Explorer**.

2. In the tree view pane, right-click a loop for which you want to generate the CAD loop generation error report.

3. On the shortcut menu, point to **Reports** and click **CAD Loop Drawing Generation Errors**.

CAD Drawing Blocks

Overview

The first step in generating a loop drawing is to create drawing blocks. Drawing blocks are objects that graphically represent the drawing components. A standard set of CAD drawing blocks is available in SmartPlant Instrumentation. You can use or modify them as required.

Standard drawing blocks exist for each of the following CAD applications (shown with the appropriate file extension):

- Intergraph SmartSketch — .sym

- AutoCAD — .dwg

- MicroStation — .cel

Notes

- To enable MicroStation SE to work with long file paths, open the INTOOLS.INI file, and under the [LOOP] section, change the value of the MicroStation parameter from 95 to SE or J.

- There must be a correlation between the file formats supported by your CAD application and the file format in which you save a new drawing block.

- When defining a new CAD drawing block for MicroStation, the files you select represent cell libraries. To specify a particular cell in the library, you must type its name manually in the appropriate field.

Depending on the CAD application that you use, the following equivalent terms apply.

SmartPlant Instrumentation Term	SmartSketch	AutoCAD	MicroStation
Block	Symbol	Drawing	Cell
Macro	Text element	Attribute element	Tag

Adding Blocks to New Drawings

The drawing block position is defined by the relation between its insertion point and the drawing origin (lower left corner). Therefore, ensure the following:

- The origin points of the drawing and the block are linked.

- You use the same scale for the drawing and the drawing blocks.

- You are not using nested blocks.

Defining Insertion Points for Blocks

In SmartSketch, AutoCAD or MicroStation, you can define the insertion point for individual blocks relative to the drawing origin (lower left corner). This enables you to use the same block in different positions in different loop drawings.

➢ **To define the insertion point for a block**

1. On the Loop Drawings module toolbar, click 🔲 to open the **Loop Explorer**.

2. Expand the loop-tag-block hierarchy and then do one of the following:

   - In the tree view pane, right-click a desired block.

   · In the **Entities** pane, select and right-click several blocks.

3. On the shortcut menu, point to **Actions** and click **Define Block Origin**.

4. In the **Define Block Origin** dialog box, select a block.

   💡 **Tip**

   - If there is a large number of blocks in the data window, under **Find**, select a column heading and then type a value. For example, if you select Tag Number as a column heading and type a specific tag number, in the data window, the software locates the block assigned to this tag number.

5. Under **Tag assignments**, enter the required values (in the CAD application units) for the block new insertion point X- and Y-coordinates.

   💡 **Tip**

   - The block origin can consist of numbers that include decimal values, for example, 9.85.

Inserting Macros in Drawing Blocks

Drawing blocks in loop drawings generated using CAD applications can display SmartPlant Instrumentation data only if the macros that they include are recognized by SmartPlant Instrumentation. The following procedure explains how to include the macro names in the blocks.

SmartSketch

Macros are entered as text strings. The application processes as a macro any text string that is recognized as a macro by SmartPlant Instrumentation.

MicroStation

Macros are entered as tags. The application processes as a macro any tag that is recognized as a macro by SmartPlant Instrumentation.

AutoCAD

Macros are entered as attribute elements. Macro colors need to be assigned by layer property. It is also recommended that macros be written in layer zero.

➢ **To insert macros in a drawing block**

1. Draw the required image.

2. Perform the following actions for each command line prompt. If you have finished drawing and want to perform these actions immediately, use the supported ATT AutoLISP function or do the following:

   a) Set AutoCAD system variable AFLAGS = 8

   b) For AutoCAD 2000 – 2004, make sure that the AutoCAD system variable SDI=0.

   c) Perform the following actions at the command line prompts:

Command line prompt	Action
ATTDEF	Press **Enter**
Attribute modes	Press **Enter**
Attribute Tag:	Type in [macro name] and press **Enter**
Attribute prompt:	Press **Enter**.
Default attribute value	Type in [macro name] and press **Enter**
Justify/Style/<Start point>:	Position cursor and click once to define the bottom left position of the string.
Height:	Position cursor and click once to determine the height of the macro text. (The height of the macro text is also dependent on the current font size.)
Rotation angle <0>:	Type in a number to represent the angle of rotation of the macro text from the horizontal (the default is 0)

🔆 **Tip**

- You can have AutoCAD automatically enter most of the above parameters by using the ATT AutoLISP application. This application is automatically installed if you select AutoCAD as your default CAD interface. By using the ATT application you only need to enter the macro name once, and enter the start point, height and rotation parameters.

Plotting AutoCAD Drawings

The Loop Drawings module uses the shipped LISP function PLOTE.LSP for plotting generated drawings. If this function is not suitable for the specific current AutoCAD plotter, you can select the **Current Systems Plotter** option from the **AutoCAD Plotter Configuration menu**, or modify the PLOTE.LSP file. If you do the latter, you should be aware that the last 'Enter' for the AutoCAD plot command is supplied automatically by the system; you do not need to press **Enter** manually.

📝 **Note**

- The locations of various files used by the module appear in **Preferences** dialog box.

SmartSketch Drawing Blocks

When generating loop drawings using SmartSketch, you need to create a template for each <plant>. The template contains the attributes (for example, size, scale) of the drawings that will be used for the generation process. Each drawing block is a SmartSketch symbol (.sym file). The symbol, used to create the template must be the biggest one in the domain; usually, this is the border.

The following procedures explain how to create:

- Templates
- Drawing block symbols

SmartSketch Drawing Blocks - Creating a Template

The following procedure explains how to create a template in SmartSketch format.

➢ **To create a drawing template in SmartSketch**

1. Launch SmartSketch and load the file containing the border.

2. On the **Edit** menu, click **Select All**, and then click **Copy**.

3. On the **File** menu, click **New** and choose any template.

4. On the **Edit** menu, click **Paste**.

5. On the **View** menu, click **Fit** to view the entire drawing.

6. On the **Edit** menu, click **Select All**, and then click **Delete**.

7. On the **File** menu, click **Save As Template** and name the file PID.SYM (the file is saved in the SmartSketch TEMPLATE folder).

8. Copy the saved file to the path shown in Preferences (Loop Drawings - CAD File Locations) for the **CAD function folder** property.

SmartSketch Drawing Blocks - Drawing Block Symbols

The following procedure explains how to create drawing blocks in SmartSketch format.

➢ **To create drawing blocks in the SmartSketch format**

1. Create or open the required drawing file in SmartSketch.

2. On the **Edit** menu, click **Select All**.

3. On the **Draw** toolbar, click .

4. Click the bottom left corner of the drawing to define the symbol origin.

5. In the **Save as Symbol** dialog box, navigate to the required location and type the name of the symbol file to be used as the drawing block.

Working with Blocks

Blocks in SmartPlant Instrumentation: An Overview

A drawing block must belong to a particular block type, therefore you must define at least one new block type before you can add a new block (apart from the default general block types). Therefore, after you defined the loop drawing blocks in the CAD application you need to define the blocks and block types in SmartPlant Instrumentation.

You can set up blocks in two block categories: general and non-general. The general category includes the border (revisions, reference, and so forth), and the logo. For example, the border and logo block types and general blocks with the same name are shipped with SmartPlant instrumentation. You can change these as required. Non-general blocks are drawing blocks that can be referenced to any data associated with an instrument in a loop.

You can use similar blocks to represent identical components in different drawing positions by defining several blocks that use the same drawing file.

💡 Tip

- To save time, create all block types and blocks before starting to generate the loop drawings.

Predefined Drawing Blocks

SmartPlant Instrumentation is shipped with several examples of blocks that are used in the demo database. The wiring methodology behind these blocks includes the use of an overall shield with single cables and multiple pair cables. After the installation, these blocks can be found in the path <SmartPlant Instrumentation home folder>\CAD\Blocks.

💡 Tip

- Some of the blocks are used in the demo database. It is therefore recommended that you make copies of any blocks you intend to modify.

The blocks are categorized by block types, and include the available formats for each file. The formats corresponding to specific CAD applications are as follows:

- SmartSketch – .sym
- AutoCAD – .dwg
- MicroStation – .cel

Block Types: An Overview

Block types are groups of similar blocks. A block type allows you to organize and sort your block list according to the block role in the loop drawing. Blocks always belong to a block type.

 Note

- The BORDER and LOGO block types are default general blocks and cannot be edited or deleted.

Adding Block Types

You can add a new block type as part of your preferred block list sorting convention.

➢ **To add a block type**

1. Open the Loop Drawings module and do one of the following:

    - On the toolbar, click .

    - On the **Tables** menu, click **Block Types**.

2. In the **Block Types** dialog box, click **New**.

3. In the **Block Type Properties (New)** dialog box, type the new block name and description in the appropriate fields.

 Tip

- Select **Set category as general** if you want the new block type to belong to the general category.

Editing Block Types

You can edit existing block types to fit your preferred block list sorting convention.

➢ **To edit a block type**

1. Open the Loop Drawings module and do one of the following:

    *   On the toolbar, click .

    *   On the **Tables** menu, click **Block Types**.

2. In the **Block Types** dialog box, select the desired block type.

3. Click **Properties**.

 Note

    *   You cannot edit the default BORDER and LOGO block types.

4. In the **Block Type Properties** dialog box, modify the data as you need.

Adding Blocks to a Block Type

The following procedure explains how to add a block to an existing block type.

➢ **To add a block to a block type**

1. Open the Loop Drawings module and do one of the following:

   - On the toolbar, click 🔳.

   - On the **Tables** menu, click **Blocks**.

2. In the **Blocks** dialog box, click **New**.

3. In the **Block Properties (New)** dialog box, enter information as follows:

   a) In the **Block** field, enter the name to be used for the block by SmartPlant Instrumentation.

   b) In the **Description** field, type in a description for the block.

   c) Under **Block type**, select a block type from the list.

   📝 **Note**

   - The block types displayed in the list depend on whether the **General category** check box has been selected. When selected, the list displays general blocks only. When cleared, the list displays drawing blocks. A block belonging to the general category cannot hold instrumentation data and, therefore, cannot be assigned to a tag number. General blocks are usually blocks used as borders or logos in loop drawings. Blocks that belong to the general category appear on all loop drawings automatically.

   d) Under **Drawing block file**, navigate to the block source file by clicking **Browse**.

   e) Specify the file path.

   f) Under **Origin**, enter the required X and Y parameters to position the new block on the drawing.

   💡 **Tips**

   - You can enter decimal numbers as well as whole numbers.

   - Click **View** to view the drawing block in your default CAD application.

Editing Block Properties

Use this procedure to edit properties of an existing block.

➢ **To edit block properties**

1. Open the Loop Drawings module and do one of the following:

   - On the toolbar, click 🔁.

   - On the **Tables** menu, click **Blocks**.

2. In the **Blocks** dialog box, to display blocks in the data window, do one of the following:

   - To display blocks that belong to a specific block type, from the **Block type** list, select the desired block type.

   - To display all the available blocks, select **Show blocks of all types**.

   💡 **Tips**

   - If there is a large number of blocks in the data window, under **Find block**, select a column heading and then type a value. For example, if you select Block as a column heading and type block name Control, in the data window, the software locates the block whose name starts with Control.

   - If you want to display only specific blocks in the data window, click **Filter** and specify a filter parameter.

3. Select the desired block in the data window and click **Properties**.

4. In the **Block Properties** dialog box, modify the data as you require.

5. Click **Browse**.

6. Navigate to the required file and click **OK**.

7. In the **Origin** section, if needed, type the X and Y parameters to locate the new block in the drawing.

   💡 **Tip**

   - You can type decimal numbers as well as whole numbers.

Copying Blocks to Another <Unit>

Use this procedure to copy one or more blocks from the current <unit> to another <unit> in the domain (or in the current project if the domain type is Operating owner).

➢ **To copy blocks to another <unit>**

1. Open the Loop Drawings module and do one of the following:

   • On the toolbar, click .

   • On the **Tables** menu, click **Blocks**.

2. In the **Blocks** dialog box, to display blocks in the data window, do one of the following:

   • To display blocks that belong to a specific block type, from the **Block type** list, select the desired block type.

   • To display all the available blocks, select **Show blocks of all types**.

3. In the data window, select blocks that you want to copy to another <unit>.

4. Click **Copy To**.

5. In the **Select Target <Unit>** dialog box, select a desired <unit> and click **OK**.

Defining Default General Blocks

SmartPlant Instrumentation supports two types of general blocks for **all** the CAD drawings: BORDER and LOGO. Use this procedure to determine which general blocks your CAD application uses to generate loop drawings.

➢ **To define a default general block**

1. Open the Loop Drawings module.

2. On the **Actions** menu, click **Default General Blocks**.

3. In the **Default General Blocks** dialog box, select a block from each category to be the border and logo.

Loop Title Block

Border and Logo

The files in the list are the predefined drawing blocks that can be used to represent the border / title (one of which should be defined as the default border) and the INTERGRAPH logo (one of which should be defined as the default logo).

File Name	Description	Available Formats
BORD_DEF	Border and title.	.dwg, .sym
BORD_DEM	Border and title.	.dwg
BORDER	Border and title.	.cel
BRDDEM	Border and title.	.cel
TITLE	Border and title.	.cel
INGR	Intergraph logo.	.cel
LOGO_DEF	Intergraph logo.	.dwg, .sym
LOGO_DEM	Intergraph logo.	.dwg, .sym
LOGO_IN	Intergraph logo.	.dwg
LOGOIG	Intergraph logo.	.cel

Field Device Blocks

File Name	Description	Available Formats
CV2	Control valve.	.dwg, .sym
ORIF_1	Orifice on a steam traced line.	.dwg
ORIF_2	Orifice.	.dwg
PLOC_IND.DWG	Parallel connection of an indicator.	.dwg
PT_1	Pressure element.	.dwg
SLOC_IND.DWG	Series connection of an indicator.	.dwg
TC_1	Skin temperature controller (welded).	.dwg
TC_2	Temperature element.	.dwg
TE	Temperature element wired to a temperature transmitter.	.dwg

Field Device Blocks

Wiring Routing Blocks

File Name	Description	Available Formats
AIT_ROUT	Typical analog input routing; field device, junction box, marshaling rack, and an analog input I/O card.	.dwg
AOT_ROUT	Typical analog output routing; I/P transducer, junction box, marshaling rack, and an analog output I/O card.	.dwg
PAIT1	Use for parallel indicator and transmitter.	.dwg
SAIT	Use for serial indicator and transmitter.	.dwg

DCS and Control Blocks

File Name	Description	Available Formats
CTRL1	Analog input control.	.dwg
CTRL2	Analog input and analog output control.	.dwg

Associating Blocks

Block Association Methods

After defining the blocks and their macros, it is necessary to associate your tag numbers with the drawing blocks that represent them graphically.

The following is the description of the association methods available in SmartPlant Instrumentation:

- Automatic method: block – instrument type association. Using this method, you first assign one or more existing blocks to a specific instrument type, and then apply these blocks, so that in the **Loop Explorer,** they are automatically associated with all tag numbers belonging to a particular instrument type. These blocks are marked in green: .

- Manual method: block – tag number association. Using this method you, manually associate one or more blocks with a specific instrument tag. In the **Loop Explorer**, manually associated blocks are marked in red: .

After changing the block association method from automatic to manual, these blocks are no longer associated with an instrument type, and therefore are not affected by any changes you or other users can make to the block – instrument type association.

Associating Blocks with Instrument Types

In an instrumentation design setup with a very large number of control loops, similar tag numbers are represented repeatedly using the same drawing block, the only difference being the data that is retrieved by the macros in the drawing.

Therefore, instead of having to associate each block with a tag number manually, you can configure SmartPlant Instrumentation to automatically associate typical blocks with specific instrument types.

Use this procedure to associate a block with an instrument type. Then, you can apply the associated block to loops or tags selected in the **Loop Explorer**, so that the block is automatically associated with all tag numbers belonging to a particular instrument type.

🔆 Tip

- To save time, make all your default block – instrument type association before starting to generate the loop drawings.

➢ **To associate one or more blocks with an instrument type**

1. In the Loop Drawings module, do one of the following:

    - On the toolbar, click ⬚.

    - On the **Actions** menu, click **Block – Instrument Type Assignment**.

2. In the **Block – Instrument Type Assignment** dialog box, in the **Type** data window column, select an instrument type.

🔆 Tip

- If there is a large number of instrument types in the data window, under **Find instrument type**, select a column heading and then type a value. For example, if you select Description as a column heading and type Control, in the data window, the software locates the instrument types whose description starts with Control.

3. Click **Assign**.

4. In the **Assign Blocks to Instrument Type** dialog box, to display blocks in the data window, do one of the following:

    *   To displays blocks that belong to a specific block type, from the **Block type** list, select the desired block type.

    *   To display all the available blocks, select **Show blocks of all types**.

5. In the data window, select blocks that you want to assign to the instrument type.

 Tip

    *   Click **View** to open a block in your CAD application.

6. Click **OK**

Applying Associated Blocks

This procedure enables you to apply associated blocks to selected loops or tag numbers. This means that you can add the blocks that have been associated with a particular instrument type to the selected loops or tag numbers. Also, you can overwrite previously applied or associated blocks.

Note

• You can only perform this procedure from the Loop Drawings module.

➢ **To apply associated blocks**

1. On the Loop Drawings module toolbar, click to open the **Loop Explorer**.

2. Do one of the following:

   • In the **Entities** pane, select and right-click one or more loops.

   • In the tree view pane, select a specific loop and then in the **Entities** pane, select and right-click tag numbers.

3. On the shortcut menu, point to **Actions**, and click **Apply Associated Blocks**.

4. In the **Apply Associated Blocks** dialog box, do one of the following:

   • Select the **Apply to all selected tags** check box to automatically apply the associated blocks to all the selected tag numbers Click **Skip** to open the **Block Initialization** dialog box for another tag number.

   • Clear the **Apply to all selected tags** check box to apply the associated blocks to each tag number separately.

5. Select one of the following options:

   • Select **Add block** to add a block to the current tag number, or to all the tag numbers if you have selected the **Apply to all selected tags** check box.

   • Select **Overwrite** to replace all blocks already associated with the current tag number, or with all the tag numbers if you have selected the **Apply to all selected tags** check box.

   Caution

   • When selecting the **Overwrite** option, any existing blocks associated manually with the current tag number are dissociated.

6. Click **OK**.

Associating Blocks with Tag Numbers

You can manually associate one or more blocks with tag numbers belonging to the same loop number. In this case, that the software draws blocks after you select a loop number in the **Domain Explorer** (or in the **Loop Explorer**, which is only accessible from the Loop Drawings module).

Note

- If you have a large number of loops with tag numbers that have identical properties, you can automatically associate blocks by instrument type. .

➢ **To associate blocks with tag numbers**

1. Do one of the following:

    - Press F7 to open the **Domain Explorer** and then expand the hierarchy to the tag level in the **Loops** folder.

    - On the Loop Drawings module toolbar, click to open the **Loop Explorer** and then select a loop.

2. In the **Entities** pane, select and right-click tag numbers to which you want assign blocks.

3. On the shortcut menu, point to **Actions** and click **Associate Blocks with Tag Numbers**.

4. To display blocks in the data window, do one of the following:

    - To displays blocks that belong to a specific block type, from the **Block type** list, select the desired block type.

    - To display all the available blocks, select **Show blocks of all types**.

Tips

- If there is a large number of blocks in the data window, under **Find block**, select a column heading and then type a value. For example, if you select Block as a column heading and type block name Control, in the data window, the software locates the block whose name starts with Control.

- If you want to display only specific blocks in the data window, click **Filter** and specify a filter parameter.

5. In the data window, select the blocks that you want to associate, and then click **OK**.

6. In the **Specify Drawing Pages for Blocks Assignment**, under **Page**, use spinners to specify pages on which you want the selected blocks to appear after loop drawing generation.

 Note

- Selecting pages is available when a loop number is assigned to a multi-page drawing or multi-drawing.

Copying Associated Blocks

Use this procedure to copy one or more blocks that are associated with a particular tag number to other tag numbers displayed in the **Loop Explorer**.

Note

- You can copy only those tags that belong to loops that are assigned to the CAD generation method in the **Loop Explorer**.

➢ **To copy blocks to the target tag numbers in batch mode**

1. On the Loop Drawings module toolbar, click 🖳 to open the **Loop Explorer**.

2. In the tree view pane, expand the loop hierarchy to the tag level and select a source tag.

3. In the **Entities** pane, select and right-click the blocks that you want to copy to other tag numbers.

4. On the shortcut menu, click **Copy Blocks to Tag Numbers**.

5. In the dialog box that opens, do one of the following to find tags belonging to the loops whose generation method is CAD:

   - Click **Find** to find all the tag numbers.

   - Set search parameters and then click **Find** to display tag numbers that match the search parameters.

6. Under **Search results**, select tag numbers to which you want to copy the blocks.

7. Click **OK**.

Note

- When any of the source blocks is associated with an instrument type, in the target tag number, the software changes the block association method from automatic to manual. In the **Loop Explorer**, the software changes the block icon from green 🟩 to red 🟥.

Dissociating Blocks from a Tag Number

Use this procedure to dissociate blocks from a tag number.]

➢ **To dissociate blocks from a tag number**

1. On the Loop Drawings module toolbar, click to open the **Loop Explorer**.

2. In the tree view pane, expand the loop hierarchy to the tag level and select a tag.

3. In the **Entities** pane, select and right-click the blocks that you want to dissociate from the tag number.

4. On the shortcut menu, point to **Actions** and click **Dissociate Blocks from Tag Number**.

Viewing Blocks

Use this procedure to display a block it in your CAD application — SmartSketch, MicroStation or AutoCAD — depending on the settings you made in the **Preferences** dialog box.

 Note

- The difference between viewing and generating is that viewing only shows the code fields of blocks and macros while generating actually retrieves the data and displays it on the drawing layer.

➢ **To view a block in a CAD application**

1. On the Loop Drawings module toolbar, click to open the **Loop Explorer**.

2. In the tree view pane, expand the loop hierarchy to the tag level and select a tag.

3. In the **Entities** pane, right-click a block.

4. On the shortcut menu, point to **Reports** and click **View Block in CAD Application**.

Changing Block Association Method

This option enables you to change the association method from automatic to manual.

After changing the block association method from automatic to manual, these blocks are no longer associated with an instrument type, and therefore are not affected by any changes you or other users can make to the block – instrument type association.

Also, manually associated blocks become available for selection in the **Generate Loop Drawings** dialog box, where you can generate a CAD loop drawing per block.

 Note

- You can also change the association method back to block – instrument type association..   Note, however, that blocks associated with tag numbers through the instrument type are not available for selection in the **Generate Loop Drawings** dialog box.

➢ **To change the block association method**

1. Do one of the following:

   - Press F7 to open the **Domain Explorer** and then expand the hierarchy to the tag level in the **Loops** folder.

   - On the Loop Drawings module toolbar, click to open the **Loop Explorer**.

2. Do one of the following:

   - In the **Entities** pane, select and right-click one or more loops.

   - In the tree view pane, select a specific loop and then in the **Entities** pane, select and right-click tag numbers that contain blocks assigned using the block – instrument type association method.

3. On the shortcut menu, point to **Actions** and click **Change Block Association Method**.

4. Click **Yes** when prompted.

 Note

- In the **Loop Explorer**, the block icon changes from green to red.

Macro Structure, Descriptions, and Conventions

Macros in SmartPlant Instrumentation: An Overview

Macros are used to extract data from a given loop or instrument. Each macro corresponds to specific information available in SmartPlant Instrumentation, for example, the color of a wire. SmartPlant Instrumentation provides you with an easy way to search the macro that needs to be placed on the blocks. The software also comes with a standard set of macros that can be customized to your needs.

The data can be retrieved to the loop drawing from:

- Domain definitions

- <Plant>, <area>, and <unit>

- Loop data and tag numbers in the Instrument Index module

- Wiring data

- Specifications data

- Loop drawing data

- Hook-up data

All of the standard SmartPlant Instrumentation macros are displayed in the **Macro Definitions** dialog boxes. In these dialog boxes you can also set your macro names.

If you have used the standard drawing blocks supplied with SmartPlant Instrumentation, or if you created your own drawing blocks with the predefined macros, SmartPlant Instrumentation recognizes these macros when generating the loop drawings, and you do not need to make any modifications to them. However, you may want to modify macros in the following circumstances:

- Customized Macro Definitions: If your drawings include macro names other than the predefined ones, you can customize the default macro definitions as required.

- User-defined macro functions: This option enables you to modify the display format of the macros in a drawing.

- External macro data sources: You can specify an external data source other than the SmartPlant Instrumentation database for displaying information defined by a macro.

The Macro Language

Component macro names consist of the macro name that comprises two parameters - **Macro Type** and **Macro Attribute**. The macro type describes the entity itself, as for the examples in the following table.

Macro Type	Entity Description
CS	Control system tag
TAG	Instrument
PNL	Panel
STRP	Strip
TERM	Terminal
W	Wire

The macro attribute applies to the selected macro type, and examples are shown in the following table for macro type TAG.

Attribute	Entity Description
NUM	Number
MFR	Manufacturer
TYP	Type
CLR	Color

The macro type and attribute are used to build the description of the macro, for example:

- An instrument number macro is named TAG_NUM.

- A terminal color macro is named TERM_CLR.

SmartPlant Instrumentation Macro Structure

This topic lists the most common SmartPlant Instrumentation macros and describes the structure of these macros. If you want to see the entire list of SmartPlant Instrumentation macros, you can generate a macro report in the Loop Drawings module.

If required, you can associate each macro with a macro function, which determines how the software performs string manipulation on the target data. Therefore, a macro function can retrieve the entire target data string or just part of the data string and display it on the generated drawing. Associating a macro function with a macro is optional. For details, see User-Defined Macro Functions.

The structure of a macro including a user-defined macro function is as follows:

[Macro_name.F1]

F1 is the macro function abbreviation. If you set a macro function as a default, the software automatically adds the abbreviation of the function to the macro in the actual CAD drawings (SmartSketch, AutoCAD and MicroStation). If you do not set a macro function as default, you need to add the macro function abbreviation manually on the drawing.

Certain SmartPlant Instrumentation macros that include a group level have the following structure:

[Macro_name.x]

The letter 'x' designates the group level. The group level depends on the number of macros in a group. The software creates a group when several macros of the same macro type are associated with a tag number. For example, when associating several tag categories with a tag number, the system combines all the TAG_CAT macros into a TAG_CAT.x group so that you set a separate level for each macro: TAG_CAT.1, TAG_CAT.2, TAG_CAT.3, and so forth.

Notes

- Within a macro name segment, using a period (.) is not allowed.

- Some of the Wiring module macros have a different structure. For details, see Wiring Macro Structure.

- Hook-up drawing macros are listed in the topic Hook-Up Macro Conventions.

Domain Data

Macro Name	Description
PROJ_NAME	Domain name
PROJ_DESC	Domain description
PROJ_NUM	Domain number
ENG_PROJ_NAME	Project name

Macro Name	Description
ENG_PROJ_DESC	Project description
OWNER_NAME	Owner name

Loop Information

Macro Name	Description
TYPE_GEN	Loop drawing generation method (C for CAD, E for Enhanced SmartLoop, S for SmartLoop, and M for manual)
LOOP_NAME	Loop name
LOOP_NUM	Loop number
LOOP_SERV	Loop service
LOOP_SUF	Loop suffix
LOOP_PRFX	Loop prefix
LOOP_NOTE	Loop note
LOOP_OLD	Old loop name
LOOP_PROC	Loop measured variable
LOOP_TYP	Loop type
LOOP_FUNC	Loop function
LOOP_PID_NAME	Loop P&ID drawing name
LOOP_UDF_C01 - LOOP_UDF_C100	Loop custom tables

Tag Number Information (Instrument Index)

Macro Name	Description
CMPNT_CERTIF_NAME	Instrument certification
TAG_CAT.x	Tag category, where 'x' designates the group level
TAG_NUM	Tag number (full)
TAG_SERV	Tag number service
TAG_PRFX	Tag prefix
TAG_SUFF	Tag number suffix
TAG_MFR	Manufacturer
MFR_COMPANY_IDENTIFICATION	Manufacturer company identification number
TAG_MOD	Model
MOD_COMPANY_IDENTIFICATION	Model company identification number
TAG_MODE_DESC	Model description

Macro Name	Description
TAG_NUMC	Numeric segment of tag number
TAG_NOTE	Tag number note
TAG_OLD	Old tag number
TAG_REM1	Remark 1
TAG_REM2	Remark 2
TAG_REM3	Remark 3
TAG_REM4	Remark 4
TAG_REM5	Remark 5
TAG_FUNC	Instrument type (ISA abbreviation)
TAG_INST	Instrument type (description)
LINE_NO	Line number
IO_TYP	System I/O type
TAG_LOC	Location
TAG_PIPE_CLASS	Pipe spec
TAG_STAT	Status
TAG_EQP	Equipment
PLNT_NAME	<Plant> name
AREA_NAME	<Area> name
UNIT_NAME	<Unit> name
UNIT_NUM	<Unit> number
UDF_C01 - UDF_C100	Custom field data (character)
UDF_N01 - UDF_N10	Custom field data (numeric)
UDF_D01 - UDF_D05	Custom data (date)
LINE_ UDF_C01 - LINE_ UDF_C100	Line custom fields
EQP_UDF_C01 - EQP_UDF_C100	Equipment custom fields
UDT_NAME1 - UDT_NAME16	Custom tables created by the Domain Administrator
UNIT_NOTE	<Unit> note

SmartPlant Electrical Tag Number Information

Macro Name	Description
CELL	Cell name
CIRCUIT	Circuit name
DWG_SCHEMATIC_NAME	Drawing schematic name

Macro Name	Description
ELECT_EQUIP_NAME	Name of associated electrical equipment (for example, control station)
ELECT_EQUIP_TYPE_NAME	Associated electrical equipment type name
ELECT_LOAD_EQUIP_NAME	Name of main electrical equipment (for example, motor)
ELECT_LOAD_EQUIP_TYPE_NAME	Main electrical equipment type name
FREQUENCY_NAME	Frequency value
FULL_LOAD_ CURRENT	Full load current value
PHASE1	Phase 1 of the associated bus
PHASE2	Phase 2 of the associated bus
PHASE3	Phase 3 of the associated bus
NUMBER_OF_PHASES	Number of phases value
OPERATING_MODE	Power supply operating mode value
POWER_DISTRIBUTION_BOARD	Power distribution board name
POWER_FACTOR_FULL_LOAD	Power factor full load value
POWER_SUPPLY_TYPE_FLAG	AC/DC power supply type
RATED_ACTIVE_LOAD	Rated active load value
RATED_APPARENT_LOAD	Rated apparent load value
RATED_REACTIVE_LOAD	Rated reactive load value
RATED_VOLTAGE_NAME	Rated voltage value
STARTING_CURRENT	Starting current value
TYPICAL_SCHEMATIC	Typical schematic name
XCOINCIDENCFACTOR	Xcoincidencfactor value
ZCOINCIDENCFACTOR	Zcoincidencfactor value
ZZCOINCIDENCFACTOR	ZZcoincidencfactor value

 Note

- In Enhanced SmartLoop reports and CAD drawings for a signal that applies to a circuit, the loop macros cannot retrieve main or associated electrical equipment names and types because the signal is not directly linked with these items. Furthermore, it is possible for a circuit to feed more than one item of equipment, and this is incompatible with the way that macros function, where each macro can retrieve only a single data value at a time. Likewise, the CIRCUIT macro works only for a signal linked directly to the circuit, therefore if the signal is linked to the electrical equipment, this macro cannot retrieve the circuit data.

Fieldbus Tag Number Information

Macro Name	Description
FB_TAG_NAME	Fieldbus tag name
FB_DC_CONSUMPTION	DC consumption
FB_DEVICE_ID	Device ID
FB_FD_ADDRESS	Field device address
FB_FUNCTION_BLOCK	Function block
FB_MIN_TRANS_LEVEL	Minimum transmit level
FB_OPER_VOLT_HIGH	Maximum operating voltage
FB_OPER_VOLT_LOW	Minimum operating voltage

Telecom Tag Number Information

Macro Name	Description
LOAD_WATT	Load power value (watt)
TC_FIRE_AREA	Fire area name
TC_LOCATION_LAYOUT	Location layout name
TC_SIGNAL_NAME	Signal level name
TC_EX_CODE	Field equipment explosion-proof code
TC_LINE_NUMBER_NAME	Telecom line number

To extract the user defined columns of the Instrument Index module, we have used the following convention for the macro names:

Character type macros correspond to the columns by placing the identifying number of the column in the UDF_Cxx format where xx designates the number. For numbers smaller than 10, use a leading zero. For example, in order to extract the second column data, use macro name UDF_C02. This number can have a range of 1 to 40.

Numeric type macros correspond to the columns by placing the identifying number of the column in the UDF_Nxx format where xx designates the number. For numbers smaller than 10, use a leading zero. For example, to extract the second column data, use macro name UDF_N02. This number can have a range of 1 to 10.

Date type macros correspond to the columns by placing the identifying number of the column in the UDF_Dxx format where xx designates the number. For numbers smaller than 10, use a leading zero. For example, to extract the second column data, use macro name UDF_D02. This number can have a range of 1 to 10.

Calibration Data

Macro Name	Description
TRIP_SET_POINT_HI	Trip set-point high
TRIP_SET_POINT_HIHI	Trip set-point high-high
TRIP_SET_POINT_LO	Trip set-point low
TRIP_SET_POINT_LOLO	Trip set-point low-low

Specifications

Macro Name	Description
SPEC_DWG_NUMBER	specification drawing number
SPEC_NOTE	specification note
SPEC_UDF_C01 – C100	specification custom tables

Loop Title Block Macros

This topic lists and describes macros used in a loop drawing title block.

Macro Name	Description
DWG_NAME	Document number
DWG_DESC	Drawing description

Revisions

These macros have the structure [MACRO_NAME.XX.F1] where XX is the revision number (max. value =5), and F1 is the abbreviation for a user-defined function (optional).

Macro Name	Description
DWG_REV_NUM	Drawing revision number
DWG_REV_DESC	Revision description
REV_DRAWN_BY	Revision by
REV_APPR_BY	Approved by
REV_CHK_BY	Checked by
DWG_REV_DATE	Revision approval date
CURR_REV_NUM	Last active revision

Approvals

Macro Name	Description
VEN_PROP_BY	Proposed by (vendor)
VEN_PROP_DATE	Proposal date (vendor)
VEN_DSGN_BY	Designed by (vendor)
VEN_DSGN_DATE	Designed date (vendor)
VEN_DRWN_BY	Drawn by (vendor)
VEN_DRWN_DATE	Drawn on date (vendor)
VEN_CHK_BY	Checked by (vendor)
VEN_CHK_DATE	Check date (vendor)
VEN_APPR_BY	Approved by (vendor)
VEN_APPR_DATE	Approval date (vendor)
CL_CHK_BY	Checked by (client)
CL_CHK_DATE	Check date (client)
CL_ENGR_BY	Instrument Eng. check (client)
CL_ENGR_DATE	Inst. Eng. check date (client)

Macro Name	Description
CL_APPR_BY	Approved by (client)
CL_APPR_DATE	Approval date (client)

Document References

You can display references to documents in the title block using the macros REF_NAME and REF_DESC. It is possible to retrieve to the loop up to ten references.

To extract the references, use the formats:

[REF_NAME.XX.F1] to get the document number, and [REF_DESC.XX.F1] to get the document description. The XX segment denotes a two digit number ranging from 1 to 10 and F1 is the abbreviation for a user-defined function (optional).

If you have more than one document number, in the title block, the software sorts the document numbers by name and displays them in ascending order. For example, if you have document numbers Document A, Document C, Document B, and Document A1, the software sorts them as follows:

Document A
Document A1
Document B
Document C

If you need to display only a specific document number , in the macro name REF_NAME.XX.F1, substitute the XX segment with 01 to display the first number sorted by name, 03 to display the third number, and so forth.

Drawing Generation Time and Date

Macro Name	Description
DATE	Date of generation
TIME	Time of generation

Wiring Module Macros - Conventions

To retrieve wiring information, SmartPlant Instrumentation uses macros. These macros are identified by their macro name, used with the appropriate parameters to retrieve specific tag data. The following paragraph explains how the software maps the wiring information associated with tag numbers.

When you create a tag number and a matching device panel, the software creates a Group that includes all the wires associated with that specific tag. Each wire can belong to one group only. Once you connect a cable with the device panel, the software sorts the wires into different group levels. Every connection of a wire to a terminal is assigned a **Group Sequence** that maps and numbers the connections starting with the value '1' that is usually the device panel side.

These are the principles required to know when building typical wiring graphic blocks:

- How to identify typical wiring routings.

- The number of routing levels, usually determined by the number of wires connected to the device-panel, but it might change for example in case a power supply is involved.

- The group sequence that each wire and terminal connection has in the current typical routing.

The following diagram is an example that illustrates this wiring data extracting method.
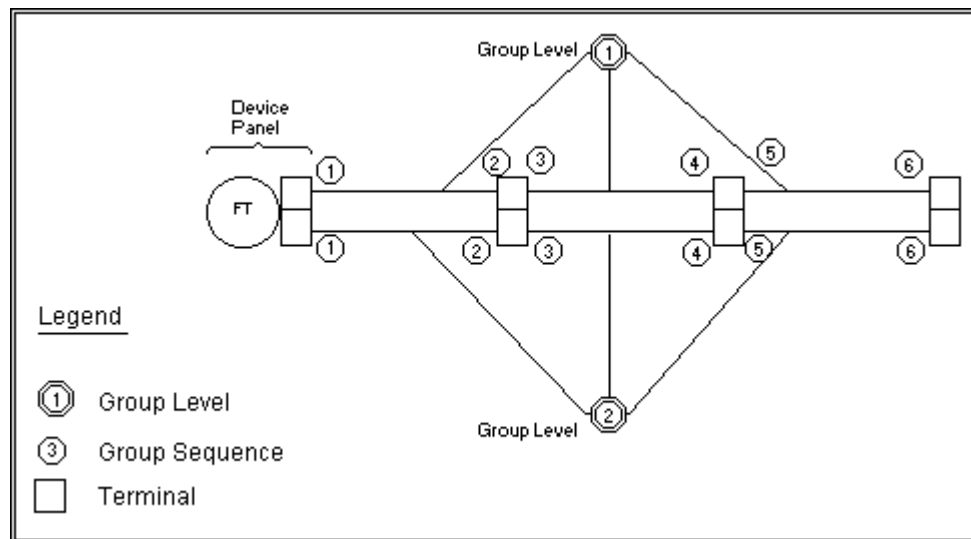
Examples

- The macro used for retrieving the terminal number for a group level 1 and Group Sequence 3 terminal is named TERM_NUM.1.3.

- The macro used for retrieving the color of a group level 2 and Group Sequence 2 wire is named W_CLR.2.2.

📝 **Note**

- In the above example, to extract the wire name for the wire at group level 2 whose ends have sequence numbers 3 and 4, you may use either macro W_TAG.2.3 or W_TAG.2.4.

Control System Tag Propagation

Propagation of a signal along wires may vary according to whether a field device exists and whether a tag number was assigned to an I/O channel. If the tag number was assigned to an I/O device the cables connected to the I/O terminals are also considered during the signal propagation. The method of propagating sequence numbers (within a level) used by SmartPlant Instrumentation depends on whether a field device is associated with that tag number, and whether the wiring routing is complete (there is a signal continuity between the field device and the I/O channel).
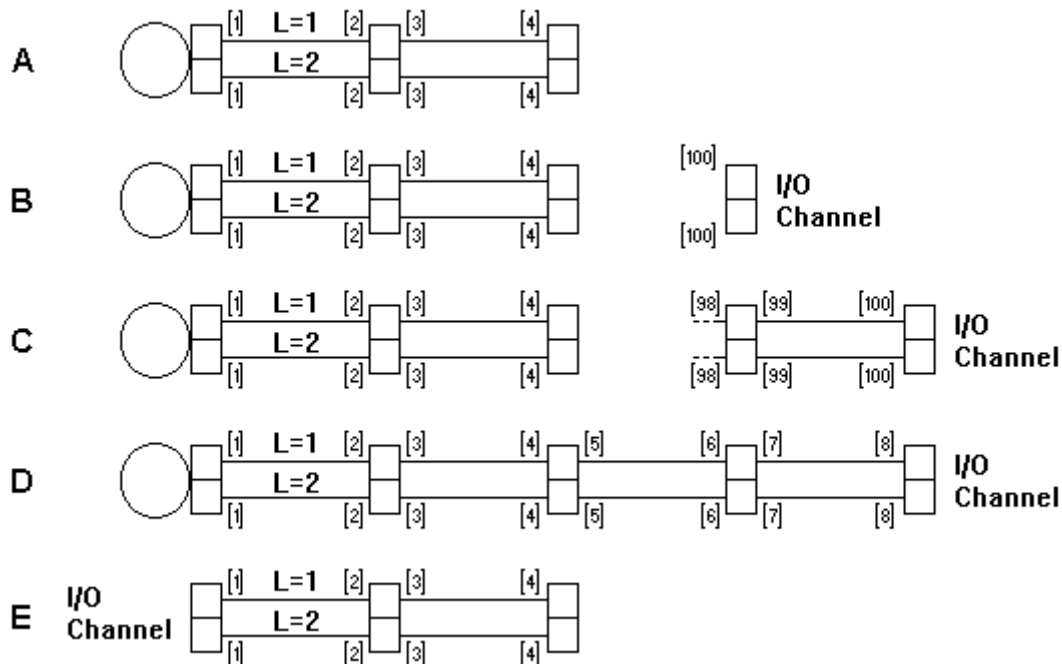
Diagram key:

A - Field device without a channel.

B - Field device and Control System channel assigned without cables connected to the Control System channel.

C - Field device and Control System channel assigned with cables. Wiring routing is not completed.

D - Field device and Control System channel assigned. Wiring routing is completed.

E - Control System channel assigned without a field device.

Wiring Macro Structure

Macro names are grouped according to their parent entities, for example, terminal, strip, panel, cable, set, wire and so forth. Each of these entities has a list of attributes (name, type, model, manufacturer and so forth). Wiring module macros are also addressed in AutoCAD and MicroStation drawings by their group level (designated by the letter x), and the group sequence (designated by the letter y). Therefore the overall structure of the Wiring module macros is as follows:

[Macro_name.x.y.F1]

Where F1 is the abbreviation for a user-defined function (optional).

Wiring Module Macros - Macro Name List

Terminals

Macro Name	Description
TERM_NUM.x.y	Terminal number
TERM_MFR.x.y	Terminal manufacturer
TERM_MDL.x.y	Terminal model
TERM_TYP.x.y	Terminal type
TERM_CLR.x.y	Terminal color
TERM_UDF_C01 - TERM_UDF_C20	Terminal custom fields

A special macro was designed to extract the data for a terminal connected to an overall shield. For the macro to function properly, two conditions must be followed:

- The shield flag is on the wire.

- The level and sequence of one of the wires in the cable is defined.

The macro name for this purpose has the format [OSHT_NUM.x.y.]

Terminal-Strips

Macro Name	Description
STRIP_CONTROLLER.x.y	I/O card controller name
STRIP_DIMENSIONS.x.y	Strip or I/O card dimension details
STRP_NAME.x.y	Terminal strip name
STRP_MFR.x.y	Strip manufacturer
STRP_MDL.x.y	Strip model
STRIP_RAIL.x.y	Strip or I/O card rail name
STRIP_SERIES.x.y	I/O card series name
STRP_TYP.x.y	Strip type
STRIP UDF_C01 - STRIP UDF_C20	Strip custom fields

Panels

Macro Name	Description
PANEL_DIMENSIONS.x.y	Panel dimension details
PNL_NAME.x.y	Panel name
PNL_MFR.x.y	Panel manufacturer

Macro Name	Description
PNL_MDL.x.y	Panel model
PNL_MOUNT.x.y	Panel mounting
PNL_TYP.x.y	Panel type
PNL_UDF_C01 - PNL_UDF_C100	Panel custom fields

Wires

Macro Name	Description
W_TAG.x.y	Wire name
W_POL.x.y	Polarity
W_CLR.x.y	Wire color
W_NOTE.x.y	Wire note
W_GRP.x.y	Wire group name
W_TYP.x.y	Wire type
W_UDF_C01 - W_UDF_C20	Wire custom fields

A special macro was designed to extract the wire name of a cable's Overall Shield. For the macro to function properly, two conditions must be followed:

- The shield flag should be on the wire.

- Know the level and sequence of one of the sets in the cable.

The macro name for this purpose has the format of [OSHW_TAG.x.y.]

Cable Sets

Macro Name	Description
SET_NAME.x.y	Set name
CABLE SET UDF_C01 - CABLE SET UDF_C20	Cable set custom fields

Cables

Macro Name	Description
CABLE_HARNESS.x.y	Cable harness name
CBL_NUM.x.y	Cable name
CBL_DESC.x.y	Cable description
CBL_LEN.x.y	Cable length

Macro Name	Description
CBL_EPNT1.x.y	Cable end point #1
CBL_EPNT2.x.y	Cable end point #2
CBL_MFR.x.y	Cable manufacturer
CBL_MDL.x.y	Cable model
CBL_TYP.x.y	Cable type
CBL_CLR.x.y	Cable color
DRUM_NAME.x.y	Cable drum name
DRUM_PULLING_AREA.x.y	Drum pulling area
ROUTING_SECTION_POS.x.y	Cable routing section/position
CBL_UDF_C01 - CBL_UDF_C100	Cable custom fields

User Defined Macro Groups

The macros listed below are user define macro groups to which you can assign values according to your needs.

Table Name	Macro Group Name
CABLE	CBL_UDF_C01 – 100
CABLE_SET	SET_UDF_C01 – 20
WIRE	W_UDF_C01 – 20
EQUIPMENT	EQP_UDF_C01 – 100
LINE	LINE_UDF_C01 – 100
PANEL	PNL_UDF_C01 – 100
STRIP	STRIP_UDF_C01 – 20
TERMINAL	TERM_UDF_C01 – 20
COMPONENT	UDF_SQL_C01 – 140

Control System Macros

In order to retrieve control system data into the loop drawings, you need to provide only the group level (sequence is not needed as a parameter).

The following macros are available for the control systems:

Macro Name	Description
CS_NAME	Control system tag number (CS_NAME.X)
CNL_NAME	I/O channel number (CNL_NAME.X)
CS_NET_NO	Network/Highway or Address 1 (CS_NET_NO.X)
CS_NODE_NO	Device/Node or Address 2 (CS_NODE_NO.X)
CS_MDL_NO	I/O module number (CS_MDL_NO.X)
RACK_NAME	Cabinet Rack Name (RACK_NAME.X)
RACK_DESC	Rack Description (RACK_DESC.X)
CS_POS	Strip I/O Type (CS_POS.X)
CNL_NAME	I/O Channel Number (CNL_NAME.X.Y)
CS_IO_TYPE	System I/O Type (CS_IO_TYPE.X)
CS_MDL_NO	System I/O Module Number (CS_MDL_NO.X)
CS_NAME	Control System Tag Name (CS_NAME)
SC_NET_NO	Network Number (SC_NET_NO.X.Y)
CS_POS	System Position (CS_POS.X)
CTERM_NUM	Seq. Term. Numb. in Channel (CTERM_NUM.X)
PNL_LOC	Panel Location (PNL_LOC.X.Y)
RCS_NET_NO	Remote Network Number (RCS_NET_NO.X.Y)
RCS_NODE_NO	Remote Node Number (RCS_NODE_NO.X.Y)
RCS_POS	Remote System Position (RCS_POS.X.Y)
RPNL_NAME	Remote Panel Name (RPNL_NAME.X.Y)
RRACK_DESC	Remote Cabinet Rack Desc. (RRACK_DESC.X.Y)
RRACK_NAME	Remote Cabinet Rack Name (RRACK_NAME.X.Y)
RSYS_CBL	Remote System Cable (RSYS_CBL.X.Y)
SCS_POS	Secondary System Position (SCS_POS.X.Y)
SRACK_DESC	Sec. Cabinet Rack Desc. (SRACK_DESC.X.Y)
SRACK_NAME	Sec. Cabinet Rack Name (SRACK_NAME.X.Y)
SRPNL_NAME	Sec. Remote Panel Name (SRPNL_NAME.X.Y)
SRSYS_CBL	Sec. Remote System Cable (SRSYS_CBL.X.Y)
CS_UDF_C01 -	Control System custom fields

Macro Name	Description
CS_UDF_C100	
CS_HEADER1	Panel address1 header
CS_HEADER2	Panel address2 header
MODULE_HEADER	Card header
POSITION_HEADER	Slot header

 Caution

- If you generate a loop drawing with tag numbers whose wiring is incomplete, you will **not** be able to retrieve all the required wiring information since the propagation is bi-directional: from the field device, and from the Control System.

- Wire sequence numbers will not be ordered sequentially until the propagation process finds signal continuity between the field device and the Control System.

Generating Macro Reports

This procedure enables you to generate a macro report of all SmartPlant Instrumentation macros used in loop drawings. The report displays the macro name, description, name used in SmartPlant Instrumentation and, if applicable, the macro function.

➢ **To generate a macro report**

1. Start the Loop Drawings module. .

2. On the **Reports** menu, click **Macros**.

3. In the **Macro Report Print Request** dialog box, do one of the following:

    - Click **No grouping** if you want to generate a report in which all the macros are displayed in alphabetical order.

    - Click **Group by module** if you want to generate a report in which the macros are grouped by module alphabetically.

 Note

- When you group the macros by module, the software organizes the report pages according to the module titles.

4. Click **OK** to generate a report.

Macro Definitions

Customized Macro Definitions

The database contains a number of default macros predefined for convenience of use and uniformity. If needed, you can customize the default macro names and descriptions. You can also associate a default macro name with a macro function. To extract SmartPlant Instrumentation data, you can use in the drawings both the default and the user -defined macro name of the same macro. A macro name can contain a maximum of 30 characters. To see the entire list of SmartPlant Instrumentation macros, you can generate a macro report.

Note

- If you want to associate a macro with an appropriate macro function, you can add a function abbreviation of up to 4 characters to a macro. The total number of characters, including separators (periods) is no more than 50.

Setting New Macro Definitions

When setting new macro definition properties, you can customize the name, description, and macro function of an existing default macro to fit your drawing requirements. This way you save the macro under the customized name while the default macro remains unchanged in the database. Also, other users working in the same domain, or in the same project (when the domain type is Operating owner) can use the customized macro definition.

🔆 Tip

- This option can be useful when you want to display the same data in several places on a drawing and apply different display formats to each of the iterations. (For details of how to apply the formats, see User-Defined Macro Functions.)

➢ To set new macro definition properties

1. Open the Loop Drawings module, and do one of the following:

   - On the toolbar, click 📇.

   - On the **Tables** menu, click **Macro Definitions**.

2. In the **Macro Definitions** dialog box, click **New** to add a new macro definition to the list of existing macro definitions.

3. In the **Macro Definition Properties (New)** dialog box, filter the macro display as follows:

   a) From the **Macro group** list, select a macro group for the required module.

   b) From the **Macro type** list, select a macro type to display the appropriate macro attributes.

   c) From the **Macro attribute** list, select a macro attribute to filter the displayed macro definitions.

4. In the data window, select the required default macro to be used as the basis for the new macro definition.

5. Under **User-defined properties**, in the **Name** field, type the required macro name.

6. In the **Description** field, if required, overtype the default macro description with your custom description.

7. If required, select a user-defined function to apply to the macro by doing the following:

   a) Click **Function**.

   b) In the **Assign User-Defined Macro Function** dialog box, select the required macro function and click **OK**.

8. Click **Apply**.

9. Click **Close** to return to the **Macro Definitions** dialog box and display the new macro definition in the data window.

10. In the **Macro Definitions** dialog box, click **Close**.

Modifying Macro Definition Properties

This procedure explains how to modify macro name, description, and macro function of an existing user-defined macro definition.

➢ **To modify macro definition properties**

1. Open the Loop Drawings module, and do one of the following:

   - On the toolbar, click 🖼.

   - On the **Tables** menu, click **Macro Definitions**.

2. In the **Macro Definitions** dialog box, filter the macro display as follows:

   a) From the **Macro group** list, select a macro group for the required module.

   b) From the **Macro type** list, select a macro type to display the appropriate macro attributes.

   c) From the **Macro attribute** list, select a macro attribute to filter the displayed macro definitions.

3. In the **Macro definition properties** data window, select the macro definition for which you want to modify the properties.

   💡 **Tips**

   - Select the **Show all macro definitions** check box to display all macro definitions that exist in the domain.

   - If there is a large number of macro definitions in the **Macro definition properties** data window, under **Find macro definition**, select a column heading and then type a value. For example, if you select Function as a column heading and type a macro function value, in the data window, the software locates the macro definition to which that macro function belongs.

4. Click **Properties** to open the **Macro Definition Properties** dialog box.

5. In the **Name** field, type the required macro name.

6. In the **Description** field, type the macro description.

7. If required, select a user-defined function to apply to the macro by doing the following:

    a) Click **Function**.

    b) In the **Assign User-Defined Macro Function** dialog box, select the required macro function and click **OK**.

8. In the **Macro Definition Properties** dialog box, click **OK** to display the modified macro definition in the data window of the **Macro Definitions** dialog box.

Deleting Macro Definitions

A macro definition consists of a macro name, description, and function. This procedure explains how to delete a macro definition from the list of macro definitions in the Loop Drawings module.

You can delete any macro definition. The list of macro definitions contains both the default and the user-defined macro definitions, set per domain, or per project (when the domain type is Operating owner).

Note

- If you delete a macro definition from the list, you do not delete the macro itself from the database, but only the macro definition properties. If required, you can restore a deleted macro definition.

➢ To delete a macro definition

1. Open the Loop Drawings module, and do one of the following:

   - On the toolbar, click .

   - On the **Tables** menu, click **Macro Definitions**.

2. In the **Macro Definitions** dialog box, filter the macro display as follows:

   a) From the **Macro group** list, select a macro group for the required module.

   b) From the **Macro type** list, select a macro type to display the appropriate macro attributes.

   c) From the **Macro attribute** list, select a macro attribute to filter the displayed macro definitions.

3. In the **Macro definition properties** data window, select the macro definition that you want delete.

   ### Tips

   - Select the **Show all macro definitions** check box to display all macro definitions that exist in the domain.

   - If there is a large number of macro definitions in the **Macro definition properties** data window, under **Find macro definition**, select a column heading and then type a value. For example, if you select Function as a column heading and type a macro function value, in the data window, the software locates the macro definition to which that macro function belongs.

4. Click **Delete**.

5. Click **Yes** when prompted.

Macro Functions

Standard Functions

A standard macro function is a function that is supplied with the software and contains a set of predefined commands. You can associate a standard function with SmartPlant Instrumentation macros or use a combination of standard functions to create user-defined macro function.

SmartPlant Instrumentation has six standard macro functions. The following table shows how each of the standard functions modifies the macro string.

Function	Description	Resulting String
Trim	Trims trailing spaces	FT - 100 a
Upper	Changes characters to upper case	FT - 100 A
Lower	Changes characters to lower case	Ft - 100 a
Substr (7,5)	Returns a substring of the original string	- 10 (Substring starts from 7th character and selects 5 characters including the starting character)
RemSpace (1)	Clears internal spaces when exceeding the max characters limit.	FT – 100 a (Where spaces exist, a maximum of 1 space is allowed)
RemSpace (0)		FT-100a (without spaces)

Note

- You cannot change the code of standard macro functions.

Standard Function Parameters

Some of the standard functions require additional parameters in their definitions, as shown.

Function	Parameters	Explanation
Substr	Start position Length	The position of the first character of the sub-string from the start of the string to be displayed, and the number of characters to be displayed in the sub-string
RemSpace	Max. contiguous spaces	Maximum number of internal spaces allowed in the string

User-Defined Macro Functions

You can perform string manipulation on the retrieved data by assigning user-defined functions to the macro that retrieved the data. You can create a user-defined macro function on the basis of SmartPlant Instrumentation standard macro functions.

The macro function affects the value which the macro returns after generation. For example, you can specify a user-defined function which removes all the space characters in the retrieved values and displays the remaining characters as upper case.

You can associate a function with a macro when setting new macro definitions or when modifying the existing macro definition properties. For more details, see the following topics:

- Setting New Macro Definitions

- Modifying Macro Definition Properties

There is a hierarchy of priority for which you can apply macro functions. For more information, see Macro Function Priority Hierarchy.

If you want to apply the highest macro function priority do the data retrieval, you can associate a function with a macro directly on the CAD drawing using the function abbreviation. The function abbreviation is a string with maximum of 4 characters, the first of which must be a letter. The following is an example of macros with functions:

W_CLR.1.1.RS

W_CLR.1.1.L

In this example, the macro name is W_CLR.1.1 and the macro function is RS or L. Use a period (**.**) separator to separate the macro name from the macro function.

Macro Function Priority Hierarchy

You can apply macro functions to macros directly on the CAD drawing or in the database.

The priority hierarchy for macro functions is as follows:

1. CAD drawing level — at this level, you type the macro function abbreviation directly in the generated drawing in your CAD application. At the CAD drawing level, you define a function per macro. A function that you specify directly on a CAD drawing has the highest execution priority.

2. Database level (when using a function with specific macro).

3. Database level (when using a default function with all macros). To learn how to set a macro function as default, see Creating User-Defined Macro Functions.

Creating User-Defined Macro Functions

Each user-defined macro function includes a set of selected standard functions. Use the following procedure to create user-defined macro functions at the database level. For more information about the levels at which you can use macro functions, see Macro Function Priority Hierarchy.

➢ **To create a user-defined macro function**

1. Open the Loop Drawings module, and on the **Tables** menu, click **User Defined Macro Functions**.

2. In the **User-Defined Macro Functions** dialog box, click **New** to open the **New User-Defined Function** dialog box.

3. Type a name and abbreviation for the new macro function.

4. Click **Insert** to open the **Standard Functions** dialog box.

5. Select the required standard function and click **OK**.

6. If you selected a standard function that requires parameters, specify the appropriate values in the respective fields.

7. Do one of the following:

   - Repeat the last three steps to add standard functions as required.

   - Use the spinners to change the order of the standard functions.

   - Click **Delete** to remove a selected standard function from the superposition.

8. Click **OK** to confirm your changes and close the **New User Defined Functions** dialog box.

9. Do one of the following:

   - Select **Set macro function as default** to use the specified macro functions with all the existing macros.

   - Clear **Set macro function as default** to be able to associate the specified macro function with a specific macro.

10. Click **Close**.

 Note

- The user-defined function operates in the macro string according to the order of the standard functions superposition.

Editing User-Defined Macro Functions

Each user-defined macro function comprises a set of selected standard functions. The following procedure shows how to edit user-defined macro functions at the database level.

➢ **To edit a user-defined macro function**

1. Open the Loop Drawings module, and on the **Tables** menu, click **User-Defined Macro Functions**.

2. In the **User-Defined Macro Functions** dialog box, click **Edit** to open the **Edit User Defined Function** dialog box.

3. Modify the user-defined function name, abbreviation, and description as required.

4. Click **OK**.

Generating a Report of User-Defined Macro Functions

This procedure enables you to generate a report of all the macro functions you created on the basis of SmartPlant Instrumentation standard macro functions.

➢ **To generate a User-Defined Macro Functions report**

1. Start the Loop Drawings module. .

2. On the **Reports** menu, click **User-Defined Macro Functions**.

External Macro Data Sources

Overview

You can use macros from an external source with the macros in your domain. This means that macro sources that are both external and internal in relation to SmartPlant Instrumentation are used when you generate a loop drawing.

Working with external macro sources includes the following steps:

1. Establishing a connection to an external data source from which SmartPlant Instrumentation can retrieve macro data.

2. In the **Domain Explorer** (or **Loop Explorer**, which is only accessible from the Loop Drawings module), assigning the connected external data source to the loop numbers for which you want to generate loop drawings.

Caution

- Before you start working with external macro data sources, make sure that the external source data naming and structure conventions comply with the SmartPlant Instrumentation macro conventions. For details, see Macro Structure Conventions and Macro Naming Conventions.

Macro Naming Conventions

The macro names for external macros in a drawing must be identical to the names in the external database file or table.

Note

- You define the external database in the **Define External Macro Source** dialog box.

To use external macros in your drawings, use the format E_<macro name>

Replace <macro name> with the name of the external macro. For example, a drawing contains the E_PROCESS external macro that retrieves the string 'Flow'. Therefore, after generating the drawing, every occurrence of the label E_PROCESS in the drawing will be replaced with the value Flow.

Macro Structure Conventions

SmartPlant Instrumentation has set a few basic requirements regarding the structure of external data sources from which you add macros to your loop drawings. Therefore, before you use an external macro make sure that its structure complies with SmartPlant Instrumentation conventions. The structure has to include the tag number (cmpnt_name) as the first (leftmost) column, and occupying the rest of the columns, the required macros and their retrieval values.

The following table is an example of an external source structure (the table headers below exemplify column names):

cmpnt_name	macro1	macro2	macro3
108-PI -2212	Pressure	Bar	Indicator
108-FE -2225	Flow	Ft^3/s	Element

In the above example, using the macro **macro1** in a loop drawing generated for the Tag Number 108-PI   -2212, will display the retrieved **Pressure** value instead of **macro1**. You can create the above structure by using any text editing application (for Text format files), dBase (*.dbf) editing tools, or any other appropriate database editing utility. See your database platform User Guide for further details.

 Caution

- The name of the first (leftmost) column name must always be cmpnt_name (Tag Number).

Connecting to an External Data Source

The first stage is connecting to the appropriate source data form whose macro data you want to use. After connecting to the database or the database file you can generate CAD drawings using the retrieved macros as required.

You can connect to any of the following database platforms and files:

- Oracle (only if you have selected Oracle as your current database platform during Setup or you installed an Oracle client manually)

- Microsoft SQL Server (only if you have select SQL Server as your current database platform during Setup or you installed a SQL Server client manually)

- Sybase Adaptive Server Anywhere (only if you have selected Sybase Adaptive Server Anywhere as your current database platform during Setup or you installed a Sybase Adaptive Server Anywhere client manually)

- Microsoft Access

- *.dbf

- ASCII delimited files

- Excel worksheets

- Text files

The currently available database profiles (in the **Data Source** list) are those that you have installed with the installation and those that you have manually installed.

Note

- You can connect to database platforms (Oracle, SQL Server or Sybase Adaptive Server Anywhere) other than your current database platform but you have to comply with the external database platform requirements.

    For example, If your current database platform is Sybase Adaptive Server Anywhere and you want to import data from an Oracle database platform, you must have Oracle client installed and running on your computer. See Internal Setup Utility to learn how to configure your environment to connect to database platforms other than your current database platform.

Prerequisites for Connecting to a dBase ODBC Profile

You can generate loop drawings in SmartPlant Instrumentation using your dBase (.dbf) external macro source. Prior to connecting to dBase (.dbf) external macro source, you need to create a dBase ODBC profile in accordance with the driver requirements of your Windows system.

The prerequisites are:

1. In your ODBC Data Source Administrator, select the ODBC driver in accordance with the your Windows system:

System	ODBC Driver
Windows XP	Microsoft dBase Driver
Windows 2000	PB Intersolv 2.12 32-BIT dBASEFILE (*.dbf)

2. Create an ODBC profile using your Windows ODBC Data source Administrator. See your database platform User Guide for further details.

Connecting to an ODBC Profile

This procedure describes the steps required to connect to a database file (for example, .dbf, ASCII, and so forth) using the ODBC profile. This way you can generate your CAD drawings using macros from an external macro source.

The following file types are available:

- dBase (.dbf)

- Excel worksheets

- ASCII delimited files

- Text files

 Note

- If you want to connect to dBase (*.dbf), you must create an ODBC profile in accordance with your Windows system. For more details, see Prerequisites for Connecting to a dBase ODBC Profile.

➢ **To connect to an ODBC profile**

1. On the **Actions** menu click **Define External Macro Source**.

2. In The **Define External Macro Source** dialog box, from the **Database type** list, select ODBC.

3. From the **Data source** list, select the source database file profile, for example, .dbf, ASCII files, and so forth.

4. Click **Browse** and select macro source file.

5. Click **OK**

Connecting to Database Platforms Other Than ODBC

This procedure describes the steps required to connect to a **database platform** such as Oracle or SQL Server. To connect to an external database platform, you must have access rights to the database server as well as the appropriate database drivers. You must also have the appropriate settings in the configuration files (for example, .ini files) and in the Windows registry.

➢ **To connect to database platforms other than ODBC**

1. On the **Actions** menu, click **Define External Macro Source**.

2. In The **Define External Macro Source** dialog box, from the **Database type** list, select the source database platform: Oracle or SQL Server.

3. In the **Server** box, type the name of the server that you use to connect to the database server:

    - <alias name> — if you connect to Oracle

    - <server name> — if you connect to SQL Server

4. In the **Database schema logon name** box, type the logon name required to connect to the source domain in the database server.

5. In the **Database schema logon password** box, type the logon password required to connect to the source domain in the database server.

6. Click **OK**.

Assigning an External Macro Data Source

This procedure enables you to assign an external macro data source to one or more loop numbers.

Note

- To be able to use an external macro source, you need to establish a connection to the macro source in the **Define External Macro Source** dialog box. Also, in your target drawing blocks or symbols, you need to add the **E_** prefix to the macros which you want to use to retrieve data from an external macro source.

➢ **To assign an external macro data source to one or more loop numbers**

1. Do one of the following:

   - Press F7 to open the **Domain Explorer** and then expand the hierarchy to the loop level in the **Loops** folder.

   - On the Loop Drawings module toolbar, click ⏹ to open the **Loop Explorer**.

2. Do one of the following:

   - In the tree view pane, right-click a desired loop.

   · In the **Entities** pane, select and right-click several loops.

3. On the shortcut menu, point to **Actions** and click **Assign External Macro Source**.

4. Under the **Macro Source** column, do one of the following:

   - If your macro source is an ODBC profile, type the exact path and the file name of the source database file.

   - If your macro source is a specific table on Oracle or SQL Server, you need to type the exact table name.

   Tip

   - If you want to assign the same macro source to all the loop numbers displayed in the **Assign External Macro Source** dialog box, select a row with the desired macro source and then select **Apply same source to all displayed loop numbers**.

5. Click **OK**.