General Features

Finding Tag Numbers: An Overview

In SmartPlant Instrumentation, you can search for tag numbers in different ways. If you need to find specific tag numbers, you define search parameters. When you do not specify any search parameters, the software displays all the existing tag numbers. You can either search for tag numbers in the current <unit>, or in all the <plant> <units>. You can also look for typical tags created in the current domain.

You search for tag numbers in the **Find Tag** dialog box. There are many cases where the software opens the **Find Tag** dialog box. For example, you can look for tag numbers after clicking **Find** in dialog boxes that prompt you to enter a tag number. Also, you can look for tag numbers when opening a process data sheet, when editing, duplicating, or deleting a tag number, or when calculating or calibrating tag numbers, and so forth.

The following search options are available:

- Searching for All Tag Numbers
- Using Search Parameters to Find Tag Numbers
- Searching for Typical Tag Numbers

Searching for All Tag Numbers

This topic explains how to find all the existing tag numbers in the current <unit>. In the Calibration and Process Data modules you can search for tag numbers in all the <plant> <units>. In the Process Data module, you can find tag numbers in all <units> when searching for tag numbers to generate instrument reports.

➢ To find all the existing tag numbers

1. In the **Find Tag** dialog box, do not enter any other values in the **Search parameter** fields.

   Note

   - In the Calibration or Process Data module, select **Look in all <units>** if you want to find tags in all the <units> of the current <plant>.

2. Click **Find**.

   Tip

   - Select the **Show more search results** check box to hide the search parameter fields and enlarge the **Search results** data window to display more tag rows. Clear this check box to return to normal view.

3. Select the required tag numbers in the **Search results** data window and click **OK**.

Searching for Typical Tag Numbers

This help topic explains how to find all the existing typical tag numbers in the current domain.

 **Note**

- This option is available only in the Instrument Index module.

➢ To find the typical tag numbers in the domain

1. When editing, deleting, duplicating or moving tags in the Instrument Index module, in the **Enter Tag Number** dialog box, click **Find**.
2. In the **Find Tag** dialog box, select Typical tag from the **Tag class** list.
3. Do not enter any other values in the **Search parameter** fields.
4. Click **Find**.

    **Tip**

   - Select the **Show more search results** check box to hide the search parameter fields and enlarge the **Search results** data window to display more tag rows. Clear this check box to return to normal view.

5. Select the desired tag numbers in the **Search results** data window and click **OK**.

Using Search Parameters to Find Tag Numbers

This help topic explains how to find tag numbers by entering search parameters.

Entering search parameters in the **Find Tag** dialog box enables you to narrow down your search. You can narrow your search to the maximum by entering all the search parameters. It is up to you what search parameters to specify. You can use wildcards in the fields where you type values.

Leaving some of the **Search parameter** fields empty widens the search. If you do not specify any search parameters, the software finds all the existing tag numbers.

After the search is complete, the software displays the tag numbers that match your search parameters in the **Search results** data window.

Note

- To specify search parameters in the Calibration module, you need to perform a separate procedure.

➢ To find a tag number using search parameters

1. In the **Find Tag** dialog box, from the **Tag class** list, select the desired tag class.

   Tip

   - A conventional tag is an instrument that does not belong to the Fieldbus, Telecom, Typical, or Electrical classes.

2. Use the following fields and lists in the **Search parameters** group box for search parameters that narrow your search:

Search Parameter	Explanation	Example
Tag number	Type the whole tag number you are looking for. Include any prefix, suffix, and separator characters. You can use wildcards if needed.	101-FT –2225/1 Or use a wildcard: 101-FT%
Process function	Select a process function to narrow your search to tags belonging to the selected process function.. (This parameter is available in the Instrument Index and Process Data modules only.)	Flow Pressure
Instrument type	Select an instrument type to narrow your search to tags belonging to the selected instrument type.	D/P Type Flow Element (FE), Mass Flow Transmitter (FT)

Search Parameter	Explanation	Example
Status	Select a tag status to narrow your search to tags associated with the selected status.	An existing device, a new instrument, a relocated device
Location	Select a tag location, for example, Field, to narrow your search to tags for which you have defined this location.	Equipment room, junction box
System I/O Type	Select a system I/O type to narrow your search to tags for which you have defined this system I/O type.	AO (analog output) DI (Digital input)
Prefix	Type the tag number prefix to find all the tag numbers that contain this prefix in their names. Do not include the separator characters. The <unit> number segment in the tag number name is usually the tag number prefix. You can also use wildcards if needed.	101
Number	Type the numeric segment of a tag number to find all the tag numbers that contain this numeric segment. You can also use wildcards if needed.	2315
Suffix	Type the suffix segment of the tag number to find all the tags that contain this suffix. Do not type the slash (/) character. You can also use wildcards if needed.	1 (the number following the slash (/) in tag number 101-FT –2225/1)
Equipment	Select equipment to narrow your search to tags for which you have defined this equipment.	
Line	Select a line to narrow your search to tags with which you have associated this line. (This parameter is not available in the Process Data and Specifications modules.)	
Form number	Select a specification form number to narrow your search to tags for which you have defined this form number. (This parameter is available in the Specifications module only.)	
Date range	Type dates or use spinners to narrow your search to a given date range. (This parameter is available in the Specifications module only.)	

3. To broaden the search to include all of the <units> in a given <plant>, select **Look in the entire <plant>**.

Note

- This field is available in the Calibration module, and in the course of various procedures in the Process Data and Specifications modules.

4. Click **Find**.

   Tip

   - Select the **Show more search results** check box to hide the search parameter fields and enlarge the **Search results** data window to display more tag rows. Clear this check box to return to normal view.

5. In the **Search results** data window, do one of the following:

   - Select the desired tag number.
   - Press and hold down **Ctrl** to select multiple tag numbers.

   Tip

   - Multi-selection is available in the Calculation module when you select tags for batch calculation, in the Process Data module for instrument report generation, and in the Specifications module. You can also select the **Select all** check box to select all the displayed tag numbers.

6. Click **OK**.

Finding Loop Numbers: An Overview

You can search for existing loop numbers in the current <unit>. You enter search parameters and find the required loop numbers that match the search parameters that you specified. You can find all the existing loop numbers in the current <unit> without specifying any search parameters or find the existing typical loops as needed.

The software opens the **Find Loop** dialog box whenever you need to search for a loop number. There are many cases where the **Find Loop** dialog box opens: clicking **Find** in a dialog box that prompts you to enter a loop number opens the **Find Loop** dialog box. For example, you can click **Find** when you want to edit or duplicate one or more loop numbers in the Instrument Index module.

The following search options are available:

- Using Search Parameters to Find Loop Numbers
- Searching for All Loop Numbers
- Searching for Typical Loops

Searching for All Loop Numbers

This help topic explains how to find all the existing loop numbers in the current <unit>.

➢ To find all loop numbers in the current <unit>

1. In the **Find Loop** dialog box, do not enter any values in the **Search parameter** fields.
2. Click **Find**.

Tip

- Select the **Display more results** check box to enlarge the **Search results** data window and display more tag rows. Clear this check box to return to normal view.

Using Search Parameters to Find Loop Numbers

This help topic explains how to find loop numbers by entering search parameters.

Entering search parameters in the **Find Loop** dialog box enables you to narrow down your search. You can narrow your search to the maximum by entering all the search parameters. Leaving some of the **Search parameter** fields empty will widen the search. It is up to you which search parameters to specify. If you do not specify any search parameters, the software finds all the existing tag numbers in the current <unit>.

Note that you can use wildcards in the fields where you type values.

After the search is complete, the tag numbers that match your search parameters are displayed in the **Results** data window.

➢ To find loop number in the Find Loop dialog box

1. Use the fields and lists in the **Search parameters** group box to enter your search parameters to narrow your search. The following search parameters are available:

Search Parameter	Explanation	Example
Loop name	Type the whole loop number you are looking for. Include any prefix, suffix, and separator characters. You can use wildcards if needed. The **Search Results** data window will display this loop number if it exists.	101F -2225\A 101F-%
Loop number	Type the numeric segment of the loop number to find all the loop numbers that contain this numeric segment. You can also use wildcards if needed.	2225
Loop prefix	Type the loop number prefix to find all the loop numbers that contain this prefix. Do not include the separator characters. The <unit> number segment in the loop number name is usually the loop number prefix. You can also use wildcards if needed.	101
Loop suffix	Type the suffix segment of the loop number to find all the loops that contain this suffix. You can also use wildcards if needed.	A
Measured variable	This search parameter is used to find all the existing loop numbers that have the measured variable that you select from this drop-down list.	Density (D) Pressure (DP)
Loop type	This search parameter is used to find all the existing loop numbers that have the loop type that you select from this drop-down list.	Electrical Loop (Electrical), Open Loop (Open)

Search Parameter	Explanation	Example
Loop function	This search parameter is used to find all the existing loop numbers that have the loop function that you select from this drop-down list.	Indication and Alarm (IA), Control (C)
Generation type	This search parameter is used to find all the existing loop numbers that used a specific type of loop drawing generation.	CAD (a loop drawing generated by an external CAD engine) SmartLoop (a loop drawing generated by SmartLoop) Manual (a loop drawing generated by using the manual method)

2. If required, select the **Fieldbus** check box to narrow your search to the loop numbers that contain tags with a fieldbus system I/O type.

3. Click **Find**.

4. Select one or more loop numbers in the **Search results** data window and click **OK**.

Note

- The **Select all** option is not available when duplicating an existing loop number.

Tip

- Select the **Show more search results** check box to hide the search parameter fields and enlarge the **Search results** data window to display more tag rows. Clear this check box to return to normal view.

Searching for Typical Loops

This help topic explains how to find all existing typical loops in the current domain.

➢ To find all typical loops numbers in the current domain

1. In the Instrument Index module, do one of the following:
   - Open the **Typical Loop Management** dialog box, where you can manage all your typical loop data.
   - Open the **Batch Loop Creation** dialog box and do the following:
     a) Click **Find**.
     b) In the **Find Typical Loop** dialog box, do not enter any search parameters and click **Find**.
   - Open the **Find Tag** dialog box and do the following:
     a) Click **Find** and select a tag.
     b) Select the **Find typical tags only** check box and click **OK**.
     c) In the **Tag Number** dialog box, click **Loop** to open the **Select Typical Loop** dialog box, where all the existing typical loops are displayed automatically.

Interface Language

Overview

You can replace the SmartPlant Instrumentation interface language. The System Administrator must first add languages to the database. You can purchase each language as a separate add-in. After adding a language to the database, you can replace your current language with the added one.

The following languages are currently available as add-ins:

- English (default)
- French
- German
- Custom

After the appropriate interface language is added, you can do the following:

- Replace the existing interface language with the imported language.
- Edit interface text phrases.
- Define new phrases to replace existing ones.

Replacing the Interface Language with a Language from the Database

This procedure explains how to replace the current SmartPlant Instrumentation interface with a language from the database.

➢ **To replace the interface language with a language from the database**

1. Start SmartPlant Instrumentation or the Administration module.
2. On the **Tools** menu, click **Select Language**.
3. In the **Select Language** dialog box, do one of the following:
   - From the **Language** data window, select the language with which you want to replace the current interface language.
   - Click **Default** to revert the language to the default language (English).
4. Do one of the following:
   - Select the **Optimize Speed** check box to speed up the translation process. This allows the software to load the interface text to your computer memory and retrieve it from there.
   - Clear the **Optimize Speed** check box to retrieve the interface text from the database. This frees the memory resources but decreases the performance.
5. Click **OK** to close the dialog box and change the interface language.

Replacing the Interface Language with a Language from an External File

Use this procedure to replace the interface language with a language from an external file. In this mode, all the terms and phrases are retrieved from a language file instead of being retrieved from the database, a change that improves the performance.

If you do not have an external language file, you can create one.

Caution

- Use this procedure only if you experience memory problems while working on your local computer.

➢ To replace the interface language with a language from an external file

1. Start SmartPlant Instrumentation or the Administration module.
2. On the **Tools** menu, click **Select Language**.
3. Select **Optimize speed**.
4. Select **Use file** to retrieve the interface text from a previously saved file (saved in .psr format), and do one of the following:
   - In the data field, type the path and filename of the appropriate language file.
   - Click **Browse** to navigate to the language file.
5. To update your language file, do one of the following:
   - Select **Overwrite existing file** to update the contents of the external language file after running an update for the application. This action overwrites all existing data in the language file with the data retrieved from the database.
   - Clear **Overwrite existing file** to leave the language file contents unchanged.
6. Click **OK** to close the dialog box and change the interface language.

Editing Interface Text Phrases

This procedure allows you to change the interface text by editing its phrases. At this stage, you change the phrases of the interface language, as described in the overview. Note that the interface text changes take effect only after exiting and re-entering the application.

Notes

- The System Administrator must first add the required language to the database. You can purchase a language that you require as a separate add-in.

Caution

- If you edit the phrases of an interface language that you have previously imported, you will **not** be able to revert to the imported interface language.
- When editing, make sure that you fill all the fields in the appropriate column. If you have a field blank, the phrase from the previous interface language remains after you switch the currently displayed language to the language that you are editing now. This can result in your interface text containing phrases or terms in more than one language.

➢ To edit the interface text

1. Start SmartPlant Instrumentation or the Administration module.
2. On the **Tools** menu, click **Edit Translation Text**.
2. Do one of the following:
    - Click **Open** to navigate to the language file that contains the interface text.
    - Click **Retrieve** to retrieve the interface text from the database.
3. Edit the text by clicking the text in the appropriate field.

    Note

    - You can modify the column sequence, for example, you can compare phrases more easily by moving two columns closer together. Drag a column header and drop it in the desired location. Then, click **Update** to save the changes to the database. If you want to save the changes to an external language file and not to the database, do not click **Update**.

4. To view the changes, replace the active language with the language that you have edited.

   Note

   - If you want to save the edited interface text to a language file (a file with a .psr extension), do the following:

     a) Click **Save File**.

     b) Type the path and filename of the language file or navigate to an existing language file.

5. Click **Save File** to save the edited interface text to a language file (a file with a .psr extension).

6. Type the path and filename of the language file or navigate to an existing language file.

7. Click **OK** in the displayed message and restart SmartPlant Instrumentation or the Administration module for the changes to take effect.

8. Click **OK** to save the changes and return to the **Edit Translation Text** dialog box.

9. Click **Close**.

Creating New Customized Text Phrases

This option enables you to replace the phrases in the current interface text with your customized text. To create customized phrases you use a special interface language called **Custom**.

After switching to the custom language, all the phrases in this column replace the original phrases of the interface language without overwriting them. Empty fields in this column do not affect the original phrases. This way you can always revert to the original phrases of the interface language.

Notes

- The System Administrator must first add the required language to the database.
- The interface text changes take effect only after exiting and re-entering the application.

➢ To create a user-defined interface language

1. Start SmartPlant Instrumentation or the Administration module.
2. On the **Tools** menu, click **Edit Translation Text**.
2. Do one of the following:
   - Click **Open** to navigate to the language file that contains the interface text.
   - Click **Retrieve** to retrieve the interface text from the database.
3. In the **Custom Phrase** column, type phrases you want.

   Notes

   - The column header **Custom Phrase** changes to **German Phrase** if you imported the German language, and to **French Phrase** if you imported the French language.
   - In the **Custom Phrase** column, each phrase that you type replaces the corresponding phrase in the interface text. If you leave a field blank, the corresponding field in the interface text remains unchanged. Make sure that you add the same prefixes, suffixes and punctuation marks that appear in the corresponding original phrases of that interface language (for example, ~, &). For details, see Prefixes and Suffixes in the Interface Text.

4. To view the changes, switch to the appropriate language.

Note

- After the changes are saved, you are prompted to restart SmartPlant Instrumentation for the changes to take effect.

6. Click **Save File** to save the edited interface text to a language file (a file with a .psr extension).
7. Type the path and filename of the language file or navigate to an existing language file.
8. Click **OK** in the displayed message and restart SmartPlant Instrumentation or the Administration module for the changes to take effect.
9. Click **OK** to save the changes and return to the **Edit Translation Text** dialog box.
10. Click **Close**.

Prefixes and Suffixes in the Interface Text

The following table contains information about the prefixes and suffixes that you can use when customizing new interface text phrases.

Prefix/ Suffix	Function	Syntax	Example
&	Specifies a menu item. Underlines the letter following the '&' symbol.	&[interface text]	&Action
&&	Displays the '&' symbol.	&&	Operators && Functions
~n	Starts a new line.	[interface text]~n[interface text]	~nContinue?
~r	Starts a new paragraph (functions like the Enter key in MS Word). It is usually used in conjunction with ~n.	[interface text]~r[interface text]	Warning~n~r
~t	Adds a tab entry.	[interface text]~t[interface text]	&Action~tCtrl+A

Document Revisions

Revision Management Overview

When managing revisions, you can add revisions to documents in SmartPlant Instrumentation, update them as needed, archive and compare new and existing document revisions, and delete obsolete revisions. A document is a report or drawing that has a document number. Also, you can add a document number when adding a revision to an entity.

The software allows you to add a revision to a specific report, drawing, or entity, for example, to a specific I/O card in the Wiring module. You add revisions to reports in the report print preview and to entities in the dialog boxes where you can edit the entity properties. Also, you can create global revisions.

In the Administration module, in the **Report Management** dialog box, the Domain Administrator has rights to define revision management settings individually for each report that you can generate in SmartPlant Instrumentation.

In the database, each report is assigned to the report type, which can be list or non-list. The report type determines how you can manage revisions created for a specific report, for an entity, or a group of entities. For list-type reports, the Domain Administrator can enable SmartPlant Instrumentation users to manage revisions either per document or per entity.

In accordance with the revision management setting, you can either create a revision whose document number and revision number become shared for a specific entity and for reports generated for that entity (when the setting is per-entity), or create a unique revision for a particular report (when the setting is per-document).

Regardless of the revision management setting, any document numbers and revisions that you create in the software are shared with the document numbers and revisions created for the same entities in the Enhanced Report Utility. For example, a drawing created in the Enhanced Report Utility automatically inherits the same document number and revision that you assigned to a SmartLoop drawing generated in SmartPlant Instrumentation.

Report Comparison Overview

Report comparison is available if you have saved revisions for a particular report. For every saved report revision, there is an archived report, stored according to the archiving option. The Domain Administrator defines an archiving option for each report.

When comparing reports, you can generate a comparison report to view the report fields in which the values are different.

Notes

- Report comparison is only available if the System Administrator has selected **Audit trail options** in the **Domain Definition** window for the current domain, and if the Domain Administrator has selected one of the options for saving revisions of the specified report in the **Report Management** dialog box in the Administration module.
- For enhanced reports, you must perform revisions from the Enhanced Report Utility if you want them to be archived for performing report comparison.
- You cannot perform report comparison for Powersoft browsers.

Using report comparison, you can do the following:

- Compare the currently previewed report with an archived report.
- Compare two archived reports.

Setting Font and Color for Report Comparison

This option enables you to set a comparison color and font style prior to comparing reports. The preset color and font style appear in a previewed report after comparing this report with an existing archived report. You can also set the grayscale and font style to appear in a printed report after report comparison. In a previewed report, the comparison color and font style indicate the differences between the previewed report and an archived report selected for comparison.

➢ To set font and color for report comparison

1. On the **Tools** menu of any SmartPlant Instrumentation module, click **Data Comparison Display Options**.
2. Beside the **Highlight color for display** field, click ....
3. In the **Color** dialog box, choose the color that you require.
4. Click **OK** to return to the **Data Comparison Display Options** dialog box.

   Note

   - The highlight color you have set appears after report comparison in any report print preview to indicate the differences between the current and the archived report.

5. Beside the **Grayscale** field, click ....
6. In the **Grayscale** dialog box, move the slider to the required position.
7. Click **OK** to close the **Grayscale** dialog box.

   Note

   - The grayscale you have set appears after report comparison in a printed report to indicate the differences between the current and the archived report.

8. Under **Font style**, do the following if required:

   a) Select **Bold** to mark the differences between the current and the archived report in bold.

   b) Select **Italic** to mark the differences between the current and the archived report in italic.

Comparing Archived Reports

This option enables you to compare between two archived reports.

➢ **To compare the currently previewed report with an archived report**

1. In the **Print Preview** window, do one of the following:
   - On the **Print Preview** window toolbar, click 
   - On the **View** menu, click **Compare Reports.**
2. In the **Report Comparison Options** dialog box, click **Compare two archived reports**.
3. Click **OK**.
4. In the **Select Archived Revisions for Report Comparison** dialog box, do the following:
   a) From the **Compare** data window, select the revision corresponding to the source archived report.
   b) From the **With** data window, select the revision corresponding to the target archived report that you want to compare with the source report.

 **Tip**

- If there is a long list of revisions in either of the data windows, type the required revision number in the **Revision filter** box and select **Activate**.

 **Note**

- If the archived and the currently previewed reports are different, in the previewed report the software marks the differences in color. If required, you can set your own comparison highlight color and font style to indicate the differences for report display. You can also set a grayscale and font style to indicate the differences in a printed report.

Comparing the Current Report with an Archived Report

The option enables you to compare the currently previewed report with an archived report.

➢ **To compare the currently previewed report with an archived report**

1. In the **Print Preview** window, do one of the following:

   - On the **Print Preview** window toolbar, click  .
   - On the **View** menu, click **Compare**.

2. In the **Report Comparison Options** dialog box, click **Compare the current report with an archived report**.

3. Click **OK**.

4. In the **Report Revision Archive** dialog box, select the revision corresponding to the archived report that you require.

   Tip

   - If there is a long list of revisions in the data window, type the required revision number in the **Revision filter** box and select **Activate**.

5. Click **OK** to reopen the **Print Preview** window.

   Note

   - If the archived and the currently previewed reports are different, in the previewed report the software marks the differences in color. If required, you can set your own comparison highlight color and font style to indicate the differences for report display. You can also set a grayscale and font style to indicate the differences in a printed report.

Maintaining a Document Revision Archive

You use the **Document Revision Archive** to archive reports and their revisions.

Note

- The software archives revisions for browser view reports separately for each view regardless of any existing filter or sort sequence.

➢ To set up an active archive for a report

1. Start the Administration module and enter as the Domain Administrator.
2. On the **Activities** menu, click **Report Management** to open the **Report Management** dialog box.
3. From the **Archiving Options** list, for the appropriate report, select the one of the following options:
   - **Do not save** — Does not archive revision and report.
   - **Save to database** —Saves revision and report to the database.
   - **Save to file** — Saves revision and report to a file in the .psr format. Click and specify a path to save the report in a .psr format.
   - **Save to Zip file** — Saves revision and report in a compressed .zip file. Click and specify a path to compress the report in a .zip format.
4. Click **OK** and close the Administration module.
5. In the SmartPlant Instrumentation application, save a revision for the desired document.

   Caution

   - For enhanced reports, you must perform revisions from the Enhanced Report Utility if you want them to be archived for performing report comparison or viewing a changes report.

6. In the **Print Preview** window, click to open the **Document Revision Archive** dialog box where you view the revision.
7. Double click on the required revision or revisions to open the print preview.

Viewing Archived Reports

This option enables you to open a report preview of any archived report. The software automatically archives reports when saving report revisions for the currently previewed report. The Domain Administrator defines an archiving option for each report.

➢ To view an archived report

1. In the **Print Preview** window, do one of the following:

   - On the **Print Preview** window toolbar, click 
   - On the **View** menu, click **Report Revision Archive**

2. In the **Report Revision Archive** dialog box, select the revision which corresponds to the archived report you want to view.

   Tip

   - If there is a long list of revisions in the data window, type the required revision number in the **Revision filter** list, and select **Apply**.

3. Click **OK** to open the archived report in the **Print Preview** window.

Displaying Changed Documents

This option enables you to display a list of documents for which data modifications have been made.

Notes

- To use this option, the Domain Administrator must first select the **Save Document Data** option in the **Report Management** dialog box for each report that you want to include in the list.
- The software does not include in changes reports any modifications that you make to title block macros or external symbols such as DCS data or non-wiring tag lists in enhanced reports.
- The software does not display data changes:
  - if you add a new entity such as a panel or gland in list reports, because the software cannot determine whether the entity is part of the document.
  - if you remove or assign tags or items in the Hook-ups module.
  - for the Instrument Index Drawing Summary Browser.
- Changes that you make in the Instrument Index Standard Browser are included in the **Browse - Instrument Index Report** item, and not in the **Instrument Index** under the list of browsers.

➢ To display list of changed documents

1. On the **Tools** menu, click **Changed Documents**.
2. In the **Changed Documents** dialog box, do one of the following:
   - Under **Document types**, highlight the desired document types. For multiple selection, hold down Ctrl or Shift while making your selection.
   - Select the **Select All** check box.

3. Under **Filter by**, select an option to specify the method of filtering. The available options are:

   - **Documents changed since last revision** - select to filter documents for which data was changed since the last revision of the document. If there is no revision, the software does not display that document.

   - **Documents changed between specified dates** - select to filter documents for which data was changed during the date range specified by the **From date** and **To date** fields.

4. Click **Find** to retrieve the list of changed documents according to the document types and filter options that you specified.

5. Click **Changes Report** if you want to display a report of the changed documents.

Global Revisions Overview

Global revisions enable you to perform a revision activity in batch mode.

At the first stage of applying global revisions, you select a revision activity, and define the default revision settings, such as revision values, numbering method, and revision details. At the second stage, you select the required entities in a specific module, and apply the defined revision settings.

Revision activities are:

- Add revision — select the required numbering method and add a new revision
- Update revision — change the existing revision value without changing the numbering method. For example, you can update a revision from A1 to A2.
- Upgrade revision — involves changing the revision numbering method. For example, you can upgrade a revision from P0 to A.
- Delete revisions — delete all revisions for the selected entities.
- Delete last revision — delete only the last revision.

You can add global revisions to non-list-type reports reports for which the Domain Administrator selected the Per Document revision management setting in the **Report Management** dialog box.

An exception is SmartLoop drawings, that are always assigned to the Per Entity revision management setting. The document (drawing) number and global revision that you create in SmartPlant Instrumentation are shared with the document number and revision created in the Enhanced Report Utility. Reports created in the Enhanced Report Utility automatically inherit the same global revision settings that you assign to SmartLoop drawings generated in SmartPlant Instrumentation.

Filtering Entities for Global Revisions

You can filter the entities in the data window of the **Global Revisions** dialog box when adding, updating, upgrading, or deleting revisions in batch mode.

On the **Wiring** tab of the **Global Revisions** dialog box, you can also filter the Wiring module entities per report.

➢ To filter entities for global revisions

1. In the **Global Revisions** dialog box, click the appropriate tab to make one of the following tabs active: **Specifications**, **Process Data**, **Calculations**, **Loops**, **Hook-Ups**, or **Wiring**.

2. If you selected the **Wiring** tab, do the following:

   a) Under **Filter parameters**, from the **Report for** list, select the entity for which you want to apply revisions: Panels, Cables, or DCS/PLC.

   b) From the **Report type** list, select a report type appropriate for the entity you selected in the previous step.

   Tip

   - To set new filter parameters, click **Clear** first.

3. For all tabs, under **Filter parameters**, type values in one or more of the text boxes to specify the parameters that you want to use for filtering the data.

4. Select the **Display current <unit> data only** check box to display the data in the current <unit>. Clear this check box to display all data at the level of the current <plant>.

   Note

   - On the **Hook-Ups** tab, data is filtered at the <plant> level only.

5. Select the **Activate filter** check box to apply the filter parameters on the entities.

   Tip

   - If you select the **Activate filter** check box first, the software filters the data as you select or type the filter parameters.

6. Click **Refresh** to update the data window as needed.

7. Click **Apply**.

Adding Global Revisions

This option enables you to add a global revision to entities that belong to any of the following modules: Specifications, Process Data, Calculation, Loop Drawings, Hook-Ups, and Wiring.

➢ To add a global revision

1. In the main window of any SmartPlant Instrumentation module, on the **Tools** menu, click **Global Revisions**.

2. On the **Settings** tab of the **Global Revisions** dialog box, from the **Activity** list, select Add revision.

3. In the **Revision** field, enter the required revision value.

4. Do the following to enter the revision details:

   - In the **Revised by** field, enter the initials of the person performing the revision. The default is the current user's initials, if previously defined by the System Administrator.

   - In the **Date** field, enter the date of revision, if required.

   - If required, fill out the other fields as needed.

5. Click the appropriate tab to make one of the following tabs active: **Specifications**, **Process Data**, **Calculations**, **Loops**, **Hook-Ups**, or **Wiring**.

    **Note**

   - The tabs are only active for other modules if you accessed the **Global Revisions** dialog box from the main SmartPlant Instrumentation window; other modules are not available if you accessed the dialog box from a specific module.

6. In the data window, filter the entities as needed.

7. In the data window, select the entities for which you want to add a global revision defined on the **Settings** tab by doing one of the following:

   - Select the required tag numbers (hold down **Ctrl** or **Shift** to make multiple selections).

   - Select the **Select all** check box to select all the tag numbers in the data window.

8. Click **Apply** to add a revision to each of the selected entities.

9. If you want to add revisions for other modules, repeat the appropriate steps for each module.

Defining Settings for Upgrading Revisions

This option enables you to define default settings for upgrading revisions globally. These settings include revision numbering method, and revision details.

➢ **To define default settings for upgrading revisions globally**

1. With any main window open, on the **Tools** menu, click **Global Revisions**.
2. On the **Settings** tab, from the **Activity** list, select Upgrade revision.
3. Define the revision numbering as follows:

   a) To change the numbering method for entities that do not currently have revisions, from the **Initial revision numbering method** list, select one of the following numbering methods you want to start with:

   - P0: for preliminary number sequence P0, P1, P2,…
   - 0: for number sequence 0, 1, 2,…
   - A: for number sequence A, B, C,...

   Note

   - The initial revision numbering method only affects entities which do not have revisions. For entities that already have revisions, the system adds a new revision line using the **existing** revision numbering method.

   b) To change the numbering method for entities that currently use the preliminary revision numbering method (P0, P1, P2,...), select the **Switch to another numbering method** check box, and from the **Continue numbering using** list, select one of the following new revision numbering methods:

   - 0: for number sequence 0, 1, 2,…
   - A: for number sequence A, B, C,…

4. Do the following to enter the revision details:

   - In the **Revised by** field, enter the initials of the person performing the revision. The default is the current user's initials, if previously defined by the System Administrator.
   - In the **Date** field, enter the date of revision, if required.
   - If required, fill out the other fields as needed.

5. Click the appropriate module tab to apply the defined settings.

Note

- The tabs are only active for other modules if you accessed the **Global Revisions** dialog box from the main SmartPlant Instrumentation window; other modules are not available if you accessed the dialog box from a specific module.

Upgrading Revisions Globally

After defining the default settings for upgrading revisions, you use this option to apply these settings to the selected entities in batch mode.

➢ To upgrade revisions globally

1. Define the settings for upgrading revisions globally.
2. Select the desired module tab and filter the entities as needed.
3. In the data window, do one of the following to select the entities for which you want to upgrade revisions:
   - Select the required tag numbers (hold down **Ctrl** or **Shift** to make multiple selections).
   - Select the **Select all** check box to select all the tag numbers in the data window.
4. Click **Apply** to apply the new revision value to all the selected entities in batch mode.
5. If you want to upgrade revisions for other modules, repeat the appropriate steps for each module.

Updating Revisions Globally

You can globally update the existing revision values defined for the entities used in the following modules: Specifications, Process Data, Calculation, Loop Drawings, Hook-Ups, and Wiring.

➢ To update revisions globally

1. In the main window of any SmartPlant Instrumentation module, on the **Tools** menu, click **Global Revisions**.

2. On the **Settings** tab of the **Global Revisions** dialog box, from the **Activity** list, select Update revision.

3. In the **Revision** field, enter the required revision value.

4. Do the following to enter the revision details:

   - In the **Revised by** field, enter the initials of the person performing the revision. The default is the current user's initials, if previously defined by the System Administrator.

   - In the **Date** field, enter the date of revision, if required.

   - If required, fill out the other fields as needed.

5. Click the appropriate tab to make one of the following tabs active: **Specifications**, **Process Data**, **Calculations**, **Loops**, **Hook-Ups**, or **Wiring**.

    **Note**

   - The tabs are only active for other modules if you accessed the **Global Revisions** dialog box from the main SmartPlant Instrumentation window; other modules are not available if you accessed the dialog box from a specific module.

6. In the data window, filter the entities as needed.

7. In the data window, do one of the following to select the entities whose revisions you want to update:

   - Select the required tag numbers (hold down **Ctrl** or **Shift** to make multiple selections).

   - Select the **Select all** check box to select all the tag numbers in the data window.

8. Click **Apply** to apply the new revision value to all the selected entities in batch mode.

9. If you want to update revisions for other modules, repeat the appropriate steps for each module.

Revision Upgrade Example

This example is based on the following parameters:

- Initial numbering method: P0
- Switch to numbering method for preliminary revisions: A

With the above parameters, the following results are obtained for subsequent revisions (on clicking **Apply** in the module sub folders of the **Global Revisions** dialog box):

Previous	1	2	3	Comments
None	P0	A	B	First revision uses initial numbering method (P0); subsequent revisions use switch to numbering method (A, B, and so forth.)
P2	A	B	C	First and subsequent revisions use switch to numbering method (A, B, and so forth.)
A	B	C	D	Incremented by one step each time from the previous revision
B	C	D	E	Incremented by one step each time from the previous revision
1	2	3	4	Incremented by one step each time from the previous revision
2	3	4	5	Incremented by one step each time from the previous revision

Deleting Global Revisions

You can perform batch deletion of revisions if you have the appropriate access rights. You have the option to delete all revisions for the selected entities in the specified module, or delete only the last revisions.

➢ **To delete revisions**

1. In the main window of any SmartPlant Instrumentation module, on the **Tools** menu, click **Global Revisions**.

2. On the **Settings** tab of the **Global Revisions** dialog box, from the **Activity** list, select one of the following:

   - Delete revisions — select to delete **all** revisions for the selected entities.

   - Delete last revision — select to delete the last saved revision for each selected entity.

3. Click the appropriate tab to make one of the following tabs active: **Specifications**, **Process Data**, **Calculations**, **Loops**, **Hook-Ups**, or **Wiring**.

   Note

   - The tabs are only active for other modules if you accessed the **Global Revisions** dialog box from the main SmartPlant Instrumentation window; other modules are not available if you accessed the dialog box from a specific module.

4. In the data window, filter the entities as needed.

5. In the data window, do one of the following to select the required entities:

   - Select the required instrument tags (hold down **Ctrl** or **Shift** to make multiple selections).

   - Select the **Select all** check box to select all the tags in the display.

6. Click **Apply** to delete the revisions for the selected entities.

7. If you want to delete revisions for other modules, repeat the appropriate steps for each module.

Adding Local Revisions

You can add revisions locally to a specific SmartPlant Instrumentation document or entity, depending on revision management settings defined by the Domain Administrator. For all list-type reports, the revision management setting is always Per Document. For certain non-list-type reports, the Domain Administrator has rights to define the setting as either Per document or Per Entity.

➢ To add a revision for a document or an entity

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

   Note

   - If you select a revision numbering method other than preliminary revisions (P0, P1, P2…), you will not be able to return to the preliminary revision method and this option will be disabled.

2. If needed, in the **Drawing number** field, type the number of the document associated with the entity for which you create the revision. In drawings and reports, this number appears in the title block.

3. Click **New** to add a new revision.

4. Add or edit the revision data according to the following table.

Field	Description
No	The revision number. This is incremented automatically according to the revision method selected, you can overwrite it if required.
By	Type the initials of the person performing the revision. The default is the current logged-in user initials, if previously defined by the System Administrator.
Date	The default is today's date, you can modify it if required.
Description	Description of the changes made or the purpose of the revision.
Checked By	Type the name of the person checking the revision, if required.
Approved By	Type the name of the person approving the revision, if required.

Note

- If the document for which you are making a revision contains a custom-title block, the **User-defined fields** data window becomes available for you. In this data window, you can view the user-defined fields that you added to the custom title block when creating the title block. To learn more about custom title blocks, see Customizing Title Blocks for Reports: An Overview.

Editing Local Revisions

Use this procedure to edit existing local revisions.

➢ To edit a revision for a report or entity

1. In the **Revisions** dialog box, select the revision that you want to edit.
2. Click **Edit**.
3. Make the required changes according to the revision table.
4. Repeat steps 2 and 3 for other revisions as required.
5. Click **OK** for your changes to apply.

Deleting Local Revisions

The **Revisions** dialog box enables you to remove existing revisions from a selected document or entity.

➢ To delete a revision

1. In the **Revisions** dialog box, select the revision that you want to delete.
2. Click **Delete**.

Workflow Procedures

Workflow Overview

The Workflow option enables instrument engineers to control the data entry process for instrument tags by specifying if and when process data should be entered for a particular tag. Workflow defines an additional level of access rights for individual instrument tags, determined by the Workflow status of the tags. In this way, an instrument engineer, while working on instrument data for a particular tag, can lock out process engineers from making changes to process data until all the instrument data has been entered.

Example Scenario

The following stages describe a typical Workflow scenario:

1. An instrument engineer creates a new instrument tag and determines whether or not the tag requires process data. In the event that the instrument tag **does** require process data, the process data fields of the tag become available to the process engineers for editing.

2. The process engineer, after first accessing the instrument tag for editing of process data, can lock the process data fields against any changes wherever the instrument engineers have access rights.

3. On being released by the process engineer, the instrument engineer can now enter instrument data as required. The instrument engineer can lock the tag against any changes wherever the process engineers have access rights.

Workflow Prerequisites

Before you can implement Workflow in a given <unit>, the System Administrator and Domain Administrator need to perform the following tasks:

- The System Administrator enables Workflow in your Domain. For more information, see Enable Workflow in Online Help for the Administration module.
- The Domain Administrator defines two groups that will be responsible for working in the areas of process engineering and instrument engineering. For more information, see Create a New Group Profile in Online Help for the Administration module.
- The Domain Administrator does one of the following:
  - Associates Windows groups with the SmartPlant Instrumentation process engineering and instrument engineering groups. For more information, see Create Groups for Windows Authentication Login in Online Help for the Administration module.
  - Assigns users individually to the process engineering and instrument engineering groups. For more information, see Assign Users to Groups in Online Help for the Administration module.
- The Domain Administrator defines Workflow access rights for the process engineering and instrument engineering groups. For more information, see Workflow Access Rights in Online Help for the Administration module.

Note

- After initialization of a database from a source or after a version upgrade, it is **essential** to regenerate the library forms to enable the Workflow option to be used in the Specs Browser.

Creating a Workflow Browser View

Workflow needs to be set up by an instrument engineer. This involves creating an appropriate view in the Workflow Browser from which the process data statuses of the instrument tags can be set.

Note

- Make sure that the System Administrator and the Domain Administrator have prepared SmartPlant Instrumentation for Workflow. For more information, see Workflow Prerequisites.

➢ To set up Workflow

1. Log on to SmartPlant Instrumentation as a user in the **Instrument** group and start the Browser module.
2. In the **Browser Manager**, under **Browser groups**, double-click **Workflow** to expand the hierarchy.
3. Select the **Instrumentation/Process Data Browser** and create a new view for it.
4. Double-click the view to expand the hierarchy.
5. Select the **Style** icon, and in the **Style settings** section, click **Edit**.
6. Select the check boxes in the **View** column to specify the fields to be displayed in the view.

   Note

   - You must include the **Process Data Status** field among the fields selected for displaying in the view.

7. Click **Save**.
8. Select the level of the view in the tree and on the **Actions** menu, click **Open View**.

   The view opens showing the process data statuses for all the instrument tags in the database.

Implementing Workflow

Workflow implementation begins when the instrument engineer who is going to work on particular instrument tags determines whether those tags require process data. Next, those tags that do require process data are assigned to a process engineer for process data input. On completion of the data input, the tags are reassigned to the instrument engineer for instrument data input.

➢ To enter initial instrument data

1. Log on to SmartPlant Instrumentation as a user in the **Instrument** group.
2. In the Instrument Index module, create the instrument tags as you require and enter appropriate data.
3. In the **Browser Manager**, under **Browser groups**, double-click **Workflow** to expand the hierarchy.
4. Refresh the screen by clicking  .

   Any new instrument tags should be visible, with default status: Process Data not Required.
5. In the **Process Data Status** field, select the required status of each instrument tag you will be editing as follows:

   a) For instrument tags that require process data, select status: Process Data Required.

   These tags will be available for the process engineer in the Process Data and Calculation modules.

   b) For instrument tags that do not require process data, select status: Process Data not Required.

➢ To enter process data for instrument tags

1. Log on to SmartPlant Instrumentation as a user in the **Process** group.
2. In the Process Data module, for each tag that requires the addition of process data, do the following:

   a) Find and open the tag.

   b) In the **Process Data** window, enter process data as you require.

   c) On the **Actions** menu, click **Save Process Data**.

   d) In the **Select a New Process Data Status** dialog box, select Release to Instrument, and click **OK**.

➢ **To enter additional instrument data**

1. Log on to SmartPlant Instrumentation as a user in the **Instrument** group.
2. In the **Browser Groups** list, **Workflow**, select the **Instrumentation/Process Data Browser** and open the view you created for it.
3. For each tag for which you want to edit instrument data, make sure that the value under **Process Data Status** is Lock out from Process.
4. In the Instrument Index module, do one of the following:
   - Open each tag individually and enter the required instrument data.
   - Create an appropriate Instrument Index browser view from which to enter the data.

Process Data Statuses

There are five statuses that relate to tags under Workflow. The following table provides a description of each status and which groups (instrument engineers or process engineers) are allowed to set and change the status in the Workflow > Instrumentation / Process Data Browser and in the Process Data module:

Status	Description	Set / Changed by
Process Data not Required	Tags that do not require process data and therefore are not available to the process engineering group.	Set and changed by instrument engineers
Process Data Required	Tags available for process data entry by the process engineering group (in the Process Data module or Browser).	Set by instrument engineers Can be changed by process engineers
Lock out from Instrument	Tags marked for editing by the process engineering group and not available to the instrument group.	Set and changed by process engineers
Release to Instrument	Tags available to the instrument engineering group following release from the process group.	Set by process engineers Can be changed by instrument engineers
Lock out from Process	Tags not available to the process engineering group.	Set and changed by instrument engineers

Print and Save Options

Printing and Saving Documents: An Overview

You can print or save documents containing essential data from every module. Documents that can be printed include all reports, and some visual presentations of data such as SmartLoop drawings and point-to-point wiring diagrams. Selecting to preview a document ensures that it will be printed the way you want and enables you to specify the file format if you want to save the document.

Previewing Documents

The option to preview a document is available whenever you select a report, a SmartLoop drawing, or a point-to point wiring diagram for printing, or if you select the **Print Preview** option from a browser view. The **Print Preview** dialog box appears only after you select Always or Ask user in the **General** tab of the **Preferences** dialog box.

➢ To preview a document

1. Select the report you want to display and when prompted to preview the report, click **Yes**.

2. Click  to adjust the currently displayed report's magnification level.

3. In the **Zoom** dialog box, do one of the following:

   - In the **Magnification** section, select a predefined magnification level: 200%, 100% (the default level), 65%, or 30%.

   - Type the required magnification level in the field next to the **Custom** option button.

4. Click **OK** to return to the **Print Preview** window at the selected magnification.

5. In the case of reports consisting of several pages or multiple reports, navigate using the following options:

Click…	…or on the View menu, click	…to do this:
	**First Page**	Browse to the first page of a multi-page report. You can also move through the pages of a report using the vertical scroller. Clicking the scroll box displays the current page number.
	**Previous Page**	Browse to the previous page of a multi-page report
	**Next Page**	Browse to the following page of a multi-page report
	**Last Page**	Browse to the last page of a multi-page report
	**First Report**	Browse to the first report (available with multiple report selection only).
	On the **View** menu, click **Previous Report**	Browse to the previous report (available with multiple report selection only).
	**Next Report**	Browse to the following report (available with multiple report selection only).
	**Last Report**	Browse to the last report (available with multiple report selection only).

Printing Documents

Depending on the options you selected, printing is available directly when you select a document for printing, or you can print a document from a print preview. This procedure refers to general printing. For details of batch printing to .pdf files, see Batch Printing Documents to PDF Files.

Note

- If your default printer is Acrobat PDFWriter, you must perform the following operation in the Registry Editor: in the registry path HKEY_CURRENT_USER\Software\Adobe\Acrobat PDFWriter, set the SZ Busy key value to NULL.

➢ To print documents

1. Select the desired items from which you can print reports, and on the appropriate main menu or, if available, on a shortcut menu, click the report that you want to print.
2. If a print preview prompt appears, do one of the following:
   - Click **Yes** to display a print preview of one or more reports.
   - Click **No** to print the reports directly to a printer or a file.
3. To print a report from a print preview to a printer or a file, do one of the following:
   - In the **Print Preview** window, click  to print the currently selected report.
   - In the **Print Preview** window, click  to print all the retrieved reports or documents in batch mode.

Note

- If you choose to directly print a report without previewing it, and the particular report usually includes printing parameters when viewed in the **Print Preview** window (for example, selection of a group separator), a dialog box opens to enable you to select those same parameters prior to printing.

Batch Printing Documents to PDF Files

SmartPlant Instrumentation supports batch printing of the following documents to PDF files:

- Enhanced reports (for details, see Generating Enhanced Reports).
- Entity specifications (for details, see Printing Specifications into a PDF File).
- Binder packages from the Document Binder module (for details, see Printing from a Binder Package to a PDF File).

Notes

- When printing .pdf files in batch mode for the above document types, you must install GNU Ghostscript or Adobe Acrobat Distiller. You then open the **Preferences** dialog box and under the **General** tab, select the appropriate application from the **PDF generator** list.

- In general, SmartPlant Instrumentation supports batch printing to .pdf files only for the above document types. It Is possible to print to .pdf files in batch mode for other types of documents; however, the software cannot create unique file names for each document and is liable to overwrite the previously generated file, resulting in generation of the last report only in the batch. In this case, to prevent this from occurring, you must configure your printer settings to prompt for a file name for each document (for details, see the user's guide for your specific printer).

Setting Ghostscript Batch Print Options

When you print reports in batch mode using GNU Ghostscript, the software does not create unique names for multiple output files (except for enhanced reports). For this reason, you need to configure the driver settings so that the software prompts you to enter a name for each report file.

➢ To set the Ghostscript batch print options

1. On the Windows taskbar, click **Start** > **Settings** > **Printers**.
2. Right-click **Generic PostScript Printer**, and on the shortcut menu, click **Properties**.
3. Click the **Ports** tab.
4. Click **Configure Port**.
5. In the dialog box that opens, from the **Output** list, select Prompt for filename.

Setting Acrobat Distiller Batch Print Options

When you print reports in batch mode using Acrobat Distiller, the software does not create unique names for multiple output files (except for enhanced reports). For this reason, you need to configure the print settings so that the software prompts you to enter a name for each report file.

➢ To set the Acrobat Distiller batch print options

1. Start Acrobat Distiller.
2. On the **File** menu, click **Printing Preferences**.
3. On the **Adobe PDF Settings** tab, select the check boxes **Prompt for the PDF filename** and **Ask to replace existing PDF file**.

Modifying Printer Settings

This option enables you to view and modify the current printer settings, if required. You can select a standard paper size with predefined width, height, and orientation or customize these settings as required. Furthermore, you can save your settings as default for future print sessions. You can customize your own paper width, height, and orientation and include these values in the default settings. Note, that some reports have their orientation hard-coded, therefore only the hard-coded settings will apply.

Notes

- All your page settings apply to all reports and documents that you print and they are true for all plant hierarchy levels (domain, <plant>, <area>, and <unit>).
- All your page settings apply to your local machine only and do not affect other users of SmartPlant Instrumentation.
- If you want to change the page setup for the current print session only, do not click **Default**, just make your changes and click **OK**.

When saving your settings as default, the software stores the values in the INTOOLS.INI file under the [PRINTER] section. If you want the software to calculate the required paper size from the Windows printer driver settings, open the INTOOLS.INI file and under the [PRINTER] section, remove the semi-colon before the following parameters:

- LEFTMARGIN
- RIGHTMARGIN
- TOPMARGIN
- WIDTH
- HEIGHT
- ORIENTATION
- HRES
- VRES

Add a semi-colon before the PAPERSIZE parameter.

For additional information about this option, click **Help** in the **Page Setup** dialog box.

Saving Documents

When you display a document, you can choose to save it as a file. This section describes the various methods available for saving documents.

➢ To save the current document to a file

1. With the **Print Preview** window open, click  .
2. In the **Save As** dialog box, select one of the following data formats:
   - **Original**: Saves the report data in all the fields, including the ones that are not visible in the preview (for example, internal database ID numbers). This option is recommended if you want to save the file in .psr or .xls format, or if you want to re-import the data into SmartPlant Instrumentation at a later stage.
   - **Data Only**: Saves only the report data that is visible in the preview. This option is recommended for easier viewing of the data, and also enables you to manipulate the headers and select the columns you want to display.
3. Click **OK** and navigate to the location where you want to save the file.
4. Select the required file format and type the name of the file, then click **Save**.

Saving in DXF Format

You can save any report in DXF format for insertion into a CAD drawing (with AutoCAD or MicroStation). This option is available when you select to save a document as a file from a **Print Preview** window. On selecting to save the file in DXF format, you can specify whether to include the title block and frame, whether to build a script file for AutoCAD, and define certain other parameters for AutoCAD.

➢ To save a document in DXF format

1. With the **Print Preview** window open, click  .

2. In the **Save As** dialog box, select **DXF File**.

3. In the **Save As DXF Options** dialog box, select **Include title block and frame with report** to save the report with the title block (which includes the logo and generic report data) and the frame. (Clear the check box to exclude the title block and frame from the saved report if the CAD package has its own frame.)

4. If you are working in batch mode to save a number of files that are to be opened in AutoCAD, you can automate the process as far as possible. Define the parameters in the **AutoCAD Options** and **User-defined options** sections of the dialog box as follows:

Field	Description
Insertion point	The X- and Y- coordinates for the insertion point of the report in the AutoCAD file define the displacement from the lower-left corner of the AutoCAD file. The upper-left corner of the report is inserted at this location.
Scale factor	The scale factor determines the size of the report so that it will fit into the AutoCAD drawing.
Font height coefficient / Font width coefficient	Font height and width coefficients for the text characters in the report. This option is needed because the software calculates the font size independently of the size of the report, whereas AutoCAD scales the font with the report as a whole.
Change black report color to	Type the value of the AutoCAD color to be used as a substitute for black. All reports are displayed in black-and-white. If viewing the report in AutoCAD with a black background, an alternative foreground color needs to be used. The recommended value is 7.

5. Browse to the target folder and click **OK**.

Editing Document Headers

When you select to save a document as a file from the **Print Preview** dialog box and you choose to save it as data only, you have the option of saving the data with or without headers. Furthermore, if you select to include the headers with the data, you can also edit them. This includes the possibility of exporting the headers to a separate text file.

➢ To edit the headers for a report

1. With the **Print Preview** window open, click  .
2. In the **Save As** dialog box, select **Data Only**.
3. Select the **Allow Headers** check box.
4. Select the **Edit Headers** check box and click **OK**.
5. Select the required file format and file path and click **Save**.
6. In the **Edit Column Headers** dialog box, select the **Incl.** check box for the columns you want to include in the output file.
7. Under the **Header Text** column, edit the text as required.
8. Under the **Width** column, change the maximum text width (number of characters allowed) as required.
9. If you want to save the header settings in this screen to a text file:
   a) Click **Export Headers**.
   b) In the **Export to File** dialog box, navigate to the required location and enter a file name.
   c) Click **Save**.

 **Note**

- If an external header text file exists, you can click **Import Headers** to load the data into the **Export to File** dialog box.

Customized Title Blocks for Reports

Overview

Title blocks for reports are templates that SmartPlant Instrumentation uses for the design of its reports. While the standard default title blocks supplied with the software are adequate for many reports, you can also create your own custom title blocks that provide you with the required design and layout of your reports.

You can, for example, customize report titles and headers, as well as the frame size of the title block.

You first create a title block or duplicate it from an existing one using InfoMaker and save it as a .psr file. You then open the .psr file in SmartPlant Instrumentation and add it to the title block inventory. After that, you need to associate the title block with the required reports and finally, generate a report through the relevant module using the created title block.

Tip

- It is usually more convenient to edit an existing title block and save it under a different name rather than to create a new one.

Requirements for Customized Report Title Blocks

You can customize a title block for reports in InfoMaker. In either application, the custom title block has to meet several standard requirements. The title block has to be compatible with the report it is associated with. Therefore, make sure that the customization process accurately fits the SmartPlant Instrumentation customizing conventions as specified in the following topic.

The following list outlines mandatory and optional InfoMaker requirements:

- **Report type** — When creating a new report set the report type to be **External** (mandatory).
- **Report style** — Set the report style to be **Tabular** (mandatory).
- **Field prompt** — When prompted to set the required fields for a title block, type any text in the appropriate field. There is no significance to inserting any particular field name as long as something is typed (mandatory).
- **Group** — The title block customization has to be performed in the **Detail** group (mandatory).
- **Layer** — Make sure that you create the title block in the **Band** layer (the default - mandatory).
- **Design** — The frame of the title block consists of four lines. Make sure that each line has its own unique name typed in small caps as follows:

Upper side	L_width
Lower side	ln_down
Left side	ln_left
Right side	ln_right

Caution

- When adding other design-related items, such as internal lines, text boxes, and so forth, to the title block, make sure that the names of the new items contain the tb segment. For example, if you want to add a new line, name the line as <Line>_tb_<1>. This is required to distinguish between items that belong to the title block and items that belong to the rest of the report outside the title block.

- **Field type** — select any field type other than **Column** (mandatory).
- **Revisions** — Make sure that you have a number 1 revision field. For instance, a title block with solely a revision no. 2, would not apply.

- **Report measurement units** — The units of measure of the customized title block must be compatible with the units of measure of the report it is associated with. All the reports were created in inches except for the following, which use PowerBuilder units:
  - All the calculation reports
  - The SmartLoop report
  - The Segment map report
- **Fields** — SmartPlant Instrumentation created an inventory of macros available for the custom title block fields. From this inventory you can select the fields that serve your purposes most. Each field and macro has its unique name (this is the name to insert in the name field when creating the item in InfoMaker) as shown in the tables below.

Notes

- Macros retrieve the data and enter it to the fields. For these macros to function properly, you must insert the macro name accurately according to the macro tables below.
- You can implement custom title blocks that for macros custom_1 through custom_5 display field values without labels, for example, 'DCS-1' instead of 'Panel DCS-1'. To enable this option, make sure that the [Custom] section of the INTOOLS.INI file includes the line '`RemoveFixedTextFromTB=1`'.

The following table includes **Text** type fields only:

Field Name	Field Description
custom_1	Report name
custom_2 – custom_5	Entity names or other data used in reports or drawings For example, in a panel-strip report, custom_2 is used to display the panel name, custom_3 is used to display the strip name, and custom_5 is used to display the domain name. In a Smartloop drawing, custom_2 is used to display the loop service, and custom_3 is used to display the loop number. All values that the software can retrieve for these macros are hard coded. This means that even if you use all of the custom fields, in the title block, fields that do not have any associated hard coded data appear empty.
eproj_name	Project name (Operating owner domain)
project_t	Project header (Operating owner domain)
proj_name	Domain name
proj_num	Project number (Operating owner domain)
city_name	City name
owner_name	<Plant> owner.

Field Name	Field Description
location_name	Location name
plant_hierarchy	Plant hierarchy defined by the Domain Administrator. The default plant hierarchy is Plant\Area\Unit. The Domain Administrator has rights to change the number of plant hierarchy levels and customize the level names.
hierarchy_item_name_x	The name of a specific item in the plant hierarchy, where the segment X defines the plant hierarchy level number. For example, suppose that your plant hierarchy is Plant\Area\Unit, and on the Unit level, you have an item name Crude. To display Crude in the title block, you need to substitude the segment X with the number 3.
plant_name	The name of the item on the highest plant hierarchy level (the default highest level is Plant).
plant_addr1	Address 1 of the highest plant hierarchy level item
plant_addr2	Address 2 of the highest plant hierarchy level item
plant_country	Country of the highest plant hierarchy level item
plant_state	State of the highest plant hierarchy level item
plant_zip	ZIP code of the highest plant hierarchy level item
area_name	The name of the item on the intermediate level of the plant hierarchy (the default intermediate level is Area). If the plant hierarchy has more than three levels, the macro area_name retrieves the name of the lowest intermediate level, that is one level above the lowest level of the plant hierarchy.
unit_name	The name of the item on the lowest level of the plant hierarchy (the default lowest level is Unit).
hierarchy_udf_y_x	A custom field value associated with a specific plant hierarchy item. The segment 'y' retrieves the number of the custom field (the Domain Administrator can define up to twenty custom field values for each plant hierarchy item). The segment 'x' retrieves the number of the level to which the plant hierarchy item belongs. Custom fields associated with a plant hierarchy item are regular text fields and behave as any other custom fields that are available in SmartPlant Instrumentation for specific entities.
hierarchy_udf_y_x_t	The header of a custom field associated with a specific plant hierarchy item. The segments 'y' and 'x' correspond to the segments in the macro hierarchy_udf_y_x. The segment 't' indicated the header.
tb_dwg_name_t	Document number
lastrev	Last revision
rev_no_1 – rev_no_9	Revision number 1 (mandatory field)
rev_udf_cXX_1 – rev_udf_cXX_9	Revision custom fields (for more details, see below)
create_by_1 – create_by_9	Created by
date_1 – date_9	Date
chk_by_1 – chk_by _9	Checked by

Field Name	Field Description
appr_by_1 – appr_by _9	Approved by
desc_1 – desc_9	Revision description

The following table includes **Computed** type fields only:

Field name	Field description	Expression
C_page_num	Current sheet	Page()
C_page_count	Total sheets	PageCount()
comp_1	Logo name	bitmap (ProfileString (intools.ini", Project", LogoPath", " + projlogo.bmp")

- **Revision custom fields** — you can define your own fields and related macros in addition to those in the inventory of macros (designated for fields) that are supplies with SmartPlant Instrumentation. You can add up to 20 revision custom fields per title block. Inserting the right name in the field name prompt is necessary for the proper function of the field in the title block. The naming convention for the revision custom fields is as specified in the following table:

rev_udf_c01_t	Header
rev_udf_c01_1	Data field
rev_udf_c01_2	Second data field under the same header
rev_udf_c02_t	Header 2 (another UDF)
rev_udf_c02_1	The data in the field (2)

- **Document custom fields** — Using these fields you can have a record of the documents that were related to a certain report. These are fields that you define under the following limitation: each custom field must have only two fields, one for the header and one for the data. This is the naming convention for these fields:

dwg_udf_c01_t	Header
dwg_udf_c01	Data field
dwg_udf_c02_t	Header 2 (another custom field)
dwg_udf_c02	Data field (2)

Opening a New Custom Title Block from SmartPlant Instrumentation

After you have finished creating a report title block and customized it to fit your needs, you need to open it from SmartPlant Instrumentation and set it up to function as a title block for a designated report. This includes loading it, editing as required and saving. The editing stage includes defining the headers for revision custom fields and defining the headers for the drawing records (if required).

Note

- In addition to the default title blocks that appear in SmartPlant Instrumentation, there are three .psr files that are supplied together with SmartPlant Instrumentation: DEFAULT TB WITH IN UNITS.PSR, DEFAULT TB WITH PB UNITS.PSR, and SPECS DEFAULT TB WITH IN UNITS.PSR. These .psr files contain a **Signed by** field for revisions. However, in the fields that display the names of the plant hierarchy levels and the specific level items, truncation may occur if the plant hierarchy level names are too long. In the Administration module, the Domain Administrator can change the default level names Plant, Area, and Unit to long strings (up to 50 characters are allowed). To prevent truncation, we recommend that you create your own custom title block and provide enough room in the plant_name, area_name, and unit_name fields.

➢ To open a custom title block

1. In the main SmartPlant Instrumentation window, on the **Tools** menu, click **Title Blocks**.
2. In the **Title Blocks** dialog box, click **New** to open the **Title Block Properties** dialog box.
3. Do one of the following:
   - Navigate to the .psr file that you have created containing the new custom title block, and select this file.
   - Navigate to the DEFAULT TB WITH IN UNITS.PSR file, or the DEFAULT TB WITH PB UNITS.PSR file.
4. Click **Open**.

5. In the **Title Block Properties** dialog box, do the following if required:

   a) Under **Revision custom field headers**, in the **Header** culumn, type the name for each custom field. For details on naming conventions for revision custom fields, see Revision Custom Fields.

   b) In the **Title block name** field, type the name that you have selected for the title block.

   c) Under **Document custom field headers**, in the **Header** culumn, type the name for each custom field. For details on naming conventions for document custom fields, see Document Custom Fields.

   Tip

   - You can preview the layout of the title block that you have created in the preview pane in the **Title Block Properties** dialog box. If you want to modify the number of revisions of custom title blocks, you need to add more revision rows and create appropriate macros in InfoMaker. The **Revision rows** box displays the number of revision rows that you have customized for your title block.

6. Click **OK** to save the edited title block and add it to the inventory of title blocks in SmartPlant Instrumentation.

7. In the **Title Blocks** dialog box, click **Close** to return to the main window.

Editing Report Title Blocks

You can edit report title blocks under two editing levels. The first is internal and includes changing the revision custom field headers, document headers and the title block name. The second level is external to SmartPlant Instrumentation and includes changes in the design, replacing macros, fields, adding and removing fields.

➢ To edit a report title block in SmartPlant Instrumentation

1. In the main SmartPlant Instrumentation window, on the **Tools** menu, click **Title Blocks**.

2. In the **Title Block** dialog box, select the desired title block and click **Properties**.

3. In the **Title Block Properties** dialog box, do the following if required:

   a) Under **Revision custom field headers**, in the **Header** column, type the name for each custom field. For details on naming conventions for revision custom fields, see Revision Custom Fields.

   b) In the **Title block name** field, type the name that you have selected for the title block.

   c) Under **Document custom field headers**, in the **Header** column, type the name for each custom field. For details on naming conventions for document custom fields, see Document Custom Fields.

   Tips

   - You can preview the layout of the title block that you have created in the preview pane in the **Title Block Properties** dialog box. If you want to modify the number of revisions of custom title blocks, you need to add more revision rows and create appropriate macros in InfoMaker. The **Revision rows** box displays the number of revision rows that you have customized for your title block.

   - For the second level of editing you can use an external editing application (for example, InfoMaker). When editing title blocks, you must follow the title block requirements. For details, see Requirements for Customized Report Title Blocks.

PSR File Viewer

PSR Files Overview

SmartPlant Instrumentation provides you with a Power Soft Report (.psr) file viewer that enables you to retrieve, view, and manage files saved in the .psr format. You can create a .psr file when you want to save a report or a specification using the **Save As** command. You can also view and edit .psr files in InfoMaker.

Viewing PSR Files

This feature enables you to retrieve files saved in the .psr format and view their contents. This way you can view specifications, reports, and forms which were saved as .psr files.

You can:

- Retrieve a .psr report that is saved in a file.
- Open a report from the **PSR List** window.

Retrieving PSR Files

If you saved a report as a .psr file, you can navigate to that file and open it.

➢ To retrieve a previously saved PSR file

1. Within any module, on the **File** menu, click **Import PSR**.
2. Do one of the following:
   - Click  .
   - On the **Actions** menu, click **PSR Viewer**.
3. In the **Select Report File** dialog box, navigate to the required .psr file and click **OK** to display the contents of the selected .psr file in the **PSR File Viewer** window.

Saving PSR File Data

You can save data (for example, a .psr file or data from the database) to an external file in any format supported by the PowerSoft format, for example: .psr, .wmf, .html, .sql.

For general details of how to save data from a **Print Preview** window, see Saving Documents.

Opening PSR Files from the PSR List

This option enables you to select an existing .psr file from the list and view its content in the **PSR File Viewer** window. The .psr file list is useful where you have a permanent set of standard .psr files that you want to view frequently.

➢ To view a .psr file using the PSR list

1. Within any module, on the **File** menu, click **Import PSR**.
2. Do one of the following:
   - Click  .
   - On the **Actions** menu, click **List**.
3. In the **PSR List** window, select the .psr file you want to view in the **PSR File Viewer** window.
4. Click  .

   Caution

   - If you want to include your user-defined field values in the report header, when there is more than one .psr file in the **PSR List** window, you need to open the required report in the InfoMaker, and under the WHERE section, type the following: psr_storage.psr_stor_desc='<description name>'. For details, see Viewing PSR User-defined Fields.

Managing the PSR File List

You can manage the list of .psr files displayed in the **PSR List** window.

➢ To manage .psr files

1. In the **PSR File Viewer** window, click  .
2. In the **PSR List** window, in the **Description** data field, type in a description to characterize the selected .psr file.
3. Perform one of the following actions as required:

Click…	…to do the following:
Insert	Append a new line where you can type in the path statement for an existing .psr file.
File	Retrieve a .psr file from a folder and add it to the list.
Delete	Remove a .psr file from the list.
Path	Set a new default path for all the files in the list. This option is useful if you have moved **all** the .psr files in the list to the **same** folder and you need to show the new location so that the files can be found. In this case, you do not need to change each file path separately. At least one .psr file must already be present in the required path.
Select	Display the data for the highlighted file in the **PSR File Viewer** window.

 **Caution**

- If you click **Path** and choose a new folder location for the .psr files, the original paths that were displayed for the files will be overwritten. You should therefore only use this option where you are certain that all the files in the list are located in the same folder.

4. On completion of the required action, click **Save**.

Retrieving PBL File Reports

The .psr format allows you to save a multi-page report and nested reports (reports which contain sub-reports or embedded objects). The .psr file format also enables you to design your own reports in InfoMaker and use names of columns from your database. You can then view the reports with the actual data by the **PBL Viewer** option.

Caution

- Prior to retrieving the reports from the appropriate .pbl file you need to define the .pbl file location. To define the PBL file location, in the INTOOLS.INI file [Custom] section, **LibraryList** parameter, you type the PBL file path and one of the PBL files in the path.

➢ To retrieve a report from a PBL file

1. With the **PSR File Viewer** window open, click  .
2. In the standard **Select Source File** dialog box, navigate to the required .pbl file and click **OK**.

   Caution

   - Make sure you select an appropriate PBL file, as the .pbl format is also used for purposes other than storing reports.

3. In the **DataWindow List** pop-up window, scroll to the required data window and select it.
4. Click  to retrieve the appropriate column data from the database and display it in the **PSR File Viewer** window.

Adding User-Defined Fields to PSR Files

When working with reports, you sometimes need to use extra fields in addition to those currently available to you. A number of user-defined fields (entitled **Udf C01, Udf C02**, and so forth.) are appended by default to every .psr file you import.

The first step in making the user-defined fields appear in a report is opening the report using a report generator, such as InfoMaker or MicroStation, and adding the appropriate user-defined fields to that report.

After saving a report as a .psr file, you can retrieve your user-defined field values and include them in the header of the required report.

➢ To add user-defined fields to a PSR file (using InfoMaker)

1. Start InfoMaker.
2. Build a query for generating a report (for details, see Building a Query).
3. With the main report window open, on the **Objects** menu, click **Computed Field**.
4. In the **Computed Object** dialog box, make the required definitions for the user-defined field (for example, alignment, positioning, color, style).
5. Under the **Expression** section, type the following:

   profstring(<SmartPlant Instrumentation home folder>\INTOOLS.INI", External Report", <custom field name>","")
6. Repeat step 5 to add as many user-defined fields as required
7. Save the current report as a .psr file.
8. With any main module window open, from the **File** menu, click **Import PSR**.
9. In the **PSR File Viewer** window, click  to open the **PSR List** window.
10. Scroll to the right side of the window to view the user-defined columns and in the columns **Udf C0** to **Udf C10**, type the required data (for example, revisions).

    Caution

    - When there is more than one .psr file in the **PSR List** window, you need to open the required report in InfoMaker, and under the WHERE section, type the following: psr_storage.psr_stor_desc='<description name>. This is required when including your user-defined field values in the report header. For details, see Viewing PSR User-Defined Fields.

11. Click **Save** to save your user-defined data to the database.

Viewing PSR User-Defined Fields

This option enables you to retrieve your user-defined field values from the existing .psr files, and include these values in the header of the required report within the SmartPlant Instrumentation environment.

➢ To view PSR user-defined fields in a report header

1. Open the required report in the InfoMaker report generator.
2. Under the WHERE section, type the following statement:

   ```
   psr_storage.psr_stor_desc='<PSR description>'.
   ```

   Caution

   - Without adding this statement to InfoMaker, SmartPlant Instrumentation always retrieves from the database the first user-defined field row in the **PSR List** window, regardless of the report you select. Carrying out steps 1 and 2 is required to be able to view the user-defined fields of the selected .psr file when there is more than one .psr file in the **PSR List** window. Make sure that the '<PSR DESCRIPTION>' value is the same as in the **Description** column of the **PSR List** window in SmartPlant Instrumentation.

3. Save the changes to the report and close InfoMaker.
4. Access SmartPlant Instrumentation and do one of the following:
   - Retrieve the required .psr file, and on the **PSR File View** window toolbar click  .
   - Open the required .psr file from the **PSR File List** window.
5. In the data window of the **PSR List** window, select a .psr file with user-defined fields.
6. Do one of the following to open the **PSR File Viewer** window:
   - Double-click the selected .psr file.
   - On the toolbar, click  .
   - On the **Actions** menu, click **Select**.
7. On the **PSR File Viewer** window toolbar, click  .

Entity Claiming Options

Overview

Claiming entities is a procedure can be performed in an Operating owner domain. This procedure involves assigning entities to a project using the AsBuilt data as a source. You can claim the same entity for more than one project.

When there are other entities that are associated with the entity used as a source for claiming, you can perform the procedure with or without assigning the associated entities. For example, you can claim a cable with or without the associated cable sets or wires.

Claiming entities is available in the Administration module and also in some of SmartPlant Instrumentation modules, provided that you open a project with AsBuilt entities displayed.

Claiming options are available in the following modules:

- Administration module — The Project Administrator can claim entities in batch mode when defining the scope of entities for a project. Claiming entities is the final stage of defining the scope of entities for the project. Before claiming the entities, the Project Administrator copies the entities to the **Claim Buffer** (only available in the Administration module). Then, these entities can be claimed for the project in batch mode.
- Instrument Index module — Users can select tag numbers and claim them the required engineering data defined in other modules, for example, with the associated hook-up data.
- Browser module — Users can select tag, line, and loop numbers, and equipment data for claiming.
- Explorer windows — Users can select any entity and claim it according to the preferences set on the **Claim Options** page.

Claiming Tag Numbers

You can claim tag numbers in the Instrument Index module, or in the Browser module. In an Operating owner domain, use this procedure to claim the AsBuilt tags for the current project, with or without the engineering data associated with the tags.

Note

- This option is available only when working with data displayed in both the project and AsBuilt.

➢ To claim tag numbers

1. Open a project together with the AsBuilt data.
2. Do one of the following:
   - Open an Instrument Index Standard Browser view and do the following.
   - In the Browser module, open a browser view for the Instrument Index browser.
3. Select and right-click the AsBuilt tag numbers that you want to claim for the current project.

   Tip

   - AsBuilt and project tag numbers have different color coding. The Project Administrator selects which colors to use in the **Project Activities** dialog box in the Administration module.

4. On the shortcut menu, click **Claim**.
5. In the **Tag Claiming Options** dialog box, do the following:
   a) Select **Basic engineering data** to claim associated data defined for the tags in the Instrument Index, Calculation, Specifications, Hook-Ups, Process Data, and Dimensional Data for Piping modules.
   b) Select **Wiring data** to claim associated wiring data defined for the tags in the Wiring module.

   Tip

   - Clear each check box to claim the tag numbers without the specified associated engineering data.

Claiming Loops, Lines, and Equipment

The following procedure describes how to claim AsBuilt loops, lines, and equipment in the Browser module. The software claims entities without any engineering data that is associated with the entities. For example, when claiming a loop, the software does not claim the tag numbers associated with the loop.

Notes

- You can only claim entities in an Operating owner domain, when working with data displayed in both the project and AsBuilt.
- In the Browser module, you can also claim tag numbers. For more information, see Claiming Tag Numbers.

➢ To claim loops, lines, or equipment

1. Open a project together with the AsBuilt data.
2. In the Browser module, expand the Instrument Index browser group and open a browser view for one of the following browsers:
   - Loop Browser
   - Line Browser
   - Equipment Browser
3. Select and right-click the AsBuilt entities that you want to claim for the current project.

   Tip

   - AsBuilt and project entities have different color coding. The Project Administrator selects which colors to use in the **Project Activities** dialog box in the Administration module.

4. On the shortcut menu, click **Claim**.