SmartPlant Instrumentation

Tutorial

Process, Power & Marine

INTERGRAPH

Version 7.0, Service Pack 2 July 2005 DINS-07.00.0003B

Copyright

Copyright ©1995-2005 Intergraph Corporation. All Rights Reserved.

Including software, file formats, and audiovisual displays; may be used pursuant to applicable software license agreement; contains confidential and proprietary information of Intergraph and/or third parties which is protected by copyright law, trade secret law, and international treaty, and may not be provided or otherwise made available without proper authorization.

Restricted Rights Legend

Use, duplication, or disclosure by the Government is subject to restrictions as set forth in subparagraph (c) of the *Contractor Rights in Technical Data* clause at DFARS 252.227-7013, subparagraph (b) of the *Rights in Computer Software or Computer Software Documentation* clause at DFARS 252.227-7014, subparagraphs (b)(1) and (2) of the *License* clause at DFARS 252.227-7015, or subparagraphs (c) (1) and (2) of *Commercial Computer Software---Restricted Rights* at 48 CFR 52.227-19, as applicable.

Unpublished---rights reserved under the copyright laws of the United States.

Intergraph Corporation
Huntsville, Alabama 35894-0001

Warranties and Liabilities

All warranties given by Intergraph Corporation about equipment or software are set forth in your purchase contract, and nothing stated in, or implied by, this document or its contents shall be considered or deemed a modification or amendment of such warranties. Intergraph believes the information in this publication is accurate as of its publication date.

The information and the software discussed in this document are subject to change without notice and are subject to applicable technical product descriptions. Intergraph Corporation is not responsible for any error that may appear in this document.

The software discussed in this document is furnished under a license and may be used or copied only in accordance with the terms of this license.

No responsibility is assumed by Intergraph for the use or reliability of software on equipment that is not supplied by Intergraph or its affiliated companies. THE USER OF THE SOFTWARE IS EXPECTED TO MAKE THE FINAL EVALUATION AS TO THE USEFULNESS OF THE SOFTWARE IN HIS OWN ENVIRONMENT.

Trademarks

Intergraph, the Intergraph logo, SmartPlant and INtools are registered trademarks of Intergraph Corporation. Microsoft and Windows are registered trademarks of Microsoft Corporation. MicroStation is a registered trademark of Bentley Systems, Inc. Other brands and product names are trademarks of their respective owners.

Table of Contents

Introduction **7**

Administration Options 7
Instrument Engineering Options 8
General Instrument Engineering Tasks 8
Creating Instruments and Control Loops 8
Defining Process Data, Sizing, and Specifications 8
Managing Documents 9
Performing Wiring 9
Generating Loop Drawings 9
Generating Hook-Up Drawings 10
Viewing and Modifying Data for Multiple Records 10

Administration Options **11**

System Administration 11
Entering as System Administrator 11
Task 1 — Initializing a Domain and Associating it with the SmartPlant Instrumentation Database 12
Task 2 — Defining SmartPlant Instrumentation Users and Domain Administrator 14
Domain Administration 18
Entering as Domain Administrator 18
Task 1 — Defining Domain Administrator Access Rights 19
Task 2 — Defining a Plant Owner 20
Task 3 — Creating the Plant Hierarchy 21
Task 4 — Users, Groups, and Access Rights 26

General Instrument Engineering Tasks **33**

Opening SmartPlant Instrumentation 33

Creating Instruments and Control Loops **37**

Task 1 — Defining Instrument Index Supporting Tables and Properties 37
Task 2 — Defining Loops with Their Tags 56
Task 3 — Duplicating a Loop with Its Tag Numbers 64
Task 4 — Creating Plant Loops in Batch Mode 68

Defining Process Data, Sizing, and Specifications **75**

Task 1 — Defining Process Data 75
Task 2 — Performing Calculations 81
Task 3 — Viewing and Editing Specification Sheets 86
Task 4 — Create a Specification Form Data Template 90
Task 5 — Creating and Using a Multi-Tag (See-List) Spec 94
Task 6 — Creating a Customized Specification Sheet 102
Task 7 — Specification Sheet Revisions 106

Managing Documents **111**

Task 1 — Creating a Specification Binder Package 111
Task 2 — Creating Form Notes and General Notes 114
Task 3 — Editing and Revising Specification Sheets from the Binder Package 118
Task 4 — Printing Documents for a Specific Revision 121
Task 5 — Creating a General Document Binder Package 123

Performing Wiring Operations **127**

Task 1 — Overview 127
Task 2 — Creating Reference Panels 130
Task 3 — Creating Terminal Strips with a Marshaling Rack 134
Task 4 — Creating a Reference DCS Panel 137
Task 5 — Copying the Reference Panels to the Domain Manager 145
Task 6 — Creating Reference Cables 147
Task 7 — Copying Reference Cables to the Plant 152
Task 8 — Making Connections 154
Task 9 — DCS Management 163
Task 10 — Cross-Wiring the Signals in the Marshaling Rack 171
Task 11 — Adding a New Instrument to the Existing Wiring 176
Task 12 — Create a New Tag with no Associated Profile and then Add a Device Panel and a Cable to the Tag 177
Task 13 — Wiring Design that Includes a Barrier 181
Task 14 — Wiring Design for Other Loop Types 187

Generating Loop Drawings **203**

Generating Enhanced SmartLoop Reports 203
Task 1 — Setting Preferences 203
Task 2 — Generating an Enhanced SmartLoop Report 205
Task 3 — Modifying an Enhanced SmartLoop Drawing by Changing Data Directly from the Enhanced Report Utility 207
Task 4 — Modifying the Appearance of a SmartLoop Drawing by Changing the Enhanced Report Layout Properties 211
Task 5 — Adding Annotations to an Enhanced SmartLoop Report 223
Generating CAD Loop Drawings 231
Task 1 — Setting Preferences 231
Task 2 — Defining Block Types and Drawing Blocks 233
Task 3 — Assigning Blocks to Instrument Types 235
Task 4 — Defining the General Blocks 237
Task 5 — Editing the Title Block for a Specific Loop 239
Task 6 — Understanding Macros 241
Task 7 — String Manipulation for Loop Drawings 242
Task 8 — Generating CAD Loop Drawings 244

Generating Hook-Up Drawings **251**

Task 1 — Setting Preferences 251
Task 2 — Creating Hook-Up Items 252
Task 3 — Creating Hook-Ups and Attaching Them to Tag Numbers 255
Task 4 — Assigning Items to the Hook-Up Drawing 259

Task 5 — Generating a Bill of Material (BOM) 260

Viewing and Modifying Data for Multiple Records 261

Task 1 — Viewing and Editing Tag Number Data from a Browser View 262
Task 2 — Finding and Sorting Data in a Browser View Window 265
Task 3 — Using More Features of the Standard Instrument Index Browser 268
Task 4 — Creating 'Junction Box Assignment per Tag' Browser 270
Task 5 — Creating a New Customized Process Data Browser 276
Task 6 — Customizing the Printout Structure 278
Task 7 — Using the Sum and Avg. Features in Printed Reports 281

Demo P&ID 285

Wiring Block Diagram 286

Introduction

The aim of this tutorial is to familiarize you with the basic features of SmartPlant Instrumentation® (powered by INtools®) – getting the program up and running, creating a minimal setup for your plant, and all the basic instrument-engineering activities for each module. This tutorial will provide you with the fundamental understanding, skills, and practical experience you need to begin using SmartPlant Instrumentation with confidence.

In the course of the tutorial, you will be guided step-by-step to create a loop with its associated wiring, line, and instrument process data. You will also perform calculations, generate specification sheets, loop drawings, and installation details (hook-ups).

We recommend that you do the following:

- Make a backup copy of the database before using it with the tutorial.
- Work through all the objectives in a given task in one sitting.

The tutorial is divided into two parts: one for the System and Domain Administrators and the other for Instrument Engineers. If you are a novice, we strongly recommend going through the entire tutorial thoroughly.

Administration Options

The System Administration tasks that you will learn include the following:

- How to initialize and assign a domain.
- How to create a department.
- How to add users to the SmartPlant Instrumentation user list.
- How to create and assign a Domain Administrator for the domain.

The Domain Administration tasks that you will learn include the following:

- How to define a plant owner.
- How to set up the plant hierarchy.
- How to create hierarchy items and define tag and loop naming conventions.
- How to define a group and assign users to the group.
- How to define access rights for a group.
- How to add user-defined fields to the Instrument Index.

Instrument Engineering Options

If starting the tutorial here, you will go to a unit where the administration items have already been created.

General Instrument Engineering Tasks

The general tasks that you will perform in this section include:

- How to open a domain in SmartPlant Instrumentation.
- How to navigate in SmartPlant Instrumentation using the Explorer windows.

Creating Instruments and Control Loops

The tasks in this section include:

- How to define data in supporting tables, including instrument type profiles, P&ID drawing numbers, and lines.
- How to create loops and associated tags.
- How to generate and print out reports.

Defining Process Data, Sizing, and Specifications

In this section, you will learn how to define process data values, how to use those values to perform sizing calculations that generate additional data values, and how to generate specification sheets that include both process data and calculation results. The tasks in this section include:

- How to create lines.
- How to define line process data.
- How to define instrument process data (flow elements, control valves, and so forth.)
- How to perform instrument sizing calculations.
- How to create single- and multi-tag (see-list) specification sheets that include process data and calculation data.
- How to perform revisions on specification sheets.
- How to print a specification sheet.

Managing Documents

In this section, you will use the Document Binder module to link together a group of related specification sheets and associated notes into a specification binder package, which can be used for bids and in the procurement process. You will also perform revisions for the entire package. The tasks in this section include:

- How to create a Specification Binder Package.
- How to add Form Notes and General Notes.
- How to edit and revise specification sheets from a Specification Binder Package.
- How to generate and print out reports for the Specification Binder Package.

Performing Wiring

In this section, you will create the essential wiring entities and make the connections needed to specify a complete control loop for the tag numbers that you created. You will also learn how the software can create a signal from the field instruments to the control panel. The tasks in this section include:

- How to create panel-strip-terminal reference entities and copy them to your plant.
- How to create cable-set-wire reference entities and copy them to your plant.
- How to assign tags to DCS channels.
- How to connect cables / wires as appropriate.
- How to design the wiring for specific control loops, including connection type definitions and cross-wiring connections.

Generating Loop Drawings

Using the Enhanced Report Utility, you will learn how to display loop drawings for the control loops you have created. You will also learn how to create and use drawing blocks to generate loop drawings in a CAD application. The tasks in this section include:

- How to view and modify a loop drawing.
- How to change layout settings and include annotations for enhanced reports.
- How to create block types and blocks for CAD drawings.

- How to create block-tag associations.
- How to add revisions.
- How to browse macros.
- How to generate a CAD loop drawing.

Generating Hook-Up Drawings

Using the Hook-ups module, you will learn how to generate installation detail drawings for the tags that you have created, as well as Bill of Material reports. The tasks in this section include:

- How to create hook-up types and hook-ups.
- How to assign tag numbers to hook-ups.
- How to generate a Bill of Material report.

Viewing and Modifying Data for Multiple Records

The tasks in this section include:

- How to view and edit data using the Instrument Index Standard Browser.
- How to manipulate SmartPlant Instrumentation data from your browser view.

In this tutorial, you will use Demo P&ID 101-PID01-001 which is attached to the tutorial.

There is also an interconnection wiring block diagram attached to this tutorial to help you understand the wiring part of this tutorial.

Note

- You should make a backup copy of the database if you or other users intend to repeat this tutorial.

Throughout the tutorial, you will open new screens and use new options. While some of these are explained here in detail, we encourage you to browse through the *SmartPlant Instrumentation User's Guide* and *SmartPlant Instrumentation Online Help* to gain knowledge and help.

Administration Options

If you have already performed the system setup and configuration or if you are interested only in the instrument engineering options, you can skip directly to the Instrument Engineering options. Otherwise, proceed with the System Administration options.

System Administration

The System Administration procedures are usually performed at the beginning, right after the installation of SmartPlant Instrumentation. The purpose of this chapter is to acquaint you with the procedure of assigning a System Administrator who in turn will configure the basic system organization and hierarchy.

Entering as System Administrator

1. Start the Administration module as follows:

   a) On the Windows **Start** menu, point to **Programs**, point to **Intergraph SmartPlant Instrumentation**, and then click **Administration**.

   b) On the **Logon Information** dialog box, enter **DBA** as your user name and password.

Tip

- When you enter the system for the first time, the default user name and password are both **DBA**. The user name always appears in upper case characters, regardless of the keyboard settings and the password asterisks display.

   c) Click **OK** to start the Administration module.

2. On the **Open Administration Module** dialog box, click **System Administrator** and then click **OK**.

Task 1 — Initializing a Domain and Associating it with the SmartPlant Instrumentation Database

Starting a new domain involves two major procedures — initialization and association of the domain.

Initialize Your Domain

The very first thing that you do is to create your domain. This procedure is called **Domain initialization**.

1. On the **File** menu, click **Initialize**.
2. In the **Initialize** window, enter data as shown.

Tips

- Ensure that in the **Target domain type** group box, you click the **Engineering company** option.
- You may enter your own domain and schema names if you wish. Note that if you are initializing an Oracle or SQL Server domain, this window looks a little different.
- Type **DEMO_1** in the **Domain schema password** box. The software displays asterisks for this value and for the value of **View Only Domain Schema Password**.

3. Click **OK**.

   The software initializes the domain, indicating the state of progress on the screen. The process may take about ten minutes, depending on your computer. When completed, a message appears informing you that the domain initialization has been completed successfully.
4. On completion of the initialization, click **Close**.

Now you have to associate your new DEMO_NEW domain with the SmartPlant Instrumentation database.

Associate your New Domain with the SmartPlant Instrumentation Database

After initializing the domain, you have to associate it with the SmartPlant Instrumentation database.

1. On the main toolbar, click to open the **Domain Definition** window.
2. From the **Domain** list, select **DEMO_NEW** and on the **Domain Definition** toolbar, click .

The data in the window should appear as shown.

3. On the **Domain Definition** toolbar, click and then click .

Note

- System Administration and Domain Administration can be accessed by one user only at a time!

  You have successfully completed your domain initialization and association.

After completing the domain initialization and association, before SmartPlant Instrumentation users can start working, there are a number of activities that you need to carry out as the System Administrator. These activities are described in the following sections.

Task 2 — Defining SmartPlant Instrumentation Users and Domain Administrator

Before creating the user list, it is recommended that you create at least one department to which the users will belong.

Note

- If you do not have the Administration module running in the System Administration mode, first open the module. For details, see *Entering as System Administrator*, page 11.

Create a New Department

1. On the main toolbar, click to open the **Domain Definition** window.
2. From the **Domain** list, select **DEMO_NEW**.
3. On the main toolbar, click to open the **Department** dialog box.

4. Click **New**.
5. In the **Department** box, type **Instrumentation**.
6. Enter description and note as you require.
7. Click **Apply**, and then click **Close**.

You can now assign users to your department.

Add New Users and Assign Them to a Department

1. Make sure that the **Domain Definition** window is open and that **DEMO_NEW** is selected in the **Domain** field.
2. Click to open the **User** dialog box.

3. Click **New**.
4. In the **User** field, type **USER1** (it is displayed in upper case characters automatically).
5. In the **User initials** field, type **U1** (using upper case characters). These initials will appear in various documents.
6. From the **Department** list, select **Instrumentation** to assign the new user to this department.
7. In the **Password** field, type **USER1** as the user password (using upper case characters).

   Tips

   - The password is displayed as asterisks.
8. In the **Verify new password** field, type **USER1** again.
9. In the **Note** text field, type a note if required.

   Tips

   - The System administrator check box is used to grant System Administrator rights to a new user. Usually, this check box is cleared.
   - It is advisable that you keep a note of the password for each user.
10. Click **Apply**.
11. Now add the more users to the Instrumentation department so that the completed user information appears as shown.

User	Initials	Password
USER1	U1	USER1
DOMAIN	DA	DA
USER2	U2	USER2

12. Click **Close** to close the **User** dialog box.

Assign a User as a Domain Administrator

After defining all the users and assigning them to groups, you have to select a user to be assigned as the Domain Administrator who will handle all the internal configuration of the Domain.

Note

- If you do not have the Administration module running in the System Administration mode, first open the module. For details, see *Entering as System Administrator*, page 11.

1. Click to open the **Domain Definition** window.
2. From the **Domain** list, select **DEMO_NEW**.
3. On the **Domain Definition** toolbar, click .
4. From the **Administrator** list, select user **DOMAIN** as Domain Administrator.

   Tip

   - As System Administrator, you also may edit the domain name, number and description, as well as activating the audit trail mechanism.

5. Under **Domain features**, clear the **Activity tracking** and **Audit trail options** check boxes.
6. On the toolbar, click and then click .

Generate System Administration Reports

As System Administrator, you can generate a number of reports concerning System and Domain Administration.

1. Log in as the System Administrator.
2. On the **Reports** menu, generate each of the available reports.
3. When done, on the **File** menu, click **Exit** to close the Administration module.

Note

- This concludes the System Administration activities required for your domain configuration. The Domain Administrator is responsible for performing subsequent administration tasks.

Domain Administration

The System Administrator must finish configuring the system and assigning a Domain Administrator before you can access and perform Domain Administrator activities.

Entering as Domain Administrator

1. On the Windows **Start** menu, point to **Programs**, point to **Intergraph SmartPlant Instrumentation**, and then click **Administration**.
2. On the **Logon Information** dialog box, type **DOMAIN** as your user name and **DA** as your password.
3. Click **OK** to open the **Open Administration Module** dialog box.

   Tip

   - Because you logged on as the Domain Administrator, only the **Domain Administrator** option is available in this dialog box.
4. In the data window, select **DEMO_NEW**.

5. Click **OK** to open the **Domain Administration** window.

You have now successfully logged on as the Domain Administrator and you are ready to carry out the Domain Administration activities. These include defining a plant or site owner and setting up a plant hierarchy (the software creates a default plant hierarchy when you define a new domain).

Task 1 — Defining Domain Administrator Access Rights

Before you can perform certain activities, you must ensure that you have appropriate access rights as Domain Administrator.

Define Domain Administrator Access Rights

1. Click to open the **Domain Definition** window.
2. Do one of the following:
   - Click .
   - On the **Options** menu, click **Access Rights**.
3. In the **Access Rights** window, click .
4. On the **Global Access Rights** dialog box, from the **Access mode** list, select **Full (Add / Delete /Update)**.
5. From the **Group name** list, select **All**.
6. Select all the check boxes and in each of the remaining lists, select **All**.

7. Click **OK** to save your selections and close the dialog box.

8. On the toolbar, click and then click .

This procedure ensures that you have granted yourself maximum access rights for all operations in your domain. Later, you will create other users and limit their access to certain options. However, before doing so, you are going to set up your plant hierarchy.

Task 2 — Defining a Plant Owner

For these activities, you will use the domain created by the System Administrator.

Define an Owner

1. With the **Domain Administration** window open, do one of the following:
   - Click .
   - On the **Activities** menu, click **Owner** to open the **Owner** dialog box.
2. Click **New** to define a new owner.
3. Type the owner name as shown.

4. Complete the rest of the owner data as you require.
5. Click **Apply** and then click **Close**.
6. On the toolbar, click .

Task 3 — Creating the Plant Hierarchy

When a user starts a module to access information, such as loops or tag numbers in the Instrument Index, the information is grouped on a per process unit basis. The Domain Administrator is responsible for setting up and organizing the plant hierarchy.

Create a Plant Hierarchy

1. With the **Domain Administration** window open, on the **Activities** menu, click **Plant Hierarchy Explorer**.

   Tip

   - The software allows you to define a flexible hierarchy with any number of levels, for which you can specify the name of each level as you desire. For the purpose of this Tutorial, you will be creating a plant hierarchy consisting of three levels — PLANT, AREA, and UNIT.

2. Right-click **Plant Hierarchy Explorer**.
3. On the shortcut menu, click **New** > **Plant**.
4. On the **Plant Properties** dialog box, in the **Plant** box, name your plant **New Refinery**.
5. From the **Owner** list, select **Global Chemical Supplies**.
6. Ensure that the **Do not propagate wire tag names** check box is cleared.

The **Plant** dialog box should appear as shown.

7. Type the rest of the plant data in the appropriate fields.
8. When done, click **OK**.
9. In the **Plant Hierarchy Explorer**, right-click **New Refinery**.
10. On the shortcut menu, click **New** > **Area**.
11. On the **Area Properties** dialog box, in the **Name** box, type **Crude Area** and click **OK** to close the dialog box.
12. In the **Plant Hierarchy Explorer**, click ⊞ beside **New Refinery** to expand the hierarchy, and then right-click **Crude Area**.
13. On the shortcut menu, click **New** > **Unit**.
14. On the **Unit Properties** dialog box, in the **Name** box, type **Crude Unit 1**.
15. In the **Number** box, type **101**.
16. Click **OK** to close the dialog box.
17. Click ⊞ beside **Crude Area** to expand the hierarchy.

The **Plant Hierarchy Explorer** should now appear as shown.

18. Click ☒ to close the **Plant Hierarchy Explorer**.

You have just completed organizing the process plant hierarchy. Next, you will define tag and loop number naming conventions for the new unit.

Define Tag and Loop Naming Conventions for Your New Unit

1. With the **Domain Administration** window open, do one of the following:
   - Click $x.
   - On the **Activities** menu, click **Naming Conventions** to open the **Naming Conventions** dialog box.
2. Beside **Plant hierarchy**, click **Browse** and navigate to the unit you created.

   The **Plant hierarchy** field should display the hierarchy **New Refinery/Crude Area/Crude Unit 1**.
3. In the **Convention** list, select **Component** and note that the text box label indicates the ISA naming convention standard that the System Administrator defined for the domain.

   Tip
   - According to the ISA standard, the Component tag string can contain up to four (4) different segments. Each segment can be separated by a character in the **Separator** column in the data window. Each segment is composed of different data fields from the Instrument Index module data input with a starting character position and a length.
4. Enter separators for the second, third and fourth tag segments and type or modify the string lengths in the **Length** column as shown.

The **Sample** field shows the changes in the tag structure as you make your naming convention definitions in this dialog box.

Tips

- The **Description** field for the **PREFIX** segment is **UNIT NUMBER** by default. This prefixes all new tags in this unit with the unit number you entered on the **Unit** dialog box.
- You can only change the tag convention for a unit so long as no tags are defined for that unit.

5. Click **Apply**.
6. At the prompt, click **OK**.
7. Define the loop naming conventions as follows:
   a) In the **Convention** list, select **LOOP** and note how the number of segments in the data window changes.
   b) Type data as shown.

8. Click **Apply**.
9. At the prompt, click **OK**, and then click **Close**.

Create Custom Fields

The Domain Administrator can add custom fields in addition to the fields appearing in the default view of the Instrument Index Standard Browser.

1. With the **Domain Administration** window open, do one of the following:
   - Click .
   - On the **Activities** menu, click **Custom Fields** to open the **Custom Fields** dialog box.
2. From the **Plant** list, select **New Refinery**.
3. From the **Entity/data type** list, select **Instrument**.
4. Type definitions for fields 1 through 7 as shown.

5. Click **Apply**.
6. At the prompt, click **OK**, and then click **Close**.

The fields you selected have been added to the Instrument Index database and the software will now recognize them in the Instrument Index. When requested to do so, the software will retrieve the data contained in them for loop drawings, specification sheets, reports, and so forth.

Task 4 — Users, Groups, and Access Rights

Create a domain group

This option enables the Domain Administrator to create a group of users.

1. With the **Domain Administration** window open, on the **Activities** menu, click **Group**.
2. On the **Group** dialog box, click **New**.
3. In the **SmartPlant Instrumentation group** box, type **Instrumentation**.
4. Type a group description and note if you require.

5. Click **Apply**, and when prompted whether to copy access rights from another group, click **No**.
6. Click **Close**.

Assign a User to the New Group

After creating the new Group, the Domain Administrator assigns users to that group.

Tip

- A group can contain several users or one user only. When a group contains one user only, you can exercise tight control or to grant special access rights to one person.

1. With the **Domain Administration** window open, on the **Activities** menu, click **Assign Users to Groups** to open a dialog box where you can view the entire list of the existing SmartPlant Instrumentation users.

2. From the **Group** list, select **Instrumentation**.
3. Select and drag the following users from the **User list** pane into the **Group users** pane — **USER1**, **USER2**, and your own name **DOMAIN**. If you want to remove a user from a group, drag that user from the **Group users** pane to the **User list** pane.

   Tips

   - You can assign the same user to more than one group.
   - You can select a user and click **User** to open the **User** read-only dialog box to display a complete user description for the selected user.

   The **Assign Users to Group** dialog box should now appear as shown.

4. When done, click **Apply**.
5. At the prompt, click **OK**, and then click **Close**.

Grant Access Rights for a Group

Once groups and users have been defined, the Domain Administrator grants them access rights to various activities. Note that access rights are granted **per group**. This means that all users in the selected group are granted the same access rights.

1. In the **Domain Administration** window, click to open the **Domain Definition** window.
2. On the **Domain Definition** toolbar, click to open the **Access Rights** window with the user group displayed in the **Group list** section of the window.

3. Double-click **Instrumentation** to display the access levels for that group.
4. Expand **Unit Level** by clicking the ⊞ icon to display all the units in the domain.
5. Select the unit — **New Refinery/Crude Area/Crude Unit 1**.

   The **Entity or activity** section of the window displays the list of entities and activities that apply at the unit level.

6. From the **Entity or activity** column, scroll down the **Name** list to **Hook-Ups Module Access**.
7. From the **Mode** list, select the desired access rights for the particular entity or activity, as shown in the following example.

8. On the toolbar, click .
9. Test your access rights as follows:
   a) Log on to SmartPlant Instrumentation as USER1.
   b) In the **Open** window, select the unit — **New Refinery/Crude Area/Crude Unit 1**.
   c) Click the **Modules** menu and note that all modules are available except for the Hook-Ups module.
   d) Exit SmartPlant Instrumentation.
10. In the Administration module, reopen the **Access Rights** window and reset **Hook-Ups Module Access** to **Full (Add / Delete /Update)**.
11. On the toolbar, click to close the **Access Rights** window; at the prompt to save your changes, click **Yes**.

Apply Access Rights to More than One Group or Entity

You can grant the same access rights for all the user groups in the domain or for all the entities at each level (for a particular group or for all groups). You have already used this option to grant full access to the Domain Administrator for all entities. You will now modify access rights globally for the **Instrumentation** group and perform other exercises to learn how this feature works.

1. In the **Domain Administration** window, click to open the **Domain Definition** window.
2. On the **Domain Definition** toolbar, click to open the **Access Rights** window.
3. Click to open the **Global Access Rights** dialog box.
4. From the **Access mode** list, select **Modify (Add / Update)**.
5. From the **Group name** list, select **Instrumentation**.
6. Define access rights at the plant level by doing the following:
   a) Under **Access rights on the level <Plant>**, select the **Enable entity selection** check box.
   b) In the **Plant** list, select **New Refinery**.
   c) In the **Entity** list, select **All**.
7. Click **OK**.

8. On the **Access Rights** dialog box, check the results of applying these access rights by doing the following:
   a) Double-click the **Instrumentation** group to display the access levels.
   b) Expand **Plant Level** by clicking the ⊞ icon to display all the plants in the domain.
   c) Click **New Refinery**.

   In the **Mode** column of the right pane, all the entities for the New Refinery plant should be defined as **Modify (Add / Update)**.
9. Click to reopen the **Global Access Rights** dialog box.
10. From the **Access mode** list, select **Access Denied**.
11. From the **Group name** list, select **All**.
12. Define access rights at the unit level by doing the following:
    a) Under **Access rights on the level <Unit>**, select the **Enable entity selection** check box.
    b) From the **Unit** list, select **New Refinery/Crude Area/Crude Unit 1**.
    c) From the **Entity** list, select **Process Data Change in Specs**.
13. Click **OK**.
14. On the **Access Rights** dialog box, check the results of applying these access rights by doing the following:
    a) Double-click **Instrumentation** to display the access levels for that group.
    b) Expand **Unit Level** by clicking the ⊞ icon to display all the units in the domain.
    c) Click **New Refinery/Crude Area/Crude Unit 1**.

    In the **Mode** column of the right pane, the **Process Data Change in Specs** entity for Crude Unit 1 should be defined as **Access Denied**. The same condition should apply for this unit in the ADMINISTRATORS group.
15. Click to save your selection to the database.

Caution

- At the end of this exercise, make sure that all the entities are returned to **Full (Add / Delete / Update)** access.

16. When done, click to close the **Access Rights** dialog box and return to the **Domain Definition** window.

Assign a Domain Logo

1. With the **Domain Administration** window open, click to open the **Domain Definition** window.
2. On the **Domain Definition** toolbar, click .

   The **Select Logo** dialog box appears as shown when opened for the first time.

3. Click **Browse** to open the **Select a Logo File** dialog box.
4. Navigate to the required logo (.bmp file) and click **Open** to display the in the **Logo preview** area of the **Select Logo** dialog box.
5. Click **OK** to assign the displayed logo with all your domain documents.

Generate Domain Administration Reports

As Domain Administrator, you can generate a number of reports concerning Domain Administration.

1. With the **Domain Administration** window open, on the **Reports** menu, click **Access Rights**.
2. On the **Generate Access Rights Report** dialog box, select the **Unit** tab.
3. Select the **ADMINISTRATORS** and **Instrumentation** rows for **Crude Unit 1** and click **OK**.

   Tip

   - To select non-adjacent rows, hold down the **Ctrl** key while making your selection.
4. At the print preview prompt, click **Yes**.
5. When done, on the **File** menu, click **Exit** to exit the Administration module.

This completes the Domain Administration activities required for your domain configuration. You will perform all subsequent tasks in SmartPlant Instrumentation.

General Instrument Engineering Tasks

Opening SmartPlant Instrumentation

Caution

- Before starting SmartPlant Instrumentation, ensure that the Domain Administrator has created a unit and a user group, and defined the naming conventions for the unit and access rights for the group.

Note

- If find that you are unable to perform certain tasks, make sure that the Domain Administrator has granted you appropriate access rights.

Log on to SmartPlant Instrumentation

For the plant hierarchy example that you have created for this Tutorial, the lowest plant hierarchy level is Unit. When you log on to SmartPlant Instrumentation, you must select a unit in which you are going to work.

1. On the Windows **Start** menu, point to **Programs**, point to **Intergraph SmartPlant Instrumentation**, and then click **SmartPlant Instrumentation**.
2. On the **Logon Information** dialog box, type **USER1** as both your user name and password. (You can also enter as Domain Administrator by typing **DOMAIN** as your user name and **DA** as the password).
3. On the **Open** dialog box, expand the **DEMO_NEW** domain hierarchy to the unit level, as indicated below.

4. Click **OK** to open SmartPlant Instrumentation.

Define Units of Measure

1. On the **File** menu, click **Units of Measure and Accuracy** to display the **Units of Measure and Accuracy** dialog box.
2. On the **Units and Measure and Accuracy** dialog box, select the desired level of accuracy and units in the list for each quantity, and type a value for the ambient temperature as shown in the following screen shots.

Units of Measure and Accuracy

Find property:

Properties	Accuracy	Units of Measure	UOM Flag	Default Values
Ambient Temperature	10%	°C		22.00000
Area	0.1%	cm²		
Barometric Pressure	0.01%	atm(stand)		1.00000
Base Pressure	0.1%	bar		1.00000
Base Temperature	10%	°C		15.50000
Beta Ratio	1%			
Calibration Settings	1%			
Compressibility factor	1%			
Conductivity	0.1%	S/cm		
Control Valve Coefficients	0.1%			
Control Valve Cv and Kv	1%			
Control Valve Opening	1%	%		
Critical Pressure	0.1%	bar	gage	1.00000
Density	0.1%	kg/m³		
Design Pressure	1%	bar	gage	
Diameter	0.1%	cm		
Diff. Pressure (Control Valve)	0.1%	bar		
Diff. Pressure (DP instrument)	0.1%	psi		

Apply to all units

OK | Cancel | Copy From... | Print | Help

Units of Measure and Accuracy

Find property:

Properties	Accuracy	Units of Measure	UOM Flag	Default Values
Diff. Range (Flow)	0.1%	mmH2O 4°C		
Flow Meter Coefficients	0.1%			
Frequency	10%	1/s		
Gas Flow	0.1%	kg/h		
Latent Heat	0.1%	kcal IT/kg		
Length	0.1%	mm		
Liquid Flow	0.1%	m³/h	@flow	
Molecular Mass	0.1%			
Other	0.1%			
Percentage	0.01%	%		
Pressure	0.1%	bar	gage	
Relief Valve Coefficients	0.1%			
Resistance/Length	0.1%	MOhm/km		
Return Pressure	0.1%	bar	gage	
Return Temperature	0.1%	°C		
Sound Level	1%	dBA		
Specific Gravity	1%			
Specific Heat	0.1%	kcal M/kgK		

Apply to all units

OK | Cancel | Copy From... | Print | Help

3. Click **OK** to accept the values and close the dialog box.

Creating Instruments and Control Loops

You can perform most of the operations needed to create instruments and control loops from the Explorer windows. First, you must define values for supporting table properties from the Instrument Index module.

Task 1 — Defining Instrument Index Supporting Tables and Properties

Defining Instrument Types

1. Start the Instrument Index module by doing one of the following:
   - On the main toolbar, click .
   - On the **Modules** menu, click **Instrument Index**.
2. On the **Tables** menu, click **Instrument Types**.
3. On the **Instrument Types** dialog box, from the **Process function** list, select **Flow**.

4. Scroll down the dialog box and check that definitions exist for the **FE** and **FT** instrument types.
5. Create a new instrument type as follows:
   a) Click **New** to add a new line.

b) Type the instrument type name (for example, **FE**).

c) Press the **Tab** key and type in the description (for example, **D/P TYPE FLOW ELEMENT**).

d) Press the **Tab** key again and type a value in the **CS Tag Instrument Type Alias** column (with the same value as you typed in the **Instrument Type** column).

e) Click **Apply**.

6. Make sure that all the instrument types that appear in the table below have been defined. Perform steps 5a to 5e if any instrument types do not exist.

Notes

- The **Description** field is a required and unique part of an instrument type definition. If your SmartPlant Instrumentation database contains instrument types that differ from those shown in the table, even in the **Definition** field, make sure to create additional instrument types and profiles as instructed.
- When prompted to save a new instrument type, click **Yes**.

Process Function	Instrument Type	Description
Flow	FE	D/P TYPE FLOW ELEMENT
Flow	FI	LOCAL FLOW INDICATOR
Flow	FT	D/P TYPE FLOW TRANSMITTER
General	FY, HY, LY, TY	I/P TRANSDUCER
Level	LT	LEVEL TRANSMITTER
Pressure	PI	PRESSURE GAUGE
Pressure	PSH	HIGH-PRESSURE SWITCH
Pressure	PT	PRESSURE TRANSMITTER
Temperature	TE	THERMOCOUPLE
Temperature	TI	BI-METAL THERMOMETER
Temperature	TT	TEMPERATURE TRANSMITTER
Temperature	TW	THERMOWELL
Control Valve	FV, HV, LV, PV, TV	CONTROL VALVE
Control Valve	ZY	SOLENOID VALVE
Relief Valve	PSV	PRESSURE RELIEF VALVE

7. When done, click **OK** to accept the new values and close the dialog box.

Define Reference Wiring for a Field Device

You are going to specify the automatic creation of device panels, wires, and connections when you create instruments of type FT (D/P TYPE FLOW TRANSMITTER) by defining these options in the instrument type profile. Prior to doing this, you must define appropriate reference device panels and cables in your database, as described in this procedure.

1. On the **Tools** menu, click **Reference Explorer**.
2. Click the **Panels** folder to expand it and right-click the **Device Panels** folder.
3. On the shortcut menu, click **New** > **Device Panel (Conventional)**.
4. On the **Device Panel Properties** dialog box, type the information as shown.

5. In the **Reference Explorer** window, click the ⊞ icon to expand the **Device Panels** folder.
6. Right-click the **REF FIELD DEVICE 2-WIRE** panel.
7. Create a strip and terminals for this device panel by doing the following:
   a) On the shortcut menu, click **New** > **Terminal Strip**.
   b) On the **Strip Terminal Configuration** dialog box, click **New**.

c) On the **Terminals in Pattern** dialog box, enter **2** as the number of terminals in the pattern, and then click **OK** to return to the **Strip Terminal Configuration** dialog box.

d) In the **Configuration name** box, type **DP 2 Terminals**.

e) Enter the other values in this dialog box as shown.

f) Click **Save**.

g) Click **Create**.

h) On the **Terminal Strip Properties** dialog box, in the **Terminal Strip** box, type **TS - 1**.

i) Click **OK** and double-click the panel and then the strip to expand the panel-strip-terminal hierarchy in the **Reference Explorer** window.

The window should appear as shown.

8. Create a reference cable by doing the following:
   a) Right-click the **Cables** folder and on the shortcut menu, click **New** > **Cable**.
   b) On the **Cable Configuration** dialog box, click **New**.
   c) On the **New Cable Configuration** dialog box, enter **1** as the total number of sets and select **PAIR W/SHIELD** as the default cable set type.
   d) Click **OK**.
   e) In the **Cable configuration** box, type **DP 2 Wires**.
   f) In the **Cable Set** column, type **PR #1**.
   g) In the **Cable default name** box, type **1P#20 BK,WH I/S**.
   h) Under **Cable set details**, check the wire tag label is **SPARE** for the first two wire tags and **Shield** for the third wire tag.

   The dialog box should now appear as shown.

i) Click **Save**.

j) Click **Create**.

k) On the **Cable Properties** dialog box, enter information as shown and click **OK**.

Define the FT (D/P TYPE FLOW TRANSMITTER) Instrument Type Profile

At this stage, before creating new tags in the Instrument Index module, we will populate the supporting tables for our instrument types. First of all, we will define the instrument type profiles to ensure automatic device panel creation when the tags are created.

1. On the **Tables** menu, click **Instrument Types** to open the **Instrument Types** dialog box.

   Tip

   - If you cannot find the desired instrument type, create it as described on page 37.

2. From the **Process function** list, select **Flow**.
3. Scroll down the **Instrument Type** list and select instrument type **FT (**with description **D/P TYPE FLOW TRANSMITTER)**.
4. Click **Profile** to open the **Instrument Type Profile** dialog box.
5. Click the **General** tab and do the following:

   a) In the **Instrument specifications** group box, select the **Include instrument specification** check box.

   b) From the **Instrument specification** list, select **Diff. Pressure Instr. (flow)** – Form No. **56**.

   c) In the **Hook-ups** group box, select the **Include hook-ups** and **Include in BOM** check boxes.

   d) In the **System I/O type** group box, select the **Include system I/O type** check box and select **AI** from the list.

   e) In the **Location** group box, select the **Include location** check box and select **Field** from the list.

The **General** tab of the **Instrument Type Profile** dialog box should now appear as shown.

6. Click the **Wiring and Control System** tab.
7. Do the following to define basic parameters and select a reference device panel for tags that you will base on this instrument type:
   a) Select the **Include wiring** and **Control system** check boxes.
   b) From the **Reference device panel** list, select **REF FIELD DEVICE 2-WIRE**.
8. Do the following to select the reference device cable and its connections:
   a) In the **Conventional connections** group box, click **New**.
   b) On the **Conventional Connection Properties** dialog box, from the **Reference cable** list, select **1P#20 BK, WH I/S**.
   c) From the **Cable set** list, select **PR #1**.
   d) Accept the default selection of **Apply to subsequent cable sets**.
   e) From the **Terminal strip** list, select **TS - 1**.
   f) From the **Starting terminal** list, select +.

g) From the **Connection type** list, select **2 In a row**.

h) Accept the default selection of **Propagate tag signal**.

The **Conventional Connection Properties** dialog box should now appear as shown.

i) Click **OK** to save your values and close the **Conventional Connection Properties** dialog box.

The **Wiring and Control System** tab of the **Instrument Type Profile** dialog box should now appear as shown.

Tips

- The above parameters are based on the assumption that the device panel to be created will be a two-terminal field device with two wires connected to it.
- The panel name and cable name are defined in the Wiring module. You will learn more about these later.

9. Click **OK** to return to the **Instrument Types** dialog box.
10. Click **Apply**.

Define the FY (I/P TRANSDUCER) Instrument Type Profile

1. On the **Instrument Types** dialog box, from the **Process function** list, select **General**.
2. Scroll down the **Instrument Type** list and select instrument type **FY (**with description **I/P TRANSDUCER)**.
3. Click **Profile** to open the **Instrument Type Profile** dialog box.

4. Click the **General** tab and do the following:
   a) In the **Hook-ups** group box, select the **Include hook-ups** and **Include in BOM** check boxes.
   b) In the **System I/O type** group box, select the **Include system I/O type** check box and select **AO** from the list.
   c) In the **Location** group box, select the **Include location** check box and select **Field** from the list.

   The **General** tab of the **Instrument Type Profile** dialog box should now appear as shown.

5. Click the **Wiring and Control System** tab.
6. Do the following to define basic parameters and select a reference device panel for tags that you will base on this instrument type:
   a) Select the **Include wiring** and **Control system** check boxes.
   b) From the **Reference device panel** list, select **REF FIELD DEVICE 2-WIRE**.
7. Do the following to define the reference device cable and its connections:
   a) In the **Conventional connections** group box, click **New**.

b) On the **Conventional Connection Properties** dialog box, from the **Reference cable** list, select **1P#20 BK, WH I/S**.

c) From the **Cable set** list, select **PR #1**.

d) Accept the default selection of **Apply to subsequent cable sets**.

e) From the **Terminal strip** list, select **TS - 1**.

f) From the **Starting terminal** list, select +.

g) From the **Connection type** list, select **2 In a row**.

h) Accept the default selection of **Propagate tag signal**.

i) Click **OK** to save your values and close the **Conventional Connection Properties** dialog box.

8. Click **OK** to return to the **Instrument Types** dialog box.
9. Click **Apply**.

Define the FV (CONTROL VALVE) Instrument Type Profile

1. On the **Instrument Types** dialog box, from the **Process function** list, select **Control Valve**.
2. Scroll down the **Instrument Type** list and select instrument type **FV (**with description **CONTROL VALVE)**.
3. Click **Profile** to open the **Instrument Type Profile** dialog box.
4. On the **General** tab, do the following:

   a) In the **Instrument specifications** group box, select the **Include instrument specification** check box.

   b) From the **Instrument specification** list, select **Control Valve** – Form No. **1**.

   c) In the **Hook-ups** group box, select the **Include hook-ups** and **Include in BOM** check boxes.

   d) In the **Location** group box, select the **Include location** check box and select **Field** from the list.

   The **General** tab of the **Instrument Type Profile** dialog box should now appear as shown.

5. Click **OK** to return to the **Instrument Types** dialog box.
6. Click **Apply**.

Define Additional Instrument Type Profiles

Use the table below to adapt the above procedures for all the following instrument types. Carefully read the notes below the table before beginning.

Process Function Type	Instr. Type	Description	Instrument Spec	Hook-Ups / BOM	Sys. I/O Type	Control System	Wiring
General	FY, HY, LY, TY	I/P TRANSDUCER	–	Y	AO	Y	Y
Flow	FE	D/P TYPE FLOW ELEMENT	–	Y	–	–	–
Flow	FI	LOCAL FLOW INDICATOR	–	–	–	–	Y
Flow	FT	D/P TYPE FLOW TRANSMITTER	56	Y	AI	Y	Y
Level	LT	LEVEL TRANSMITTER	–	Y	AI	Y	Y
Pressure	PI	PRESSURE GAUGE	–	Y	–	–	–
Pressure	PSH	HIGH-PRESSURE SWITCH	24	Y	DI	Y	Y

Process Function Type	Instr. Type	Description	Instrument Spec	Hook-Ups / BOM	Sys. I/O Type	Control System	Wiring
Pressure	PT	PRESSURE TRANSMITTER	–	–	AI	Y	Y
Temperature	TE	THERMOCOUPLE	–	Y	–	–	Y
Temperature	TI	BI-METAL THERMOMETER	–	Y	–	–	–
Temperature	TT	TEMPERATURE TRANSMITTER	–	Y	AI	Y	Y
Temperature	TW	THERMOWELL	–	–	–	–	–
Control Valve	FV, HV, LV, PV, TV	CONTROL VALVE	1	Y	–	–	–
Control Valve	ZY	SOLENOID VALVE	2	Y	DO	Y	Y
Relief Valve	PSV	PRESSURE RELIEF VALVE	7	–	–	–	–

Notes

- For **Control System** and **Hook-Ups / BOM**, if the value in the table is **Y**, select the appropriate check boxes in the **Instrument Type Profile** dialog box.
- For **Wiring**, if the value in the table is **Y**, use the definitions that you applied for FT (D/P TYPE FLOW TRANSMITTER) above.
- For **Instrument Spec**, if the value in the table is not '-', in the **Instrument specifications** section of the **General** tab, select **Include instrument specification**, and then from the **Instrument specification** list, select the form indicated. If the value is '-', clear the **Include instrument specification** check box.
- For **System I/O Type**, if the value in the table below is not '-', on the **General** tab, select **Include system I/O type**, and from the list, select the I/O type indicated.
- For all of the instrument types in the table, in the **Location** section of the **General** tab, select **Include location**, and then from the list, select **Field**.

Create a P&ID Drawing Number

1. On the **Tables** menu, click **P&ID Drawing References**.
2. On the **P&ID Drawing References** dialog box, click **New** to add a new data row.
3. Complete the information for the P&ID (drawing number) and description as follows:
   a) In the **P&ID Drawing** column, type **101-PID01-001**.
   b) In the **Description** column, type **P&ID TUTORIAL FOR DEMO PROJECT**.

The dialog box should appear as shown.

4. Click **OK** to save the data and close the dialog box.

Create Lines (Piping)

Lines and line data are defined in the Process Data module since these definitions are usually the responsibility of process engineers. We will define line numbers in the Instrument Index, so that it will be possible to associate instruments with the lines on which they are located.

1. On the **Tables** menu, click **Lines**.
2. On the **Lines** dialog box, beside the **Line type** list arrow, click ... to open the **Line Types** dialog box.
3. Click **New** to add a new data row.
4. In the **Line Type** data field, type **PROCESS**.
5. Enter another line type — **UTILITIES —** as shown.

6. Click **OK** to save the data and close the dialog box.
7. On the **Lines** dialog box, from **Line Type** list, select **PROCESS**.
8. Click **New** to open the **Line Properties** dialog box.
9. Add the following information:
   a) In the **Line number** box, type **4"-P-1501-11H**.
   b) From the **Pipe material** list, select **Plain Carbon Steel (ANSI…)**.
   c) Select **ANSI** as the pipe standard.

      The **Pipe Data** dialog box opens.
   d) In the **Find nominal size** box, type **4** so that you can easily select 4 inches as the nominal size.
   e) Select the pertinent pipe data as shown.

   f) Click **OK** to close the **Pipe Data** dialog box and return to the **Line Properties** dialog box.

All other pipe details are filled in automatically and the **Line Properties** dialog box should appear as shown.

10. Click **OK** to select the line settings and to return to the **Lines** dialog box.
11. Add another line — **4"-P-1502-11H** — with the same settings.

Note

- You can also create lines in the Process Data module.

The remaining objectives in this task deal with entering additional data in the supporting tables. These include **Status**, **I/O Type**, **Location**, **Manufacturer**, and **Model**.

Enter Status Data

1. On the **Tables** menu, click **Instrument Statuses** to open the **Instrument Statuses** dialog box.
2. Click **New** to add a new data row.
3. Type **N** in the **Instrument Status** column.
4. Type **New Instrument** in the **Description** column.

The dialog box should appear as shown.

5. Click **OK**.

Enter I/O Type Data

1. On the **Tables** menu, click **System I/O Types**.
2. On the **System I/O Types** dialog box, click **New** to add a new data row.
3. In the **System I/O Type** column, type **T/C**.
4. In the **Description** column, type **T/C Input**.
5. Click **OK**.

Enter Location Data

1. On the **Tables** menu, click **Instrument Locations**.
2. On the **Instrument Locations** dialog box, select the **Field** instrument location.
3. In the **Description** column, type **Installed in the field**.
4. Click **OK**.

Enter Manufacturer Data

1. On the **Tables** menu, click **Instrument Manufacturers** to open the **Instrument Manufacturers** dialog box.
2. Click **New** to add a new data row.
3. In the **Instrument Manufacturer** column, type **ROSEMOUNT**.
4. Enter additional manufacturers **ASCO** and **ASHCROFT**.

Your dialog box should appear as shown (the rest of the data already appears by default).

5. Click **OK**.

Enter Model Data

1. On the **Tables** menu, click **Instrument Models** to open the **Instrument Models** dialog box.
2. From the **Manufacturer** list, select **ASCO**.
3. Click **New** to add a new data row.
4. In the **Instrument Model** column, type **814B12**.
5. In the **Process Function** column, select **General** from the list.
6. If desired, type a model description.
7. Enter additional models for various manufacturers as shown (note that the data for FISHER may already be present).

Manufacturer	**Model**	**Process Function**
ASCO	8351B23	General
ASHCROFT	EVERY-ANGLE-12/01	General
	EVERY-ANGLE-13/02	General
	MGS - 136	General
FISHER	ED	Control Valve
	EZ	Control Valve
	8500	Control Valve
	V500	Control Valve

Manufacturer	Model	Process Function
FISHER	V150	Control Valve
ROSEMOUNT	1151DP4E22S2B1M2	General
	3051S1256	General

8. Click **OK**.

This completes the basic supporting table data required for this tutorial. If you want to add more data, do so according to the above instructions.

Task 2 — Defining Loops with Their Tags

In this section, you are going to explore various methods of defining control loops with their associated tag numbers. You will perform most of these tasks from the **Domain Explorer**.

Define a Loop and Add Tag Numbers

1. On the **Tools** menu, click **Domain Explorer**.
2. Under **Domain Explorer**, expand the plant hierarchy **New Refinery** > **Crude Area** > **Crude Unit 1**.
3. Under **Crude Unit 1**, right-click the **Loops** folder, and on the shortcut menu, click **New** > **Loop**.
4. On the **New Loop Number** dialog box, type the loop number as shown.

Tips

- Use the **Tab** key to move the cursor to the character separators.
- In this case, the first segment of the loop number name has been derived from the unit number as defined on the **Loop Naming Convention** dialog box of the Administration module by the Domain Administrator.

5. Click **OK**.

The **Loop Number Properties** dialog box opens.

6. Enter the following information:
   a) In the **Loop service** box, type **Feed from V8**.
   b) From the **Loop type** list, select **DCS**.
   c) From the **P&ID drawing** list, select **101-PID01-001**.
   d) Select the **Apply P&ID drawing to tags** and **Apply service to tags** check boxes.

   The dialog box should appear as shown.

Tip

- To add new values to supporting tables so that they can become available in the lists, click ... next to the lists.

7. Click **OK**.
8. Click **Yes** when prompted to create a tag number.
9. On the **New Tag Number** dialog box, enter **FE** as the first new tag number on the **New Tag Number** dialog box.

10. Click **OK**.

    Because more than one instrument type designated by **FE** acronym exists, the **Select Instrument Type** dialog box opens.

11. Do the following:

    a) From the **Description** column, select **D/P TYPE FLOW ELEMENT**.

    b) Click **OK** to open the **Tag Number Properties** dialog box.

12. Enter tag number data by selecting values from the lists as shown.

Tip

- The ... command buttons enable you to enter additional data in the supporting data tables. These values will then become available in the lists.

13. Click **Apply** when done.
14. Click **New** to add another tag number to the loop.
15. To create tag number 101-FT-100, repeat steps 9-13 with the following differences:

    a) On the **Select Instrument Type** dialog box, select from the **Description** column for instrument type **FT** the description **D/P TYPE FLOW TRANSMITTER**.

Tip

- While this tag number is being created, the status bar indicates automatic device panel creation.

    b) On the **Tag Number Properties** dialog box, complete the definitions as shown.

Tip

- To display additional manufacturers and models, you must first define them using the appropriate ellipsis buttons on the **Tag Number Properties** dialog box. (See also page 54.)

16. To create tag number 101-FY-100, repeat steps 9-13 with the following differences:

    a) On the **Select Instrument Type** dialog box, select from the **Description** column for instrument type **FY** the description **I/P TRANSDUCER**.

    b) On the **Tag Number Properties** dialog box, complete the definitions as shown.

17. After completing all your tag definitions, click **OK** to close the dialog box.

In the next task, you will see how to add a tag number to a loop after saving the loop.

Add a Tag Number to an Existing Loop

In this example, you will add a control valve (FV) to loop 101-F-100.

1. On the **Tools** menu, click **Domain Explorer**.
2. Under **Domain Explorer**, expand the plant hierarchy **New Refinery** > **Crude Area** > **Crude Unit 1**.
3. Under **Crude Unit 1**, right-click the **Instruments** folder, and on the shortcut menu, click **New** > **Instrument**.
4. On the **New Tag Number** dialog box, type tag number **101-FV-100** and click **OK**.
5. Ensure that the data on the **Loop Name** dialog box appears as shown.

6. Click **OK**.
7. On the **Tag Number Properties** dialog box, define the tag number as shown.

8. Click **OK**.

Add a Single Tag Number

1. On the **Tools** menu, click **Domain Explorer**.
2. Under **Domain Explorer**, expand the plant hierarchy **New Refinery** > **Crude Area** > **Crude Unit 1**.
3. Under **Crude Unit 1**, right-click the **Instruments** folder, and on the shortcut menu, click **New** > **Instrument**.
4. On the **New Tag Number** dialog box, type tag number **101-PT-201**, and click **OK**.

   Tip

   - If the **Select Instrument Type** dialog box opens, select from the **Description** column for instrument type **PT** the description **PRESSURE TRANSMITTER**, and then click **OK**.
5. On the **Loop Name** dialog box, accept the loop name **101-P-201** and click **OK**.
6. On the **Loop Number Properties** dialog box, accept the existing values and click **OK**.

7. On the **Tag Number Properties** dialog box, do the following to create the equipment entry that you need:
   a) Beside **Equipment**, click ... to open the **Equipment** dialog box.
   b) From the **Equipment Type** list, select **TANK**.
   c) Click **New** to add a new row.
   d) In the **Equipment** column, type **F-102**.
   e) Click **OK**.
8. On the **Tag Number Properties** dialog box, complete the data entry as shown.

9. Click **OK**.
10. In the **Domain Explorer**, under **Crude Unit 1**, expand the **Loops** folder.

Tip

- After creating a new entity, refresh the **Domain Explorer** display by selecting the appropriate folder, and then, on the toolbar, click **Refresh** .

11. Select loop **101 P - 201**.
12. Right-click the loop and on the shortcut menu, click **Properties**.

13. On the **Loop Number Properties** dialog box, do the following to add a loop function to the **Loop function** list:
    a) Beside **Loop function**, click ...
    b) On the **Loop Functions** dialog box, click **New** to add a new line.
    c) In the **Loop Function** column, type **IA**.
    d) In the **Description** column, type **Indication and Alarm**.
    e) Click **OK**.
14. Enter the remaining loop data as shown.

15. Click **OK**, and if prompted to edit tag numbers, click **No**.

Task 3 — Duplicating a Loop with Its Tag Numbers

Select a Loop and View Tag Number Data

1. On the **Domain Explorer** toolbar, click .
2. On the **Search** dialog box, do the following:
    a) From the **Entity type** list, select **Loop**.
    b) In the **Entity name** box, type ***100***.

c) Click **Search Now**.

d) Under **Results**, select loop **101-F –100**.

The dialog box should appear as shown.

e) Click **Go to Entity**.

f) Click **Close**.

3. In the **Domain Explorer**, select loop **101-F -100**.
4. In the lower pane, select the instruments as shown.

5. Right-click, and on the shortcut menu, click **Properties**.
6. View the properties of each instrument on the **Tag Number Properties** dialog box, using the **Next** and **Previous** buttons to navigate to the instruments.

Duplicate a Loop

If you want to create a series of control loops that perform similar functions, rather than creating each one from scratch, you can use the SmartPlant Instrumentation duplication feature to copy the loop and its associated tag numbers.

1. In the **Domain Explorer**, select loop **101-F -100**.
2. Right-click the loop and on the shortcut menu, click **Duplicate**.
3. On the **Duplicated Loop Number** dialog box, change the number of the loop from **100** to **2212**.

4. Click **OK** to open the **Loop Number Properties** dialog box.
5. In the **Loop service** field, type **Feed to B-101 Pass A**. Ensure that the rest of the information appears as follows.

6. Click **OK** to display the **Create Loop Tags** dialog box.
7. Select the **Select all** and **Copy the instrument type from source tag numbers** check boxes and click **OK** to start the duplication process.

8. In the message prompting you to edit the tags, click **Yes**.

   The **Tag Number Properties** dialog box opens displaying data of the first tag associated with the new loop.
9. Edit the tag data as needed. Make sure that you create and select line **3"-FO-1212-4C** as displayed in the P&ID (DWG. No. 101-PID01-001).

   Tip

   - When you create line **3"-FO-1212-4C**, you have the option of defining it as in the in_demo.db file, but this is not required.
10. Click **Apply** when you have finished editing the tag data.
11. Click **Next** to display the data of the next tag. Continue editing all the tags associated with the loop.
12. Click **OK** after editing the last tag number associated with the loop.

Task 4 — Creating Plant Loops in Batch Mode

This feature enables you to create user-defined typical loops and tags which are used to create numerous plant loops in batch mode. A typical loop can be described as a user-defined combination of typical tags that are in fact various instrument types. You create your user-defined typical loop which is then used as a 'template' for batch creation of plant loops in batch mode. This feature can be very useful when you need to create numerous loops based on the same 'template'. You can create as many typical loops as required and use them to create plant loops in batch mode.

Creating Loops and Tag Numbers in Batch Mode

Naming Conventions

For typical loops, you can use any naming convention so long as it contains the **loop number** segment.

The maximum length of a typical loop name is 50 characters, consisting of the following segments:

- Loop number — containing up to 48 characters.
- Delimiter — consisting of 1 character.
- Loop suffix — containing up to 2 characters.

A typical tag naming convention consists of the following segments:

- Instrument type — containing up to 4 characters in the **name** segment, (the typical tag **name** segment is automatically truncated to 4 characters when the instrument type name contains more than 4 characters).
- Tag number — containing up to 8 characters.
- Tag suffix — containing up to 3 characters.

An example of a typical tag name is **FT-FLOW_CTLR/2**.

Create Typical Loop FL-1

Creating a new typical loop is the first stage in the procedure. This is when you make all the required typical loop definitions: enter the typical loop name, select its measured variable and loop function. At this stage, you also select the appropriate instrument types and related profiles that will serve as the typical tags contained in this typical loop.

At this point, you will create a new typical loop **FL-1**.

1. With the **Instrument Index Module** window open, on the **Edit** menu, click **Typical Loop Management**.
2. On the **Typical Loop Management** dialog box, click **New** to open the **Typical Loop Properties** dialog box.

3. Enter data as follows:
   a) In the **Name** field, type **FL-1**.
   b) From the **Loop measured variable** list, select **FLOW** as the loop measured variable for the new typical loop. Click ... to define a new loop measured variable if the required value is not available.
   c) From the **Loop type** list, select **DCS**. Click ... to define a new loop type if the required value is not available.
   d) From the **Loop function** list, select **Control Indicating** as the loop function for the new typical loop. Click ... to define a new loop function if the required value is not available.
   e) In the **Loop Service** field, type **DCS closed loop**.
   f) In the **Note** field, type a short note if needed.
4. Click **Add** to add a typical tag to the loop.
5. In the **Associated typical tags** data window, select from the **Instrument Type** list: **D/P TYPE FLOW ELEMENT - FE**. This instrument type will serve as a typical tag associated with the current typical loop.

Tip

- The **Number** field automatically displays the numeric segment of the typical tag name which is derived from the typical loop name. You can accept it or modify it as needed.

6. Click **Add** to add another typical tag to the current loop.
7. From the **Instrument Type** list, select D/P TYPE FLOW TRANSMITTER - FT.
8. Click **Add** again.
9. Add the following typical tags:
   a) I/P TRANSDUCER – FY.
   b) CONTROL VALVE – FV.

The **Typical Loop Properties** dialog box should appear as shown.

Tip

- The suffix is used when there is more than one instrument that belongs to the same instrument type. In this case, type a suffix to distinguish between these instruments.

10. Click **OK** and then click **No** to close the prompt.

This completes the procedure for creating typical loop **FL-1**.

Create Typical Loop T-10

Now you will create typical loop **T-10** with properties as shown.

1. If necessary, create a new loop function: **IA – Indication and Alarm**.
2. After you finish creating the required typical loop numbers, click **Close** in the **Typical Loop Management** dialog box.

Create Plant Loops in Batch Mode

This feature enables you to create, in batch mode, numerous plant loops based on existing typical loops. You will now create several plant loops based on typical loop FL-1.

1. With the **Instrument Index Module** window open, on the **Edit** menu, click **Batch Loop Creation**.

   The **Batch Loop Creation** dialog box opens.
2. From the **Typical Loop** list, select **FL-1**.
3. In the **New Loop Number** field, type: **905-910, 919**.

This will create Loops F-905 through F-910 and F-919.

Tips

- You can create as many plant loops based on the selected typical loop as required. To create more than one plant loop, simply type the required new plant loop number names in the **New Loop Number** field. You can enter a range of loop names or a list as needed.
- For noncontiguous alphanumeric or numeric loop numbers, type in the loop numbers with commas between them. For a range of numeric loop numbers, type in the range of loop numbers with a hyphen between them. For example, to create loops 1000, 1001, 1002, 1003, 1057, FT10, F11,and F12 type: **1000-1003, 1057, FT10, F11, F12**.

4. Click **Create** to start the plant loop creation.

   The software displays a progress bar indicating the creation progress. When the creation of all Plant loops is complete, the progress bar is replaced with an appropriate message and the **Done** check box of the appropriate data row is selected.

   Tip

   - If a problem occurs during the plant loop creation process, a message is displayed informing you that a particular set in the specified row has not been created. Also, the **Done** check box for that row remains unselected.

5. Once the batch creation process is complete, click **Close** to close the **Batch Loop Creation** dialog box.
6. Open the **Browser** to view the results.
7. Now create plant loops T-905 through T-910 and T-919 based on typical loop T-10.

Creating Additional Loops and Tag Numbers

Using the procedures described in this section, create the following loops and their associated tags according to the Demo P&ID on page 285. Make sure you create all the instrument tags shown in the Demo P&ID, because you will make use of them in Tutorial tasks later. Use suitable methods of loop and tag creation among those you learned in Task 2 and Task 3). On completion, the following loops should appear in your database.

F-100	F-102 (Duplicate loop F-100 and select only the appropriate source tags.)	P-201
F-2212	F-2213	F-201
P-208		L-201
P-100	P-101	

Notes

- In the course of creating the tags, if the **Select Instrument Type** dialog box opens to offer you a choice of instrument types, make sure that you select the instrument type/description for which you defined the profile.
- The bolded loops are the ones that you have already created.

At this stage, you have completed building your Instrument Index. You should now feel comfortable with the SmartPlant Instrumentation environment and have a good understanding of the Instrument Index functions. The next part of your training will take you to the Process Data module (like in a real-life domain) where you will assign process data to your instruments.

Defining Process Data, Sizing, and Specifications

From the Process Data and Calculation modules, you (or the process engineer) can create lines, fill in line process data, and then associate it with the instruments located on these lines. This process data will then be used in the Calculation and Specifications modules.

- To open the Process Data module, do one of the following:
  - On the main toolbar, click .
  - On the **Modules** menu, click **Process Data**.

Task 1 — Defining Process Data

Define Process Data for the Lines (Piping)

At this stage, it is assumed that you have already created lines **4"-P-1501-11H** and **4"-P-1502-11H**. If you haven't done so yet, you can also create a new line in the Process Data module by doing the following:

1. Click to open the **Select Line** dialog box.
2. From the **Line Type** list, select **PROCESS**.
3. Click **New** to open the **Line Properties** dialog box.
4. Define new line **4"-P-1501-11H** and click **OK** to return to the **Select Line** dialog box.
5. On the **Select Line** dialog box, click **OK** again to complete the new line creation process.
6. If necessary, repeat the steps to define line **4"-P-1502-11H**.

Define Process Data for a Line

1. Click to open the **Select Line** dialog box.
2. Select the **Show All Line Types** check box.

All the lines defined for the plant are shown.

3. Select line number **4"-P-1501-11H** and click **OK**.
4. On the **Process Data** dialog box, from the **Fluid state** list, select **Liquid** and click **OK**.
5. In the **Line Process Data - 4"-P-1501-11H** window that opens, in the **GENERAL** section, ensure that the values appear as shown. If any values are missing, enter them now.

6. In the **PROPERTIES** section, enter the property values in metric units of measure, as shown.

PROPERTIES

Report flags:	Density				
	@Minimum	@Normal	@Maximum	Units	
Volumetric flow:	25	30	32	m³/h	@flow
Pressure:	12	13	14	bar	gage
Temperature:	150	150	150	°C	
Viscosity:	0.1	0.1	0.1	cP	
Velocity	0.936	1.12	1.2	m/s	
Density:	890	890	890	kg/m³	
Specific gravity:	0.891	0.891	0.891		
Compressibility:					
Specific heat:	1	1	1	BtuIT/lb°F	
Specific heat ratio:					
Vapor pressure:	0.9	0.9	0.9	bar	gage
Critical pressure:	1200			bar	gage
Molecular mass:					

7. Click .
8. Click to open the **Process Data Report** window where you can print preview your line process data sheet.
9. When prompted to preview the report, click **Yes**.
10. Examine the data. You can do the following:
    - Change the magnification level by clicking .
    - Save the report as an external file by clicking .
    - Print out a single report by clicking or print out all displayed reports
      by clicking .
11. Click to open the **Revisions** dialog box.
12. Edit the revision data as follows:
    a) Select **Revision method 0,1,2…** and click **New**.

Tip

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

b) In the **Description** field, type **For Instrumentation**.

The dialog box should appear as follows.

13. Click **OK** to close the **Revisions** dialog box.

Tip

- After you have made several revisions, you can update them in batch mode. In the main window of any SmartPlant Instrumentation module, on the **Tools** menu, click **Global Revisions**. For details, see **Add Revisions Globally**, page 107.

14. Close the print preview window.
15. On the module toolbar, click to save the displayed process data.
16. To reopen the **Select Line** dialog box in order to view the list of lines, click again.

Define Process Data for Flowmeter and Control Valve

Instrument process data is derived from line process data in SmartPlant Instrumentation. We will now associate line process data with instruments.

1. On the module toolbar, click .
2. On the **Enter Tag Number** dialog box, type the instrument tag number **101-FE - 100** and click **OK**.

Tip

- Use the **Tab** key to move the cursor to the character separators.

3. At the **Process Data** prompt to copy the data from the line, click **Yes**.

   The software copies all the process data and you should see the window with all the data as shown (you can use US units of measure if appropriate).

Flowmeter Process Data - 101-FE -100

GENERAL

Case:			
Service:	Feed from V-8	Location:	Line
Fluid state:	Liquid	Line number:	4"-P-1501-11H
Fluid phase:	Single phase	Line size:	4 in
Fluid name source:	User-defined	Line schedule:	80
Fluid name:	Lean Feed		

PROPERTIES

Report flags: Specific Gravity

	@Minimum	@Normal	@Maximum	Units	
Volumetric flow:	25	30	32	m³/h	@flow
Upstream pressure:	12	13	14	bar	gage
Temperature:	150	150	150	°C	
Viscosity:	0.1	0.1	0.1	cP	
Velocity:	0.936	1.12	1.2	m/s	
Density:	890	890	890	kg/m³	
Specific gravity:	0.891	0.891	0.891		
Compressibility:					
Specific heats ratio:					
Vapor pressure:	0.9	0.9	0.9	bar	gage
Critical pressure:	1200			bar	gage
Molecular mass:					

ADDITIONAL PROPERTIES

4. Scroll down to display the **Additional Properties** section, and enter the information as shown.

ADDITIONAL PROPERTIES

Design pressure minimum:		bar	gage	Corrosive:	No
Design pressure maximum:	50	bar	gage	Erosive:	No
Design temperature minimum:		°C		Toxic:	No
Design temperature maximum:	250			Colored:	
Entrained gas:		%		Transparent:	
Angle of repose:		°		Build-up tendency:	

Required range: From: 0 To: 36 m³/h @flow

Limits on press.drop across flowmeter: mmH2O 4°C

5. Click to generate and preview the process data report.
6. If prompted to save your data, click **Yes**.
7. Click and enter a revision for this process data sheet. (See page 77 for details.)
8. Repeat the previous steps for the control valve on the same line (instrument tag 101-FV - 100), with the following general and process data properties.

9. Scroll down to display the **Additional Properties** section, and enter the following information.

10. Click to generate and preview the process data report.
11. If prompted to save your data, click **Yes**.
12. Click and enter a revision for this process data sheet. (See page 77 for details.).
13. Repeat steps 1 through 8 for the flow transmitter on the same line (instrument tag 101-FT - 100), with the same process data for instrument tag 101- FE - 100.
14. Revise and save the process data sheet as you have previously done.

Note

- You have completed entering the basic process data required for this tutorial. Three process conditions are sufficient to continue with our next section - calculation and sizing of instruments.

Task 2 — Performing Calculations

Using the process data that you have entered in the Process Data module, you will work in the Calculation module to calculate the bore of the orifice plate 101-FE-100 and size your control valve 101-FV-100.

- To open the Calculation module, do one of the following:
  - On the main toolbar, click .
  - On the **Modules** menu, click **Calculation**.

Calculate Flowmeter Parameters

1. On the module toolbar, click .
2. On the **Enter Tag Number** dialog box, click **Find**.
3. On the **Find Tag** dialog box, click **Find**.

   Tip

   - Only those tag numbers that have process data and whose process function has been designated as **Flow** are listed.
4. Select tag number **101-FE -100** and click **OK**.
5. On the **Enter Tag Number** dialog box, click **OK**.

   Tip

   - The calculation sheet for the selected instrument is displayed with instrument process data already populated.
6. On the **Options** menu, click **Highlight Process Data**. This action highlights some of the process data values.

   Tip

   - Two colors are used — **turquoise** and **yellow**. The **turquoise** highlight means that this is a required field that must contain data. The yellow highlight means that this field should contain data, and you should enter it in this window.
7. On the module toolbar, click to open the **Flowmeter Calculation** dialog box, where you will calculate the orifice diameter.

8. On the **Flowmeter Calculation** dialog box, select, as shown below, the flowmeter type and sub-type. Also, select **Orifice diameter** as the parameter to be calculated and enter the differential range as shown.

9. Click **Calculate** to perform the calculation.

On completion of the calculation, the software displays the results.

10. When done, click **Close** to return to the **Flowmeter Calculation** window.
11. On the module toolbar, click .
12. Click to generate and preview the calculation report.
13. Click and enter a revision (for details, see page 77).
14. At this stage, we will make a back calculation for this bore (2.051") for a new constant of 0-40 m^3/h as follows:
    a) Click .

       The **Flowmeter Calculation** dialog box opens with the **Orifice Diameter** radio button selected (value = 0).
    b) Click **Calculate** to rerun the calculation so that a value for the orifice diameter now appears.
    c) Under **Select calculate field**, select **Differential range**.
    d) Type **40 m^3/h** as the full-scale flow.
    e) Click **Calculate** to calculate the differential pressure.

       A result of 3137 mmH_2O 4°C should be obtained.

15. Click **Close** to close the **Flowmeter Calculation** dialog box and return to the **Flowmeter Calculation** window.

Calculate and Size Control Valve Parameters

1. On the module toolbar, click .
2. On the **Enter Tag Number** dialog box, type the instrument tag number **101-FV -100** and click **OK** to display the calculation sheet for the instrument.

3. Make sure that all highlighted fields contain data as shown above.
4. Click to open the **Control Valve Calculation** dialog box where you calculate and size the control valve.

5. Enter data as shown.

6. Click **Calculate** to perform the calculation.

   The results are displayed on completion of the calculation as shown.

	@Minimum	@Normal	@Maximum
Cv	9.68	13.5	20.4
Regime	Incip.-Turb.	Turbulent	Turbulent
Noise	77.9	69.4	65.2

7. When done, click **Close** to return to the **Control Valve Calculation** window.
8. Click to generate and preview the calculation report.
9. Click to open the **Revisions** dialog box, and enter a revision (for details, see page 77).
10. To assign an external drawing number to the calculated control valve sheet, under **Document number**, accept the number as displayed or type a new one. The drawing number appears in the **Doc. No.** field of the calculation report.
11. Click **OK** or **Cancel** to close the **Revisions** dialog box.
12. Examine the report and print it out if needed.

You have now successfully completed the calculations required for this tutorial. The software will subsequently use the calculation results in the specification sheets of these instruments.

Task 3 — Viewing and Editing Specification Sheets

At this stage, we assume that you have already entered your instrument data, assigned process data to the flow instruments, and calculated the orifice plate and control valves.

In this part of the tutorial, you will:

- Select a specification form from the Spec Library and associate this form to instruments.
- Create and use form data templates.
- Create and use multi-tag specification formats.
- Customize specification forms.

- To open the Specifications module, do one of the following:
  - On the main toolbar, click .
  - On the **Modules** menu, click **Specifications**.

View, Edit, and Save Specification Sheets for Flow Element and Flow Control Valve

1. On the module toolbar, click .
2. On the **Open Entity Specification** dialog box, click in the **Entity number** box and click **Find**.
3. On the **Find Entities** dialog box, click **Find**.

Tip

- A form number already exists for certain of the tags where that form number is specified in the instrument type profile.

4. Under **Search results**, Highlight tag **101-FE -100** and click **OK**.

   The software shows the **Open Entity Specification** dialog box again.

5. In the **Form number** field, type **11**.

   The information is now displayed as shown.

6. Click **OK**.
7. On the **New Specification** dialog box, type a value for the document number as shown and click **OK**.

The **Instrument Specification** window opens for the selected instrument tag.

GENERAL	1	Tag Number		101-FE -100			
	2	Service		Feed from V-8			
	3	Line Number		4"-P-1501-11H			
	4	Line Size	Line Sched.	4	in	80	
	5	Line I.D.		3.826			
	6						
PROCESS CONDITIONS	7	Fluid	State	Lean Feed		Liquid	
	8	Flow Max.	Flow Min.	32	Am³/h	25	Am³/h
	9	Flow Operating		30	Am³/h		
	10	Pressure Max.	Oper.	14	bar-g	13	bar-g
	11	Temperature Max.	Oper.	150	°C	150	°C
	12	Spec. Gravity at Base	Oper.			0.890	
	13	Molecular Mass					
	14	Viscosity at Operating Conditions		0.1	cP		
	15	Quality	Superheat				
	16	Supercomp. Factor					
	17	Base Press.	Base Temp	1	bar	15.5	°C
	18						
	19						
	20						
	21						

8. View the data by scrolling down and across the specification sheet. Note that this spec sheet already contains data (Service, Line Number, process conditions, and so forth) that was already entered or calculated in the previous tasks.

Tip

- Clicking certain fields such as the **State** property (line #7) opens a drop-down list. Selecting a flow or pressure unit of measure field opens a dialog box.

9. Type some additional data in the **Orifice Plate** and **Orifice Flanges** section as shown.

ORIFICE PLATE	22	Type		Square Edge Orifice		Flange Tappings [ISO 5167 (1991)]
	23	Material		PLAIN CARBON STEEL (ANSI B36.19 & B36.10)		
	24	Thickness				
	25	Bore Diameter		2.042	in	
	26	Beta Ratio		0.534		
	27	Drain or Vent Hole			in	
	28	Flow Full Scale		36	Am³/h	
	29	Differential Pressure @ Full scale		2540	mmH2O 4°C	
	30					
ORIFICE FLANGES	31	Size		4"		
	32	Rating		300#		
	33	Type		WELD NECK		
	34	Facing		RF		
	35	Flange Material		By piping		
	36	Gasket Material				
	37	Taps Type				
	38	Taps Size				
	39	Taps Orientation				
	40					
	41					

10. Enter the page and drawing numbers at the bottom of the specification sheet.

11. Click the **Notes** tab and type the text: **1. Orifice flange assembly to be provided by piping**.
12. Click the **Page 1** tab and view the note that you typed in the **Notes** section of the specification sheet.

Tip

- You can also type the note text directly in the **Notes** section.

13. Click to save the specification sheet.
14. Click to close this specification sheet and return the **Specifications Module** window.
15. On the **Open Entity Specification** dialog box, enter the values shown.

Tip

- You do not need to select the form number (1) in this case because it is already defined in the instrument type profile.

16. Click **OK**.

The specification sheet appears as shown.

Instrument Specification - 101-FV -100

Page 1 | Notes

GENERAL	1	Tag No.			101-FV -100				
	2	Service			Feed from V-8				
	3	Line No.			4"-P-1501-11H				
	4	Area Classification			Class 1, Division 2, Groups C&D				
	5	Ambient Temperature:	Min.	Max.					
	6	Allowable Sound Pressure Level dBA			90				
	7	Tightness Requirements			ANSI IV (standard)				
	8	Available Air Supply Pressure:	Min.	Max.	60 psi-g				
	9	Power Failure Position			Close				
	10								
PIPE LINE	11	Line Size and Schedule	Inlet	Outlet	4 in 80		4" 80		
	12	Pipe Material			Carbon steel				
	13	Pipe Insulation			no				
PROCESS CONDITIONS	14	Process Fluid			Lean Feed				
	15	Upstream Condition			Liquid				
	16	Differential Pressure			bar				
	17				Units	@ Max. Flow	@ Norm. Flow	@ Min. Flow	
	18	Flow Rate			Am³/h	32	30	25	
	19	Inlet Pressure			bar-g	14	13	12	
	20	Pressure Drop			bar	3	6	8	
	21	Inlet Temperature			°C	150	150	150	
	22	Inlet Density / Specific Gravity / Molecular Mass			-	0.890	0.890	0.890	
	23	Inlet Compressibility Factor			-				
	24	Inlet Viscosity			cP	0.1	0.1	0.1	
	25	Inlet Specific Heats Ratio			-				
	26	Inlet Vapour Pressure			bar-g	0.9	0.9	0.9	
	27								
CALCULATED RESULTS	28	Flow Coefficient Cv			-	20.4	13.5	9.68	
	29	Travel			%				
	30	Sound Pressure Level @ Maximum Flow:			dBA	65.1	69.4	77.9	
	31	MFR	Model	Fisher	EZ				
	32	Body Type		Single Seat Globe					
	33	Body Size	Trim Size	2 in	By Mfr				
	34	Rated Cv	Characteris.	By Mfr	By Mfr				

POSITIONER	56	MFR	Model	By Mfr	By Mfr
	57	Signal: Inlet	Outlet	3-15 psig	By Mfr
	58	Increase Signal Valve:		Opens	
	59	Cam Characteristic		=%	

17. Click ☒ to close the specification sheet.

Task 4 — Create a Specification Form Data Template

The specification sheets you have viewed so far contain specific data relating to each instrument. If you have a large number of instruments of the same type with common data, you can create a form data template and enter the data in it. You can then copy this data and use it for all instruments of the same type.

Caution

- Copying from a form data template may overwrite fields in the destination specification sheet, if their matching fields are blank. To prevent this from happening you can define – in the Spec Data Dictionary – which specification sheet fields can be overwritten.

Create a Form Data Template for a Flow Transmitter

1. With **Specifications Module** as the active window, on the **Edit** menu, click **Form Data Template Editor**.
2. On the **Select Specification Form** dialog box, type **56** in the **Find form number** box.

   The software highlights form number **56**, **Diff. Pressure Instr. (flow)** as shown.

Form	Form Name	Multiple Tag	Custom	Process Function	Custom Title Block	Notes	Form Entity Ty
51	Smart Diff. Press. Tra			Pressure	No		Instrument
52	Temperature Switch			Temperature	No		Instrument
53	Turbine Flowmeter			Flow	No		Instrument
54	Vibration Monitor			General	No		Instrument
55	Conduct. Transmit./(			General	No		Instrument
56	Diff. Pressure Instr. (			Flow	No		Instrument
57	Diff. Pressure Instr. (l			Level	No		Instrument
58	Smart Diff.Pres.Tran			Flow	No		Instrument
59	Smart Diff.Pres.Tran			Level	No		Instrument
60	Orifice Plate style 2			Flow	No		Instrument
61	Restriction Orifice sty			Flow	No		Instrument
62	Control Valve style 2			Control Valve	No		Instrument
63	Relief Valve style 2			Relief Valve	No		Instrument
64	Butterfly Valve style 2			Control Valve	No		Instrument
65	Diff. Press. Regulato			Control Valve	No		Instrument
66	Emergency Block Va			Control Valve	No		Instrument
67	Special Globe Valve			Control Valve	No		Instrument
68	Ball Valve style 2			Control Valve	No		Instrument
72	Fieldbus Page			General	No		Instrument
73	FieldBus Flow transm		✓	Flow	No		Instrument
74	FieldBus Pressure Tr		✓	Pressure	No		Instrument

3. Click **OK** to open the **Select Form Data Template** dialog box.
4. Type **D/P XMTR TYPE 1** as your form data template.

   Tip

   - You can define more than one data template for each form, for example, ROSEMOUNT XMTR, FOXBORO XMTR, and so forth.

5. Click **OK** to open the form data template.
6. In the **Select Fluid State** pop-up window, select **Liquid** and click **OK**.
7. Enter data as shown.

   Tip

   - You cannot enter data in fields that have a shaded background. These fields are instrument-specific and the values they contain will differ from tag to tag.

GENERAL	1	Tag Number			
	2	Service			
	3	Location	P&ID No.		
	4	Function		D/P FLOW INDICATING TRANSMITTER	
	5	Mounting		2" PIPE	
	6	Area Classification		CLASS 1, DIVISION 2, GROUPS C&D	
	7				
	8				
PROCESS CONDITIONS	9	Fluid			
	10	Pressure Max	Oper.		
	11	Temperature Max.	Oper.	°C	°C
	12	Oper. Spec. Gravity	Oper. Viscosity		cP
	13				
	14				
UNIT	15	Instrument Range Min.	Max.	25 H2O 4°C	625 H2O 4°C
	16	Calibration Range Min.	Max.		
	17	Elevation	Suppression	500% of span	500% of span
	18	Element Type		Piezo Resistive	
	19	Element Material		316ss	
	20	Body Material	Body Rating	316 ss	3000 psi
	21	Process Flanges Material		Hastelloy C	
	22	Wetted O-Rings Material		Viton	
	23	Fill Fluid		Silicon Oil	
	24	Bolts	Housing	Cadmium Plated Carbon steel	Aluminum
	25	Paint		Epoxy	
	26	Process Connection	Electrical Connection	1/2" NPTF	1/2" BSP
	27				
DIAPHRAGM SEAL	28	Process Connection		N/A	
	29	Rating		N/A	
	30	Diaphragm Material		N/A	
	31	Upper Housing Material		N/A	
	32	Lower Housing Material		N/A	
	33	Fill Fluid		N/A	
	34	Capillary Material		N/A	
	35	Capillary Type	Capillary Length	N/A	N/A
	36	Flushing Connection		N/A	
	37				
	38				
OPTIONS	39	Integral Meter		Yes	
	40	Integral Meter Scale		Yes	
	41	Hydrostatic Testing		No	
	42	Cleaning		No	
	43	Calibration		Yes	
	44	Certification		No	
	45				
	46				
	47				
	48				
	49				
PURCHASE	50	Manufacturer			
	51	Model			
	52	Purchase Order Number			
	53	Price	Item Number		
	54	Serial Number			

Notes: 1. Instrument to be supplied with SS Tag and calibration sheet.

No.	By	Date	Revision	INSTRUMENT SPECIFICATION Diff. Pressure Instr. (flow)	INTERGRAPH Sheet of
				Code: 56	Doc. No.: D/P XMTR TYPE 1 Rev.:

8. When done, click .
9. On the main toolbar, click to close the **Form Data Template Editor**.

Create a Specification Data Filter

To prevent undesired overwriting of data, you need to set a form-level specification data filter. You need to do this before you copy data from the active form data template to the current specification.

1. On the module toolbar, click to open the **Spec Data Dictionary**.
2. On the **Select Specification Form** dialog box, type **56** in the **Find form number** box, and then click **OK**.
3. On the module toolbar, click .
4. On the **Field Properties** tab, clear check boxes for the following column headers in the **Template** column:
   - Tag Number
   - Manufacturer
   - Model
   - P&ID No.

   Tip
   - Clearing the check boxes prevents the software from overwriting the values for these properties when copying data from a form template to a specification sheet.
5. On the module toolbar, click .

   The **Browser Format** pop-up window appears.
6. Click to close the **Spec Data Dictionary** and the **Browser Format** dialog box.

Copy Data from a Form Data Template to a Specification

Now you will learn how to automate entry of common data to your specification by using a form data template that you created for the form upon which the current specification is based.

1. On the module toolbar, click .
2. On the **Open Entity Specification** dialog box, do the following:

   a) Under **Entity type**, select **Instrument**.

   b) Under **Entity number**, type **101-FT -100**.

   Tip
   - Use the **Tab** key to move the cursor to the character separators.

   c) Under **Form number**, type **56**.

   d) Click **OK**.

3. On the **New Specification** dialog box, click **OK**.

   The **Instrument Specification** window opens, displaying the specification for Tag 101-FT-100.
4. In the **Select Fluid State** pop-up window, select **Liquid** and click **OK**.
5. On the **Actions** menu, click **Copy from Template**.
6. On the **Select Form Data Template** dialog box, select the new form data template that you created, **D/P XMTR TYPE 1**, and click **OK**.

   The software loads data from the form data template into the specification sheet.

   Tip

   - The software does not overwrite any existing tag-specific data that you specified in the **Specification Data Filter**.
7. Enter additional data on the form as you require.
8. On the module toolbar, click .
9. On the main toolbar, click to close the specification sheet.

Task 5 — Creating and Using a Multi-Tag (See-List) Spec

In the previous tasks, you assigned one single tag to one specification sheet; however, there are often cases when one common specification sheet may serve several tags. Such a spec sheet comprises one common first page and an additional page or pages for each tag. Your task, this time, is to create such a multi-tag spec sheet (called a **format** in SmartPlant Instrumentation) using the forms available in the form library.

Create a New Multi-Tag (See-List) Spec Form

1. On the module toolbar, click to open the **Format Editor**.
2. Click to open the **Select Specification Form** dialog box.
3. Select form number **8** and click **OK**.
4. On the **Select Format** dialog box, type a new format name as shown.

Tip

- A particular form may have more than one format.

5. Click **OK**.

   The **Format Editor** opens, on top of which the software displays the **Edit Headers** pop-up window.

   Tips

   - SmartPlant Instrumentation displays the column headers on the left side of the **Format Editor**, and the field selections on the right side of the window.
   - The **Edit Headers** window displays all the headers that you select in the **Format Editor,** and enables you to modify the sequence of the headers and their labels.
   - In the **Format Editor,** a **magenta** background indicates the currently selected field. A **cyan** background indicates a field that is included in the selection.

6. Assign the following fields — **Tag Number**, **Service**, **Minimum Range**, **Maximum Range**, and **Range Units of Measure** — to the **Multi-Tag List** page of the spec as follows (note that **Tag Number** has already been assigned to the first field):

   a) Drag the **Edit Headers** pop-up window out of the way so that you can see most of the **Format Editor** window.

   b) In the **Format Editor** window, move the cursor over the **cmpnt_serv** (Service) field and note that the cursor changes its shape to .

   c) Double-click in the **field selection** of the **Service** field.

d) In the **Edit Headers** pop-up window, under **Column Header**, the text **General Service** appears. Edit this text to read **Service**.

e) Point the cursor to line #13 and double-click in the leftmost field selection, **calib_range_min**. The field changes its color and appears in the **Edit Headers** pop-up window under **Column Header**.

f) In the **Edit Headers** pop-up window, change the label from **calib_range_min** to **Minimum Range**.

g) Again in line #13, double-click the second field from the left, **calib_range_uom_min**, and in the **Edit Headers** pop-up window, change the label to **Range Units of Measure**.

h) Double-click the fourth field from the left, **calib_range_max**, and in the **Edit Headers** pop-up window, change the label to **Maximum Range**.

i) Using the **Move Up** and **Move Down** commands, rearrange the column headers in the **Edit Headers** pop-up window so that they appear as shown.

Tip

- If you make a mistake, you can cancel the field assignment by highlighting the selection in the **Edit Headers** pop-up window and clicking **Remove**.

7. When done, click on the lower toolbar to generate the column headers that you have defined.
8. At the message prompt, click **OK** to display the column headers in the **Generate Format** pop-up window.
9. In this pop-up window, customize the layout of the second page of your multi-spec sheet format by doing the following:

   a) Drag the **Range Units of Measure** header to the extreme right.

b) Modify the header widths by dragging the border between headers to the right or to the left so that the headers appear as shown.

Tip

- You can also review the layout design by selecting either **Portrait** or **Landscape** in the **Display** group box or save the layout as an external file by clicking **Save As**.

10. Click to save the multi-tag list format.
11. When done, on the main toolbar, click .

You have now successfully created your multi-tag (see-list) spec sheet form, which you will use for your pressure gauges.

Create a Multi-Tag Spec

You are now going to create a multi-tag specification with a master tag.

Note

- If you want the master tag name to appear on the multi-tag specification sheet, on the **File** menu, click **Preferences**. In the tree, navigate to **Specifications** > **General**, and then select the **Show master tag** check box.

1. On the module toolbar, click .
2. On the **Open Entity Specification** dialog box, do the following:

   a) Under **Entity type**, select **Instrument**.

   b) Under **Entity number**, type **101-PI -100**.

   Tip

   - Use the **Tab** key to move the cursor to the character separators.

   c) Under **Form number**, type **8**.

   d) Click **OK**.

   The **New Specification** dialog box opens.

3. Select the option **Create new multi-tag specification**.
4. In the **Document number** field, type **101-21-100001**.
5. From the **Format name** list, select **Demo Pressure Gage See List**.

   The **New Specification** dialog box should appear as shown.

6. Click **OK**.
7. In the **Select Fluid State** pop-up window, select **Liquid** and click **OK**.

8. Type the values in the spec sheet fields as shown.

GENERAL	1	Tag Number		SEE LIST[101-PI -100]			
	2	Service		SEE LIST			
	3	Location					
	4						
	5						
PROCESS CONDITIONS	6	Fluid					
	7	Max. Pressure	Max. Temperature	14	bar-g	150	°C
	8	Oper. Pressure	Oper. Temperature	13	bar-g	150	°C
	9	Pulsation	Vibration				
	10						
	11						
GAUGE	12	Type		Indicator			
	13	Calibration Range Min.	Max.	SEE LIST	SEE LIST	SEE LIST	
	14	Figure Interval		0.1 Bar-g			
	15	Minor Graduation		1%			
	16	Mounting		Direct			
	17	Dial Size		6	in		
	18	Dial Color		Black			
	19	Case Material		Reinforced Plastic			
	20	Ring Construction	Ring Material	Integral		ss	
	21	Blow-Out Protection		Yes			
	22	Lens Material		Glass			
	23	Pressure Element Type		Bourdon			
	24	Pressure Element Material		316SS			
	25	Socket Material		By Vendor			
	26	Connection Size		1/2	in		
	27	Connection Location		Bottom			
	28	Movement Material		ss			
	29	Nominal Accuracy		1%			
	30						
	31						
	32	Type					
	33	Process Connection					
	34	Diaphragm Material					

Tip

- Clicking certain fields such as a temperature unit of measure opens a drop-down list. Selecting a flow or pressure unit of measure field opens a dialog box.

9. Select the units of pressure as shown.

Tips

- The fields that you selected to appear on the following page are marked as **SEE LIST**. The first tag of the list also appears in parentheses on the first line.
- If required, add a note by clicking the **Notes** tab.

10. Click on the module toolbar or click the **Multi-Tag List** tab to display the multi-tag page.
11. Enter the following information:
    a) In the **Service** column, type **Heat exchanger inlet**.
    b) In the **Minimum Range** column, type **0**.
    c) In the **Maximum Range** column, type **20**.
    d) Click the **Range Units of Measure** column, and on the **Select Unit of Measure** dialog box, select **bar** as the unit from the list and **gage** as the scale.

Add a Tag Number to a Multi-Tag Spec

You are now going to assign a new tag number to the multi-tag specification sheet that you just created and then you will enter some data.

1. With the the multi-tag page displayed, on the module toolbar click to associate another tag with this multi-tag specification.

   Tip

   - If you are prompted to save the data, click **Yes**.

   The **Add Tag Number** dialog box opens.
2. Do one of the following:
   - Type **101-PI-101** and click **OK**.
   - Click **Find** to open the **Find Tag** dialog box where you can find the tags that you want to add to this specification. (You can select more than one tag to be associated with this spec by holding down **Ctrl** and clicking the required tags in the search result window.)

   The **New Specification** dialog box opens.
3. Select the option **Add tag to existing multi-tag specification**.
4. In the **Document numbers** field, select **101-21-100001**.

The **New Specification** dialog box should appear as shown.

5. Click **OK**.
6. In the multi-tag specification sheet, enter the specific data and ranges for the additional tag as shown.

7. On the module toolbar, click to return to the first page of the multi-tag specification sheet.
8. Click to save the specification.
9. When done, on the main toolbar, click .

Task 6 — Creating a Customized Specification Sheet

Now you will learn how to modify and customize your specification sheets to suit your needs. Note that you will **not** be using InfoMaker to do this.

Create a Customized Page

You will now customize a new page called **Modified Flow Element**, change and move some headers and their associated data fields. You will switch the location of two fields — **Line Size** and **Line Schedule**.

1. On the module toolbar, click to open the **Page Editor**.
2. On the **Page Editor** module toolbar, click .
3. Under the **Specification Page** column, select **Orifice Plate** and click **OK**.
4. On the module toolbar, click to open the **Edit Columns and Headers** floating toolbar.
5. On the specification page, double-click the **Line Size** header.

   The **Edit Text** dialog box opens.
6. Under **Name**, highlight **Line Size** and type **Line Schedule** over the displayed text.

7. Click **OK**.

Tip

- If necessary, select the header box and drag the right edge to make it wider so that it can accommodate the longer text.

8. Repeat steps 4 through 7 to swap the text of the **Line Size** and **Line Schedule** headers.

   On the **Edit Columns and Headers** toolbar, the actual data fields that are associated with the headers are on the left side of the spec page.

9. Switch the data fields belonging to **Line Size** and **Line Schedule** as follows:

   a) Drag the **Edit Columns and Headers** toolbar to a suitable place on the screen so that you can see most of the screen.

   b) Click the **line_size** data field.

   c) Click to display the **Column Position** pop-up window showing the position of the **line_size** data field.

   d) Make a note of the **X** value and then use the spinner or type a new value to move the **line_size** data field to another place on the form.

   e) Move the **line_uom** data field in the same way.

   f) Select the **line_sched** data field and change its **X** value to the original value for the **line_size** data field so that it takes its former position.

   g) Move the **line_size** data field to where the **line_sched** data field was before you moved it.

   h) Move the **line_uom** data field to the right of the **line_size** data field.

   i) When done, click again to close the **Column Position** pop-up window.

10. Click in the **Edit Columns and Headers** floating toolbar to close it.
11. Click to exit the edit mode.
12. Click on the module toolbar to save the modified page.

13. On the **Save Specification Page** dialog box, click **New** to open the **Page Definition Properties** window, and then type values as shown.

14. Click **OK** to create the new page.
15. Click **OK** to close the **Save Specification Page** dialog box.

Create a Form Containing the Modified Flow Element Page

1. Switch to the **Specifications Module** window.
2. On the module toolbar, click .
3. On the **Form Editor** module toolbar, click .
4. On the **Select Specification Page** dialog box, highlight the **Modified Flow Element** page and click **OK**.
5. Click **Save as Form** .
6. On the **Specification Form Details** dialog box, enter information as shown.

7. Click **OK** on the **Select Specification Page** dialog box to save the new form.

Create and Print a Specification Sheet for Flow Orifices Based on Your Custom Form

1. Switch to the **Specifications Module** window.
2. On the module toolbar, click .
3. On the **Open Entity Specification** dialog box, do the following:
   a) Under **Entity type**, select **Instrument**.
   b) Under **Entity number**, type **101-FE -201**.

   Tip
   - Use the **Tab** key to move the cursor to the character separators.

   c) Under **Form number**, type **71**.
   d) Click **OK**.
4. On the **New Specification** dialog box, do the following:
   a) Select **Create new specification**.
   b) In the **Document Number** text box, type **101-FE-201**.
   c) Click **OK**.
5. In the **Select Fluid State** pop-up window, select **Liquid** and click **OK**.

   The specification sheet opens.
6. Edit the specification sheet data as you require.

   Note the positions of the **Line Schedule** and **Line Size** fields.
7. Click when completed.
8. Click to send a printout of the specification sheet to your printer.
9. When done, on the main toolbar, click to close the specification sheet.

Task 7 — Specification Sheet Revisions

During the life-cycle of a plant, you will probably make changes to specification sheets for any one of a number of reasons; for example, an instrument manufacturer may change some of the characteristics of an instrument, the process conditions may change, or you may just want to add a note explaining the conditions under which the instrument will operate. The revision feature is used to keep track of these changes.

Note

- You can view previous versions of the specification sheets that were revised. Prior to this, you must activate the revision archiving option in the Administration module. With this option, you can save the specification sheets with associated data to the database or to an external file. For complete instructions on how to set up and view revision archives, refer to *SmartPlant Instrumentation User's Guide*, *Specifications Module*.

Add Revision Data to Specification Sheets

1. Open the specification sheet for tag 101-FE-100.
2. Click to open the **Revisions** dialog box.
3. Enter revision data as follows:
   a) Select **Revision method 0,1,2…** and click **New**.

   Tip

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

   b) In the **By** field, type your initials.

   c) In the **Description** field, type **For Design**.
4. If necessary, delete any previous revisions that are no longer needed as follows:
   a) Select the revision you want to delete.

   b) Click **Delete**.

   c) Repeat steps a) and b) for each revision you want to delete.
5. When completed, click **OK**.

   You are returned to the specification sheet.
6. Make changes as desired to the specification sheet and add another revision.

7. Click **OK** to close the dialog box.
8. Click .
9. Repeat the above procedure to create revisions for the specification sheets for tags 101-FT-100 and 101-FV-100, so that on completion, the last revisions for each specification sheet are as shown in the table.

Tag	Revision Method	Last Revision
101-FE -100	P0, P1, P2…	P1
101-FT -100	0, 1, 2…	1
101-FV -100	A, B, C…	A

Make Global Revisions for Specification Sheets

Sometimes, when you make a change in the specification sheet of a particular instrument, you will want to create a formal revision indicating when the change was made, who made the change, and who approved it. In other cases, the specifications of a whole class of instruments may have changed. When this happens, you will probably want to revise a complete set of specification sheets. For this purpose, the global revisions option is available.

1. With the **Specifications Module** window displayed, on the **Tools** menu, click **Global Revisions**.
2. On the **Global Revisions** dialog box, click the **Settings** tab, and from the **Activity** list, select **Upgrade revision**.
3. Enter the required revision numbering criteria to be used as the default in all the newly created revisions as follows:
   a) From the **Initial revision numbering method** list, select method **P0** (preliminary revision numbering). This method will be used for specification sheets that have no previous revisions.
   b) Select the options to change the revision numbering type for all selected specification sheets that have **preliminary revisions** (revision numbering P1, P2, and so forth). To do this, select the **Switch to another numbering method** check box, and from the **Continue numbering using** list, select method **A** (alphabetic revision numbering).

Tip

- Revision numbering for other specification sheets will be based on the existing numbering method, and will be incremented from the previous revision.

4. Complete the information for the other fields on the **Settings** tab (**Revised by**, **Revision Date**, and so forth) as desired, so that it appears similar to that shown.

5. Click the **Specifications** tab.
6. Select the check box **Display only in lowest plant hierarchy level item**.

The software filters the list of tag specification sheets to reflect the selection.

7. Select the following specification sheets (while holding down **Ctrl**):
   - **101-FE-100** (last revision number **P1**).
   - **101-FT-100** (last revision number **1**).
   - **101-FV-100** (last revision number **A**).
   - **101-FE-201** (no last revision).
8. Click **Apply**.

   The software applies revision upgrades to the selected specification sheets. A progress bar indicates the percentage completion of the process.
9. Click **Refresh** to update the data.

   Note the following new revision numbers for the selected specification sheets:
   - **101-FE-100** (last revision number **P1**; new revision number **A**).
   - **101-FT-100** (last revision number **1**; new revision number **2**).
   - **101-FV-100** (last revision number **A**; new revision number **B**).
   - **101-FE-201** (no last revision; new revision number **P0**).
10. Repeat steps 7 – 9 twice for the same four specification sheets.

The following revision numbers should appear for the two revisions:

- **101-FE-100** (last revision number **A**; new revision numbers **B**, **C**).
- **101-FT-100** (last revision number **2**; new revision numbers **3**, **4**).
- **101-FV-100** (last revision number **B**; new revision numbers **C**, **D**).
- **101-FE-201** (last revision number **P0**; new revision numbers **A**, **B**).

Now try some examples for yourself for other single-item specifications using different revision numbering criteria (see the procedure in step 3 above).

Delete Revisions Globally

You can delete revisions for selected specification sheets. Two global deletion options are available — deletion of all revisions or deletion of the last revision for the selection.

1. On the **Tools** menu, click **Global Revisions**.
2. On the **Global Revisions** dialog box, click the **Settings** tab, and from the **Activity** list, select **Delete revisions**.

   Tip

   - This option specifies the deletion of all revisions for the selected specification sheets.
3. Click the **Specifications** tab.
4. Select the check box **Display only in lowest plant hierarchy level item**.

   The software filters the list of tag specification sheets to reflect the selection.
5. Select specification sheet 101-FV-100.
6. Click **Apply**.
7. Click **Yes** when prompted to continue with the deletion.

   A progress bar indicates the percentage completion of the deletion.
8. Click **Refresh** to update the screen.

   Note that the software deletes **all** revisions from specification sheet 101-FV-100 (the **Last Rev.** field is empty).
9. Click the **Settings** tab.
10. Click **Delete last revision** to delete only the last revision for each of the selected specification sheets.
11. Repeat steps 3 – 8 for specification sheet 101-FE-201.

    The value **A** should appear in the **Last Rev.** field.
12. Repeat steps 3 – 8 again for specification sheet 101-FV-201.

Managing Documents

SmartPlant instrumentation enables you to gather documents and maintain them together as a document package. You create and manage packages in the Document Binder module. Each collection of documents is known as a binder package, and there are two types of binder packages that you can create — Specification Binder packages, which can contain specification sheets only, and General Document Binder packages, which can contain various types of documents, including specification sheets. You can group documents according to any criteria you require, for example, instruments associated with a particular vendor, location, plant design phase, and so forth. Note that a given specification sheet may only be assigned to **one** Specification Binder package or to multiple General Document Binder packages, but not to both.

In this part of the tutorial, you will:

- Create a Specification Binder package for control valves that you defined previously.
- Create a form note template, and add a form note based on that template.
- Add a General Note to the Specification Binder package.
- Modify specification sheets in the Specification Binder package and add revisions in the Document Binder module.
- Print documents for the Specification Binder package.
- Create a General Document Binder package for calibration reports.

- To open the Document Binder module, do one of the following:
  - On the main toolbar, click .
  - On the **Modules** menu, click **Document Binder**.

Task 1 — Creating a Specification Binder Package

Create a Specification Binder Package

1. Select the **Binder Packages** icon at the top of the tree view.
2. Right-click the icon and on the shortcut menu, click **New Binder Package**.

   The **Binder Package Properties (New)** dialog box opens.
3. Under **Binder package type**, select **Specification binder**.

4. Enter a binder package name and description as shown.

5. Click **OK**.

   The software automatically creates a document hierarchy for the Specification Binder package with the right pane displaying the column headers for the documents included in the binder package.

Currently, the columns are empty because you have not assigned any documents to the binder package. In the next part of this task, you will assign specification sheets for some of the instrument tags you already created. Later, you will add other documents.

Assign Specification Sheets to the Binder Package

You are now going to assign a number of specification sheets to the instrument tags you created based on Form No. 1 (control valves).

1. If necessary, expand the folder **Demo, Specification Binder Package - CV**.
2. Right-click the **Specification Sheets** folder.
3. On the shortcut menu, click **Assign Specification**.

   The **Find Entities** dialog box opens, where you can search for the desired tags.
4. In the **Form Number** list, select **1** (Control Valve) and click **Find**.

   The dialog box should appear as shown.

Tip

- Depending on which tags you created in the Instrument Index module, the list of tags that appear on your screen may differ from those shown above.

5. From the list, select all the retrieved tags for inclusion in the binder package and click **OK**.

   The list of tags appears under the **Specification Sheets** folder for the binder package.

6. In the **Hierarchy** tree view, click the **Instrument Specification List** and **Specification Sheets** folders and view the information displayed in the right pane of the window.

Note

- The specification list is a cover sheet with instrument tag data for all the specification sheets included in the binder package. You can also configure the layout and the fields to be displayed using the **Preferences** window for the Document Binder module (see the *SmartPlant Instrumentation User's Guide*, *Document Binder* for further details).

Task 2 — Creating Form Notes and General Notes

You can add two types of notes to a binder package. The following examples include both.

- **Form Notes** — These are user notes that you assign for specification sheets based on a particular form. While there is no limit to the number of notes that you can add per form, at least one specification sheet associated with that form must exist in the binder package.
- **General Notes** — General notes provide means of adding user comments that are associated with the whole binder package.

Create a Form Note Template

A form note template provides a convenient means of creating standard text that can be used in a large number of form notes, each of which can then be edited specifically as required.

1. On the **Actions** menu, click **Form Note Templates**.

   The **Form Note Templates** dialog box opens.

2. Click **New** to open the **New Note Template** dialog box.
3. Enter the following template definitions:
   a) From the **Form** list, select Form No. **1** (Control Valve).

      The text **Control Valve, 1** (the form name) appears in the **Template** field.
   b) Under **Description**, type text as shown.

4. Click **OK**.

   You are returned to the **Form Note Templates** dialog box and the template appears in the list.

5. Select the row for the template you just created and click **Edit Text**.

   The **Form Note Templates** dialog box closes and the note editor window opens.
6. In the note pane, type text as shown.

7. On the module toolbar, click to save the note you just typed in the template.
8. Click to close the note editor and return to the **Document Binder** window.

Create a Form Note from the Template

You will now add a form note for the control valves that you have included in the binder package. You will use the template you just created as a basis for the note, which you will edit by adding more text.

1. Right-click the **Form Notes** folder.
2. On the shortcut menu, click **New Form Note**.
3. On the **New Form Note** dialog box, enter a name and description for the form note as follows:

   a) From the **Form** list, select **1** (Control Valve).

   Tip

   - In this case, because **all** the specification sheets that you assigned to the binder package use only form number 1, this is the only form that appears in the list.

   b) From the **Template** list, select **Control Valve, 1** (the template you created for the selected form).

   The dialog box should appear as shown, with the **Name** and **Description** fields automatically filled with the template name and description.

4. Click **OK**.

   The **Note Editor** window opens. The note text that was entered for the template appears by default in the note field.
5. Make changes as shown to the note name, the description, and the note text itself as shown in the example.

6. When done, click .
7. Click to close the note editor and return to the main window.

Add a General Note to the Binder Package

General notes are used for your comments relating to the binder package as a whole, and, unlike form notes, do not require specification sheets to be assigned to the binder package.

1. Right-click the **General Notes** folder.
2. On the shortcut menu, click **New General Note**.

   The **Note Editor** window opens.
3. Type a note name, description, and the note text as shown.

4. When done, click .
5. Click to close the note editor and return to the main window.

Task 3 — Editing and Revising Specification Sheets from the Binder Package

Using the Document Binder module, you can open specification sheets directly for editing, perform a collective update for the documents in a binder package and assign the same revision number to all.

Edit Specification Sheets

In this part of the task, you will edit two of the specification sheets that you assigned to the binder package.

1. In the **Hierarchy** tree view, click the **Specification Sheets** folder.
2. In the right pane, double-click tag number 101-FV-100 to open its specification sheet in the Specifications module.
3. Scroll the sheet to line #18 (**Flow Rate**) and in the **@Norm. Flow** field, type **29** and ensure that the selected units are **Am^3/h**.
4. On the module toolbar, click to save your changes.
5. On the main toolbar, click to close the specification sheet and return to the Document Binder module.
6. Repeat steps 2 – 5 to modify the specification sheet for tag number 101-LV-201 by editing line #2 (Service).
7. Select the **Specification Sheets** folder and view the information displayed in the right pane.

   The **Changed** column in the right pane displays **Yes** for specification sheets that have not been assigned to a binder package before. It also displays **Yes** when a specification sheet has been modified prior to a revision. This indicates that the specification sheet revision number will be incremented the next time you revise the binder package.

Revise a Binder Package for the First Time

A feature of binder packages is that you perform revisions for the binder package as a whole, rather than for the individual specification sheets. For this reason, once you assign a specification sheet to a binder package, you may only perform revisions from the Document Binder module. You are now going to issue a new revision for the binder package.

1. Right-click the binder package folder **Demo, Specification Binder Package - CV**.
2. On the shortcut menu, click **Revisions**.

The **Revisions** dialog box opens.

3. Select revision method **P0, P1, P2…** and click **New**.
4. In the **Description** field, type **First revision in Document Binder**.
5. Click **OK**.

   The software compresses and saves the data to the database.
6. On the **Actions** menu, click **Refresh**, and select the **Specification Sheets** folder.
7. In the right pane, move the scroll bar to view the **Changed** column and verify that the value for each of the specification sheets is **No**. This indicates that the binder package was revised and that no further changes were made to the specification sheets since that revision (for further details, see the *SmartPlant Instrumentation User's Guide*, *Document Binder*, *Notification of Specification Sheet Changes*).
8. Expand the **Specification Sheets** folder and select each tag number in turn. In the right pane, verify that the revision details appear in the appropriate columns.

   Tip

   - The first time you perform a revision on a binder package, the software increments the revision number for all the specification sheets. On subsequent revisions, the software only increments revision numbers for which the **Changed** column value is **Yes**.
9. To view the revisions that were added to the 101-FV-100 specification sheet, open the specification sheet for this instrument in the Specifications module and click .

Make Subsequent Changes and Revisions for a Binder Package

You will now modify the same two specification sheets as in the previous objective and then revise the binder package.

Note

- From now on, you can view any performed changes in the Change Summary Report.

1. Edit the specification sheet for tag number 101-FV-100 once more (making the changes in the same field that you have previously changed) and click .

The following dialog box appears.

2. Select **Mark this item as changed** to set the value of the **Changed** column in the Document Binder module to **Yes** for the specification sheet, indicating that the software will increment its revision number the next time you revise the binder package.
3. On the main toolbar, click to close the specification sheet and return to the Document Binder module.
4. Repeat steps 1 and 2 for tag 101-LV-201.
5. Select the **Specification Sheets** folder and view the values in the **Changed** column in the right pane.

   The display should appear as shown.

Tag Number	Instrument Specification	Form Number	Case	Revision	Changed
101-FV -100	101-21-199901	1		P0	Yes
101-FV -2212	101-21-144434	1		P0	No
101-FV -2213	101-21-144456	1		P0	No
101-LV -201	101-21-943334	1		P0	Yes
101-TV -202	101-21-144454	1		P0	No

   Note that the value in the **Changed** column for the modified specification sheets is now **Yes**.
6. Double-click the **Change Summary Report** folder.

   Tip

   - The **Change Summary Report** compares between a specification sheet and the sheet of the last saved binder package revision. The comparison is performed between values in the specification sheet fields that were modified.

Next you will revise the binder package in order to increment the revision numbers of the specification sheets whose data was changed.

7. Create a new revision in the Document Binder module for your binder package.

Note that the revision numbers in the **Revision** column have changed for the two modified specification sheets, and that all the values in the **Changed** column have been reset to **No**.

Tag Number	Instrument Specification	Form Number	Case	Revision	Changed
101-FV -100	101-21-199901	1		P1	No
101-FV -2212	101-21-144434	1		P0	No
101-FV -2213	101-21-144456	1		P0	No
101-LV -201	101-21-943334	1		P1	No
101-TV -202	101-21-144454	1		P0	No

8. Double-click the **Change Summary Report** folder.

   The Change Summary Report is empty because no changes were made to the specification sheets since the last binder package revision.

Task 4 — Printing Documents for a Specific Revision

You have already viewed the Change Summary Report for a working issue of the binder package. In this task, you will select a particular revision and select several of the documents associate with that revision to print. Your selection will include a Change Summary Report showing details of the changes that you made to the specification sheets since the last binder package revision.

Print Documents for a Particular Revision

1. Right-click the binder package folder **Demo, Specification Binder Package - CV**.
2. On the shortcut menu, click **Print**.

   The **Print** dialog box opens.
3. Make the following selections:
   a) In the **Select revision** group box, select **Formal issue**.
   b) From the **Revision** list, select **P1**.

      This will enable you to display a Change Summary Report for changes that occurred between Revisions P0 and P1.
   c) Under **Print what**, select **Selected documents** to be able to select which of the documents in the binder package you want to print.

The dialog box should now appear as shown.

d) Click **OK**.

4. On the **Select Documents** dialog box, select the documents as shown and click **OK**.

5. In the **Print Preview** window, click ⏩ until the Change Summary Report appears (your values may differ from the ones shown according to the changes that you made).

6. Click to print out the report.

Task 5 — Creating a General Document Binder Package

In this task, you are going to create a new package of the type General Document Binder. In this package, you will add some calculation reports and associate an external document. You can then add a revision to the General Document Binder package.

Create a General Document Binder Package

1. Select the **Binder Packages** icon at the top of the tree view.
2. Right-click the icon and on the shortcut menu, click **New Binder Package**.

   The **Binder Package Properties (New)** dialog box opens.
3. Under **Binder package type**, select **General Document binder**.
4. Enter a binder package name and description as shown.

5. Click **OK**.

   The software automatically creates a document hierarchy for the General Document Binder package with the right pane displaying the column headers for the documents included in the binder package.

Add Calculation Reports to the Document Binder Package

1. On the module toolbar, click to open the **Document Explorer** window.
2. Expand the **Document by Document Type** folder.

3. Scroll down and select **Calculation Reports**.

4. Expand the tree to display the available calculation reports as shown.

Tip

- Depending on the tags on which you performed calculations in the Calculation module, the list of sheets that appear on your screen may differ from those shown above.

5. In the **Document Explorer** window, select document **101-FE -100/CL** and drag it to the **Documents** folder in the **Calculations, Result Sheets** General Document Binder package that you created.
6. Repeat the previous steps to add calculation sheet **101-FV -100/CL** to the **Calculations, Result Sheets** binder package.

Assign External Documents to the Document Binder Package

1. In the **Hierarchy** tree view, expand the binder package folder **Calculations, Result Sheets**.
2. Right-click the **External Documents** folder, and on the shortcut menu, click **Associate External Documents**.
3. On the dialog box that opens, from the **Files of type** list, select **Document Files**.
4. Navigate to a file with extension .txt, .pdf, or .doc.
5. Select the file and click **Open**.

6. Expand the **External Documents** folder and right-click the document that you have just added.
7. On the shortcut menu, click **Open**.

   The document opens in the appropriate program.

Revise the General Document Binder Package

1. Right-click the binder package folder **Calculations, Result Sheets**.
2. On the shortcut menu, click **Revisions**.

   The **Revisions** dialog box opens.
3. Select revision method **P0, P1, P2…** and click **New**.
4. In the **Description** field, type **First revision in General Document Binder**.
5. Click **OK**.
6. On the **Actions** menu, click **Refresh**, and select the **Documents** folder.
7. Expand the **Documents** folder and select the **Calculation Reports** folder.
8. Expand the **Calculation Reports** folder and select each report in turn. In the right pane, verify that the revision details appear in the appropriate columns.
9. If desired, you can print the reports.

Performing Wiring Operations

Now you are going to learn how to create the necessary panels and cables and make the connections for the control loops. The diagram below is an example of a wiring scheme.

Caution

- Make sure that in the Domain Administration chapter of the Tutorial, under Task 3 (Creating the Plant Hierarchy), in the Create a Plant Hierarchy procedure, you cleared the **Do not propagate wire tag names** checkbox.

Task 1 — Overview

Understand Concepts and Presentation of Panel and Cable Wiring

You can create all your panel and cable wiring from the **Domain Explorer**, which opens automatically after starting SmartPlant Instrumentation. If for some reason the **Domain Explorer** is not running, press **F7** to open it. Since it is possible to perform most wiring operations from the **Domain Explorer**, you do not need to start the Wiring module at this time.

Let us now examine how SmartPlant Instrumentation arranges the wiring entities in the **Domain Explorer**.

Panels and their sub-entities are arranged in the tree view of the **Domain Explorer** according to panel location and panel category. Therefore, you can find the same panel and its sub-entities in both the **Panel by Location** and **Panel by Category** folders of the **Domain Explorer**.

Panels are classified according to their category. Click the **Panel by Category** folder to display the various panel category folders. These folders will hold all your panels and their categories. You cannot delete these folders, nor can you create new ones.

Note that each panel category is represented by a particular icon to make it easily recognizable. The panel categories are as shown.

— Junction box

— Plug-and-socket junction box

— Marshaling rack

— Cabinet

— Device panel

— Plug-and-socket device panel

— DCS (Distributed Control System) cabinet

— PLC (Programmable Logical Controller) cabinet

1. In the **Domain Explorer**, click Panels by Category > Device Panels.
2. Select a device panel.
3. Click the device panel to display the existing sub-entities. Currently the next sub-entity is a terminal strip.
4. Click the terminal strip to expand it to display its sub-entities.

Note

- The **Telecom Panels** folder contains a separate group of panels that are specific to Telecom only. These panels are not dealt with in this tutorial.

Panels by Category Hierarchy

Panels can contain a whole range of sub-entities. These sub-entities can be terminal strips, racks, wiring equipment, and so forth. There is a lot of flexibility in creating a panel hierarchy. There is no rigid structure like “panel – terminal strip – terminal” that limits your wiring design. You can create various wiring structures as you require. To create a new panel, right-click the **Panels by Category** folder, click **New** followed by a suitable command on the sub-menu. In our example, we have three types of panels (a junction box, a marshaling rack, and a DCS). We will make use of the following structures as shown.

For the junction box and marshalling rack:

For the DCS panel:

Cable Hierarchy

Cables are arranged in the **Cables** folder. Cross wiring cables are arranged in the **Cross Cables** folder. To create a new cable, right-click the **Cables** folder, click **New** followed by a suitable command on the sub-menu. You can create various sub-entities under cables. Here is one possible structure:

Task 2 — Creating Reference Panels

Reference entities are typical wiring configurations that you can use for rapid creation of various wiring entities. You use the **Reference Explorer** to create and manage all your reference entities.

We strongly recommend that you create as many reference entities in the **Reference Explorer** as possible. Try to create a reference entity for each possible configuration that you may have in your domain. You can then drag this reference entity from the **Reference Explorer** to the **Domain Explorer** and thus create the entity that you require on the fly.

Create Reference Junction Boxes and Terminal Strips

1. Press **F8** to open the **Reference Explorer**.
2. Click the **Panels** folder.
3. Right-click the **Junction Boxes** folder, and on the shortcut menu, click **New** > **Junction Box (Generic)**.
4. Complete the information in the **Junction Box** dialog box as follows.

5. Click **OK**.
6. To view the new junction box in the **Reference Explorer**, expand the **Junction Boxes** folder by clicking it.

7. To add a terminal strip, right-click the new junction box and on the shortcut menu, click **New** > **Terminal Strip**.
8. On the **Strip Terminal Configuration** dialog box, click **New**.
9. On the **Terminals in Pattern** dialog box, enter **3** and click **OK**.

10. Enter information for the new strip configuration as shown.

11. Click **Save** to save the configuration, then click **Create** to create the terminal strip.
12. On the **Terminal Strip Properties** dialog box, click in the **Terminal strip** field and name the terminal strip **16 inst+ind. shield** and then click **OK**.

Edit Terminals

1. In the **Reference Explorer**, expand Panel 16 INSTRUMENTS WITH I/S.

2. Scroll down to the third terminal of the terminal strip (the one labeled '1SH') and select it.
3. Right-click this terminal and then on the shortcut menu, click **Properties**.
4. In the **Terminal** field, change **1SH** to **1sh**.

5. Click **OK** and observe the change in the **Reference Explorer**.
6. Now change the value back to **1SH** again.

Note

- Terminal colors are not shown in the **Domain Explorer** and **Reference Explorer**. You will see them later when you open the **Connection** window.

Create a Reference Marshaling Rack

1. Press **F8** to open the **Reference Explorer**.
2. Click the **Panels** folder.
3. Right-click the **Marshaling Racks** folder, and on the shortcut menu, click **New** > **Marshaling Rack**.
4. On the **Marshaling Rack Properties** dialog box, type **TYPICAL MARSHALING** (you can leave the remaining fields blank).

5. Click **OK** to create this marshaling rack.

Task 3 — Creating Terminal Strips with a Marshaling Rack

You will now create the following terminal strips within this marshaling rack:

- JB SIDE-1 and JB SIDE-2 to accommodate your junction box cables (field side).
- DCS SIDE-AI to accommodate your DCS side cable containing analog input signals (DCS side).
- DCS SIDE-AO to accommodate your DCS side cable containing analog output signals.

Create JB SIDE-1

1. Press **F8** to open the **Reference Explorer**.
2. Click the **Panels** folder.
3. Right-click Marshaling Rack **TYPICAL MARSHALING**, and on the shortcut menu click **New** > **Terminal Strip**.
4. On the **Strip Terminal Configuration** dialog box, select the configuration **2 TERMINALS + SHIELD** (this is the same configuration you used for the junction box).
5. Click **Create** to open the **Terminal Strip Properties** dialog box.
6. Complete the **Terminal Strip Properties** dialog box as shown.

7. Click **OK** to return to the **Strip Terminal Configuration** dialog box.

8. In the **Reference Explorer**, expand the Marshaling Rack **TYPICAL MARSHALING** to view the newly created terminal strip.

Create Strip JB SIDE-2 by Duplicating JB SIDE-1

1. In the **Reference Explorer**, right-click **JB SIDE-1**.
2. On the shortcut menu, click **Duplicate**.
3. On the **Terminal Strip Properties** dialog box, change the terminal strip name to **JB SIDE-2**.

   Tip

   - Observe that the value of the sequence is **2**.
4. Click **OK** to accept the new values and close the dialog box.

Create DCS Side Terminal Strip DCS SIDE - AI

You will now create DCS side terminal strips that correspond to analog input cards and analog output cards.

1. In the **Reference Explorer**, right-click Marshaling Rack **TYPICAL MARSHALING**, and on the shortcut menu click **New** > **Terminal Strip**.

   The **Strip Terminal Configuration** dialog box opens. You will now create a new terminal configuration and a new terminal strip.
2. Click **New** and on the **Terminals in Pattern** dialog box, enter **3**.
3. Click **OK**.

4. On the **Strip Terminal Configuration** dialog box, enter all the values as shown.

5. Click **Save**.
6. Click **Create** to open the **Terminal Strip Properties** dialog box where you create the new terminal strip.
7. On the **Terminal Strip Properties** dialog box, under **Terminal strip**, type **DCS SIDE-AI** and click **OK**.

   Terminal Strip **DCS SIDE-AI** now appears in the **Reference Explorer** under Marshaling Rack **TYPICAL MARSHALING**.
8. Click Terminal Strip **DCS SIDE-AI** to view your strip terminals.

Create DCS Side Terminal Strip DCS SIDE-AO by Duplicating JB SIDE-1

1. In the **Reference Explorer**, expand the hierarchy **Panels** > **Marshaling Racks** > **TYPICAL MARSHALING**.
2. Right-click Terminal Strip **JB SIDE-1** and then on the shortcut menu, click **Duplicate**.
3. On the **Terminal Strip Properties** dialog box, change the terminal strip name to **DCS SIDE-AO**.
4. Click **OK**.

   Terminal Strip **DCS SIDE-AO** now appears in the **Reference Explorer** under Marshaling Rack **TYPICAL MARSHALING**.
5. Expand **DCS SIDE-AO** as we want to have a 4-channel AO terminal strip.

   a) You will now delete the surplus terminals in Terminal Strip **DCS SIDE-AO**.

b) In the **Entities** pane, select Terminal **5+** and while holding down **Shift**, select Terminal **16SH**; this action selects all the terminals between 5+ and 16SH.

c) Right-click the selected terminals and select **Delete** on the shortcut menu.

6. At the following prompt, click **Yes** to confirm the deletion.

Only 12 terminals (from 1+ to 4SH) now remain in Terminal Strip **DCS SIDE-AO**.

Note

- The same result can be achieved by creating a new terminal strip configuration using the **New > Terminal Strip** command on the shortcut menu.

You have now completed the creation of Marshaling Rack **TYPICAL MARSHALING** with its internal terminal strip.

Task 4 — Creating a Reference DCS Panel

You will now create a DCS panel that contains 3 racks. The first one will contain one analog input card and one analog output card.

Create a Reference DCS Panel

1. Press **F8** to open the **Reference Explorer**.
2. Click the **Panels** folder.
3. Right-click the **DCS Panels** folder and then on the shortcut menu, click **New** > **DCS Panel**.
4. On the **Distributed Control System (DCS) Properties** dialog box, under **Panel** type **DCS-1**.

5. Under **Maximum number of racks**, enter **10**.

6. Click **OK**.

Create a Rack in the New Reference DCS Panel

1. In the **Reference Explorer**, right-click the new **DCS-1** cabinet.
2. On the shortcut menu, click **New** > **Rack**.
3. On the **Rack Properties** dialog box, enter the information as shown.

4. Click **Slots**.
5. On the **Batch Slot Creation** dialog box, under **Number of slots**, type **10**.
6. Under **Start from number**, type **1**.
7. Under **Incremented by**, type **1**.

8. Click **OK** to return to the **Rack Properties** dialog box. Note that there are now 10 slots defined in the new rack.

9. Click **OK** to close the **Rack Properties** dialog box.

Create Two Additional Racks in the New Reference DCS Panel

Create two more racks in the **DCS-1** panel by duplicating the new rack that you just created. Name the new racks **File Number 2** and **File Number 3**. Make sure that the new racks have ten slots each. After creating the new racks, the hierarchy of DCS-1 should appear as shown.

Create an Analog Input I/O Card Within the New Reference DCS Panel

After creating the required racks and slots in the new reference DCS panel, you can now create an analog input I/O card.

1. In the **Reference Explorer**, click the **DCS-1** panel to display all its sub-entities.
2. Right-click slot **1** under **File Number 1** and then on the shortcut menu, click **New** > **I/O Card**.
3. On the **New Wiring Equipment** dialog box, type the new I/O card name **8 CHANNEL A/I CARD**.

4. Click **OK**.
5. On the **Wiring Equipment Properties – I/O Card** dialog box, enter values for the I/O card properties as shown.

6. Click the **Control System** tab and for the system I/O type, select **AI**.

7. Click **OK** to accept all the values and close the **Wiring Equipment Properties – I/O Card** dialog box.

Create a Terminal Strip with I/O Channels

1. In the **Reference Explorer**, expand the hierarchy **DCS-1** > **File number 1** > **1** (slot).
2. Right-click I/O Card **8 CHANNEL A/I CARD**, and then on the shortcut menu click **New** > **Terminal Strip**.
3. On the **Strip Terminal Configuration** dialog box, from the **Configuration name** list, select **8 CHANNELS A/I**.
4. Click **Edit**.

5. Change the configuration as shown.

6. Click **Save**.
7. Click **Create** and then define the new terminal strip as shown.

8. Click **OK**.

Create an Analog Output I/O Card and a 4-Channel Terminal Strip Within the New Reference DCS Panel

After creating the required racks and slots in the new reference DCS panel, you can now create an analog input I/O card.

1. In the **Reference Explorer**, expand the panel **DCS-1** to display all its sub-entities.
2. Right-click **Slot 2** under **File Number 1** and then on the shortcut menu, click **New** > **I/O Card**.
3. On the **New Wiring Equipment** dialog box, type the new I/O card name **4 CHANNEL A/O CARD** and click **OK**.
4. On the **Wiring Equipment Properties – I/O Card** dialog box, on the **General** tab, accept the I/O card name.
5. Click the **Control System** tab and from the **System I/O type** list, select **A/O**.
6. Click **OK** to complete the creation of the new I/O card.
7. Right-click the new I/O card **4 CHANNEL A/O CARD** and on the shortcut menu, click **New** > **Terminal Strip**.
8. On the **Strip Terminal Configuration** dialog box, click **New**.
9. On the **Terminals in Pattern** dialog box, enter **3**.
10. Create a configuration with definitions as shown.

11. Click **Save** and then **Create**.
12. On the **Terminal Strip Properties** dialog box, under **Terminal strip**, type **TS 4 CHANNELS** and click **OK**.

Task 5 — Copying the Reference Panels to the Domain Manager

You will now make use of the reference panels in your plant.

Create a Plant Junction Box

1. Make sure that both the **Domain Explorer** and the **Reference Explorer** are open. (Press **F7** to open the **Domain Explorer** and **F8** to open the **Reference Explorer**.)
2. In the **Domain Explorer**, expand the **Panels by Category** folder to display the **Junction Boxes** folder.
3. In the **Reference Explorer**, expand the **Panels** folder.
4. Expand the **Junction Boxes** folder and then click Junction Box **16 INSTRUMENTS WITH I/S**.
5. Drag this junction box to the **Junction Boxes** folder under the **Panels by Category** folder in the **Domain Explorer**.

Note

- When you release the mouse button to drop the junction box in the **Domain Explorer**, the software opens the **Junction Box Properties** dialog box.

6. On the **Junction Box Properties** dialog box, in the **Panel** field, type **101-JB-DCS-001** and click **OK** to close the dialog box.
7. In the **Domain Explorer**, expand Junction Box **101-JB-DCS-001**. Note that it contains the same sub-entities as in the junction box that you created in the **Reference Explorer**.
8. Right-click Terminal Strip **16 inst+ind. shield**, and select **Properties** on the shortcut menu to open the **Terminal Strip Properties** dialog box.
9. In the **Terminal strip** field, type **TS-1** over the existing name and click **OK**.

The renamed Terminal Strip appears in the **Domain Explorer** as shown.

Create a Plant Marshaling Rack

1. Based on the previous procedure, copy the reference marshaling rack **TYPICAL MARSHALING** to the **Marshaling Racks** folder under the **Panels by Category** folder in the **Domain Explorer**.
2. On the **Marshaling Rack Properties** dialog box, rename your plant marshaling rack as **101-MR-DCS-001**.

Rename the Terminal Strips in the Domain Explorer

1. Expand your new Marshaling Rack **101-MR-DCS-001** in the **Domain Explorer** to display its terminal strips.
2. Right-click Terminal Strip **DCS SIDE-AI** and then click **Properties** on the shortcut menu.
3. On the **Terminal Strip Properties** dialog box, under **Terminal strip**, type **TP-1** over the existing name.
4. Click **OK** to save the new definitions and close the screen.

5. Repeat steps 2 through 4 to rename the rest of the terminal strips. Rename them as follows.

Old Reference Terminal Strip Name	New Plant Terminal Strip Name
DCS SIDE-AO	TP-2
JB SIDE-1	FT-1
JB SIDE-2	FT-2

Copy Your DCS from the Reference Explorer to the Domain Explorer

1. Drag **DCS-1** from the **Reference Explorer** to the **DCS Panels** folder in the **Domain Explorer**.
2. Rename DCS-1 to **101-DCS-001**.

Task 6 — Creating Reference Cables

You will now create your reference cables that will be later used in your plant. You will create the following types of cable:

- Single-pair cable
- 16-pair with individual shields
- 8-pair with individual shields
- 4-pair with individual shields

Tip

- It is recommended that you create cables, sets and wires in the **Reference Explorer**. Doing so enables you to create numerous plant cables on the fly.

Create a Reference Single-Pair Cable

Note

- You will find a variety of reference cables in your **Reference Explorer**. These cables are supplied as part of the basic system with SmartPlant Instrumentation.
  As you remember, you have already used a single-pair cable in your instrument type profile to enable the system to automatically create field device objects connected to a single-pair cable. Nevertheless, for tutorial purposes, you will learn how to create reference cables.

The following steps are involved in the creation of a new cable:

- Add cable
- Select cable configuration
- Use configuration to create cable with its associated sets and wires

1. Press **F8** to open the **Reference Explorer**.
2. Right-click the **Cables** folder and then on the shortcut menu, click **New** > **Cable**.
3. On the **Cable Configuration** dialog box, click **New** to create a new cable configuration.
4. On the **New Cable Configuration** dialog box that opens, enter data as shown.

5. Click **OK** to return to the **Cable Configuration** dialog box.
6. Complete the data as shown.

7. When completed, click **Save**.

8. Click **Create** to open the **Cable Properties** dialog box where you can create a new cable.
9. In the **Name** field, type **MY REF PAIR**. In the **Glands** group box, you can define some gland types by clicking ... next to the **End 1** or **End 2** list. You can disregard all the other fields.

10. Click **OK** to close the **Cable Properties** dialog box.
11. View the cable-set-wire hierarchy in the **Cables** folder of the **Reference Explorer**.

Create a 16-Pair Cable

You will now create a multi-pair cable for your junction box.

1. In the **Reference Explorer**, right-click the **Cables** folder and then on the shortcut menu, click **New** > **Cable**.
2. On the **Cable Configuration** dialog box, click **New** to open the **New Cable Configuration** dialog box.
3. Enter **16** as the total number of sets.
4. Select **PAIR W/SHIELD** from the **Default cable set type** list and click **OK** to return to the **Cable Configuration** dialog box.
5. In the **Cable Configuration** field, type **16 PAIR W/SHIELD**.
6. In the **Cable Set** field, type **Pr 1** for the first cable set.

7. Continue to enter the pair numbering (**Pr 2**, **Pr 3**, and so forth) for all 16 cable sets.

   Tip

   - You can use the standard Windows shortcut keys **CTRL+C** and **CTRL+V** to save time when creating the values **Pr 1**, **Pr 2**, and so forth.

8. To enter data for every cable set, do the following:

   a) Select Cable Set **Pr1**.

   b) Under the **Cable set details** group box, enter the wire tag, the wire color, and the polarity.

   c) Under **Copy to all sets of current type**, clear the **Exclude wire tags** check box to copy the wire tag values for each cable set.

   d) Click **Apply**.

   The software copies the wire color, wire type, and polarity data to all the other sets as shown.

9. Click **Save**.
10. Click **Create**.
11. In **Cable Properties** dialog box, in the **Name** field, type **16 PAIRS WITH I/S**.
12. Click **OK** to close the **Cable** dialog box.

Create a Reference 8-Pair Cable and a 4-Pair Cable for Your DCS Analog Input Card Connection

You will now create these two reference cables by populating the newly created 16-pair cable.

1. In the **Cables** folder of the **Reference Explorer**, right-click Cable **16 PAIRS WITH I/S**.
2. Click **Duplicate** on the shortcut menu.
3. On the **Cable Properties** dialog box under **Cable**, type **8-PAIR CABLE**.
4. Leave all the other fields empty and click **OK**.
5. In the **Reference Explorer**, select this newly created Cable **8-PAIR CABLE**.

   Tip

   - You can also see all the existing cable sets in the **Entities** pane of the **Reference Explorer**.

6. You will now delete the surplus cable sets in Cable **8-PAIR CABLE**. Scroll down to pair number 9 (**Pr 9**) and while holding down **Shift**, select pair number 16 (**Pr 16**) to select all the sets between Pr 9 and Pr 16.

7. Right-click the selected sets and on the shortcut menu, click **Delete**.
8. At the prompt, click **Yes** to confirm the deletion.
9. Duplicate **8-PAIR CABLE** and rename the duplicated cable as **4-PAIR CABLE**.
10. Delete the 4 surplus cable sets for **4-PAIR CABLE**.

You have now completed creating all the reference cables required for the tutorial. You will now copy the reference cables you created to the **Cables** folder in the **Domain Explorer**.

Task 7 — Copying Reference Cables to the Plant

Copy Reference Cables with Sets and Wires

1. Make sure that both the **Domain Explorer** and the **Reference Explorer** are open. (Press **F7** to open the **Domain Explorer** and **F8** to open the **Reference Explorer**.)
2. In the **Reference Explorer**, expand the **Cables** folder.
3. Select Cable **16 PAIRS WITH I/S** and drag it to the **Cables** folder in the **Domain Explorer**.
4. On the **Cable** dialog box that opens, enter the cable data as shown.

Tip

- To select the cable color, cable type or gland and glands, click ... to access the supporting table where you create the required item. This item will then become available in the appropriate list.

5. When done, click **OK**.

6. Define Cable **C-101-MR-DCS-001** as follows:
   a) In the **Reference Explorer**, select **8-PAIR CABLE** and drag it to the **Cables** folder in the **Domain Explorer**.
   b) On the **Cable Properties** dialog box, enter the cable data as shown.

   c) Click **OK**.

7. Define C-101-MR-DCS-002 as follows:
   a) In the **Reference Explorer**, select **4-PAIR CABLE** and drag it to the **Cables** folder in the **Domain Explorer**.
   b) On the **Cable Properties** dialog box, enter the cable data as shown.

c) Click **OK**.

You have just completed creating all the cables required for this tutorial.

Task 8 — Making Connections

Caution

- Cables cannot be connected to terminal strips before the connection type is defined. Terminal connection type defines how the wires are to be connected to a terminal strip, which wire is connected to which terminal, and so forth. Read the appropriate section in the Wiring module documentation to learn about the connection type concept and how to create your own connection types. For the purposes of this tutorial, we will use SmartPlant Instrumentation predefined connection types to connect our cables to the panels.

At this stage of the tutorial, you will connect the field wiring (the field instruments) to the junction box. You will then connect the junction box to the field side of the marshaling rack. Subsequently, you will assign I/O tags to DCS cards and connect them to the DCS side of the marshaling rack. Once the sides are connected, you will perform an internal cross wiring within the marshaling rack between the field side terminals and the DCS side terminals.

In this set of procedures, you will make the wiring connections for three control loops (101-F-100, 101-F-102, and 101-P-201). You will make the connections starting from the field devices and working through to the DCS in the control room. In the course of connecting the wires, you will observe how the software handles propagation.

Connect Multi-Pair Cables to Junction Boxes

1. In the **Domain Explorer**, expand the **Panels by Category** folder and then the **Junction Boxes** folder.
2. Right-click **101-JB-DCS-001** and on the shortcut menu, click **Actions** > **Connection**.

3. Make sure that in the **Terminal strip** list, **101-JB-DCS-001, TS-1** is selected.
4. In the **Domain Explorer**, expand the **Cables** folder and select **C-101-JB-DCS-001/1**.
5. Drag Cable **C-101-JB-DCS-001/1** from the **Domain Explorer** to the first terminal at the right side of the terminal strip.

   Tip

   - The cursor changes to a cable icon. When you release the mouse button, the **Cable Connection Definition** dialog box opens.
6. Make sure the **End 1**option button (JB) has been selected (as you are dealing with the junction box end of this cable.)
7. Select **3 in a row** as connection type for Set 1 (Pr 1).
8. If prompted to apply to all sets, click **Yes**, or select the **3 in a row** connection type for each set in turn.
9. Select the **Select all cable sets** check box.

The dialog box should now appear as shown.

Tips

- The connection type determines the way in which the individual wires in a cable set are connected, and how many terminals to leave unconnected between the wires in adjacent sets.
- For a more comprehensive explanation of connection types, see SmartPlant Instrumentation User Guide, Wiring Module.

10. Click **Connect**.

The status bar shows the progress as connections are made. The connections are displayed in the **Connection** window as shown.

Filter the Cables in the Domain Explorer

1. Select the **Cables** folder in the **Domain Explorer**.

   Tip

   - You can keep the **Connection** window open.

2. Click the filter icon to open the **Filter Definition - Cables** dialog box. You will set a filter that allows you to display only the available unconnected field instrument cables.

3. Under **Filter name**, type **Temporary Cable Filter** and then click **Advanced** to open the **Advanced Filter Definition – Cables** dialog box.
4. In the **Connection criteria** group box, select the **Look for connections** check box and then set the filter as shown below.

5. Click **OK**.
6. Click **OK** on the **Filter Definition - Cables** dialog box and to accept the filter condition.
   The **Cables folder** in the **Domain Explorer** is now filtered.
7. Right-click the **Cables** folder and then on the shortcut menu, click **Clear Filter**.

Connecting Device Cables

There are two methods for connecting field device cables, both of which you will learn:

- Individual connection (one by one).
- Batch connection.

Connect a Single Field Device to a Junction Box

1. With the **Connection** window open, select Cable **C-101-FT-100** in the **Cables** folder of the **Domain Explorer**. Drag the **Domain Explorer** to the right side of the window for better viewing.
2. Drag Cable **C-101-FT-100** from the **Domain Explorer** to the left side of Terminal **1+** in the **Connection** window and drop it there.
3. On the **Cable Connection Definition** dialog box, select **End 2** (the JB side) for connection, select the **Select all cable sets** check box and use **3 in a row** as the connection type.

The dialog box should appear as shown.

4. Click **Connect** to make the connection.

   After the connection has been made, observe what happens in the **Connection** window.

Note

- Wires for Cable C-101-FT-100 labeled SPARE before connection of the field device take the names of the tag numbers of the instruments that have been defined as field devices. This is an example of tag number signal.

Connect Field Instrument Cables in Batch Mode

In the previous objective, you have learned how to connect a single cable to the junction box. Now we will show you how to connect multiple device cables in batch mode. Let's connect all the remaining "single" cables to the same junction box.

1. With the **Connection – JB-DCS-001, TS-1** window open, on the **Actions** menu, click **Batch Connection** to open the **Batch Device Cable Connection** window.
2. In the **Domain Explorer**, select the **Cables** folder.
3. In the **Entities** pane of the **Domain Explorer**, select Cable **C-101-FT-102**, hold down **Ctrl** then scroll down the device cable list and select Cables **C-101-FY-100** and **C-101-PT-201**. (Note that you can include any other cables that you have.)

4. Drag the selected device cables from the **Entities** pane of the **Domain Explorer** to the data window of **Batch Device Cable Connections**.
5. For Cable C-101-FT-102, in the **Connection Type** column, select **3 in a row**.
6. Click **Yes** if prompted to apply the connection type to all sets (except for shields).
7. Make sure that the following values appear in the **Starting Terminal** column.

Device Cable	**Starting Terminal**
C-101-FT-102	2+
C-101-FY-100	3+
C-101-PT-201	12+

a) The window should now appear as shown.

8. On the **Actions** menu, click **Connect** to connect the selected device cables in batch mode.
9. Minimize the **Batch Device Cable Connections** window.
10. In the **Connection** window, click on the module toolbar and observe that the tag number signals have been propagated to the right side of the terminals.

11. In the **Connection** window, select the first wire on the left side of 101-FT-100 as shown.

12. Click on the module toolbar to display the other side of the single cable.

    The data display changes as follows.

13. Click again to return to the junction box connections.
14. With the first wire on the left side of 101-FT-100 still selected, click to display the **Point-to-Point Wiring Diagram**.
15. When prompted to preview the report, click **Yes**.

Tip

- If the diagram script is too small and hard to read, increase the font size or zoom the image as described in the following steps.

16. To increase the font size:
    a) On the **View** menu, select **Font Size**.
    b) Use the spinner to set the font size to 7 (seven).
    c) Click **OK**.

Tip

- Try experimenting with different font sizes to see which suits you best.

17. To zoom the image:
    a) On the module toolbar, click .
    b) Click **Custom**, and in the adjacent field, type in 140%.
    c) Click **OK**.

    The image should appear similar to what is shown.

18. Click to print out the diagram if needed.
19. Click to close the Point-to-Point Wiring Diagram.
20. Click to close each **Connection** window.

Connect Field Side Wiring to a Marshaling Rack

1. In the **Domain Explorer**, right-click 101-MR-DCS-001, and on the shortcut menu, click **Actions** > **Connection**.
2. Ensure that in the **Terminal strip** list, Terminal Strip **101-MR-DCS-001, FT-1** is selected.
3. Drag Cable **C-101-JB-DCS-001/1** from the **Cables folder** in the **Domain Explorer** to the first terminal at the left side of the terminal strip.

4. On the **Cable Connection Definition** dialog box, see what happens when you select **End 1** (JB) to select the junction box side for connection.

   Note that all the sets on cable end 1 are already connected, as indicated by (screw heads) in the **Side** column. Let's now see the other side.
5. Click the **End 2** (MARSHALING) option button to select the marshaling rack side.
6. From the **Start at terminal** list, select 1+ and select the **Select all cable sets** check box.
7. Make sure that the connection type is 3 in a row for all sets.
8. Click **Connect**.

   Observe that the tag numbers have been propagated up to the left side of the terminals.

   Tip

   - Open the Point-to-Point Diagram again for the first wire named C-101-FT-100. Notice the progress!
9. Click to close the **Connection** window.

Task 9 — DCS Management

Now that you have completed the required field wiring you are going to work on the DCS side. You will perform I/O assignment, connect the I/O card to the marshaling rack, and connect the input and output analog cables.

Assign I/Os to the DCS

1. In the **Domain Explorer**, expand the **Panels by Category** folder and then the **DCS Panels** folder.
2. Right-click 101-DCS-001, then on the shortcut menu, click **Actions** > **I/O Assignment**.

Tip

- If the **I/O Assignment Type** dialog box opens, select the **I/O assignment** option and click **OK**.

3. In the **Assignment details** pane, make sure that under **I/O termination**, 101-DCS-001, File Number 1, 1, 8 CHANNEL A/I CARD, TS 8 CHANNELS is displayed under **I/O termination**.
4. On the module toolbar, click to open the **I/O Assignment Filter** dialog box, enter the settings below, and click **OK**.

The **I/O Assignment** window should be as shown.

5. Select Tag 101-FT-100 and drag it from the **Tag list** to **Channel 1** in the **Assignment details** pane.
6. On the **Control System Tag Properties** dialog box, over the displayed control system tag name, type **BFI100** and click **OK**.

   The I/O assignment of Tag 101-FT-100 appears as shown.

7. Repeat the above steps to do the following:
   a) Select Tag 101-FT-102, assign it to channel 2, and name its coupled Control System Tag BFI102.
   b) Select Tag 101-PT-201, assign it to channel 7, and name its coupled Control System Tag BPI201.

   The assignments should now appear as follows.

Channel	Address	Control System Tag	Tag Number
1		BFI100	101-FT -100
2		BFI102	101-FT -102
3			
4			
5			
6			
7		BPI201	101-PT -201
8			

8. Now we will do it for the analog output Tag 101-FY-100. To do this, from the **I/O termination** list in the **Assignment details** pane, select **4 CHANNEL A/O CARD, TS 4 CHANNELS**.

   Note that tag number I/O type has changed to AO.
9. Assign Tag 101-FY-100 to channel 1 and name its coupled CS Tag BFY100.
10. Click ☒ to close the **I/O Assignment** window.

You have just completed the I/O assignment of both analog input and output cards. Now you will connect the I/O cards to the marshaling rack, and connect the input and output analog cables.

Connect the Analog Input Cable to the DCS Side

1. In the **Domain Explorer**, right-click 101-DCS-001, and on the shortcut menu, click **Actions** > **Connection**.
2. Make sure that in the **Terminal strip** list, the following terminal and its parent hierarchy is selected:
   101-DCS-001, File Number 1, 1, 8 CHANEL A/I CARD. TS 8 CHANNELS
3. In the **Cables** folder of **Domain Explorer**, select Cable C-101-MR-DCS-001 and drag it to the left side of Terminal 1+v.
4. On the **Cable Connection Definition** dialog box, click the **End 1** (DCS) option button to connect the first end of the cable.
5. Select the **Select all cable sets** check box and select 3 in a row as the connection type for all the sets.
6. Click **Connect** to make the connection.

The **Connection** window now reopens displaying the cable connection to the DCS.

Tip

- The wires for Sets Pr 1, Pr 2, and Pr 7 include tag numbers because you have already made I/O assignments for these tags. Scroll down to Pr 7 and view the wire names.

7. Keep the **Connection** window open.

Connect the Analog Output Cable to the DCS Side

1. In the **Connection** window, from the **Terminal strip** list, select strip 101-DCS-001, File Number 1, 2, 4 CHANEL A/O CARD. TS 4 CHANNELS.
2. In the **Cables** folder of **Domain Explorer**, select cable C-101-MR-DCS-002 and drag it to the left side of Terminal +1OUT.
3. On the **Cable Connection Definition** dialog box, select the **End 1** (DCS) option button to connect the first side of the cable.
4. Click the **Select all cable sets** check box and select **3 in a row** as the connection type for all the sets.

5. Click **Connect** to make the connection.

   The **Connection** window now reopens where you can see the 101-FY-100 signal propagation in accordance with the I/O assignment you made.

6. Click ☒ to close the **Connection** window.

Connect the Analog Input and Output Cables to the Marshaling Rack Side

1. In the **Domain Explorer**, expand the **Marshaling Racks** folder and right-click 101-MR-DCS-001, then on the shortcut menu, click **Actions** > **Connection**.
2. In the **Connection** window, from the **Terminal strip** list, select strip 101-MR-DCS-001, TP-1.
3. In the **Cables** folder of **Domain Explorer**, select Cable C-101-MR-DCS-001 and drag it to the right side of Terminal 1+v.
4. On the **Cable Connection Definition** dialog box, make your selections as shown.

5. Click **Connect** to make the connection.

   The **Connection** window now opens where you can see the signal propagation. Use the horizontal scroll bar to see the wire, set, and cable values on the right side of the terminals.

6. In the **Connection** window, from the **Terminal strip** list, select 101-MR-DCS-001, TP-2.
7. In the **Cables** folder of **Domain Explorer**, select Cable C-101-MR-DCS-002 and drag it to the right side of Terminal 1+.
8. On the **Cable Connection Definition** dialog box, select the **End 2** (MARSHALING) option button to connect the second side of the cable.
9. Click the **Select all cable sets** check box and select **3 in a row** as the connection type for all the sets.
10. Click **Connect**.

The **Connection** window should now display the propagated tag signal as follows. Use the horizontal scroll bar to see the wire, set, and cable values on the right side of the terminals.

11. In the **Connection** window, click Wire 101-FY-100 next to Terminal 1+ and then click to open the Point-to-Point Wiring Diagram.

Note the wiring discontinuity at the marshaling rack, as the required wiring has not yet been completed

12. Print out the Point-to-Point Wiring Diagram if needed and then click ☒ to close it.
13. Click ☒ to close the **Connection** window.

Task 10 — Cross-Wiring the Signals in the Marshaling Rack

As you have seen, there is wiring on both sides of Marshaling Rack 101-MR-DCS 001. On the field (left) side of the marshaling rack, Terminal Strip FT-1 is connected to Junction Box 101-JB-DCS-001. The signals in this wiring segment are 101-FT-100, 101-FT-102, 101-FY-100, and 101-PT-201.
On the DCS (right) side of the marshaling rack, there are two terminal strips — one terminal strip (TP-1) for the analog inputs to the DCS of Tags 101-FT-100, 101-FT-102, and PT-201, and the other terminal strip (TP-2) for the analog output FY-100. The missing link is the cross wiring between these three terminal strips. You will now connect them using the SmartPlant Instrumentation cross-wiring feature.

Cross Wire One Wire Manually

1. In the **Domain Explorer**, expand the **Panels by Category -> Marshaling Racks** hierarchy and then select Marshaling Rack 101-MR-DCS-001.
2. Right-click the selected marshaling rack and then on the shortcut menu, click **Actions** > **Cross Wiring**.

   The **Cross Wiring** window opens.

3. Under **Primary terminal strip,** make sure that terminal strip 101-MR-DCS-001, FT-1 is selected.
4. Under **Primary auto cross-wiring side**, select **Right**.

5. In the **Primary Terminal Strip** pane, select the first row (to the right of Signal 101-FT-100, Terminal 1+, Level 1)

6. On the module toolbar, click to find matching terminals and display them in the **Secondary Terminal Strip** pane.

   The matching terminal is indicated by the ➡ icon.

7. Select the matching terminal in the **Secondary Terminal Strip** pane.

A prompt appears, requesting your confirmation to make the cross-wiring connection.

8. Click **Yes** to confirm the cross wiring.

   Observe how your primary and secondary terminal strips have been connected.

9. Click ☒ to close the **Cross Wiring** window.

Cross-Wire the Rest of the Signals Automatically

1. In the **Domain Explorer**, expand the **Panels by Category -> Marshaling Racks** hierarchy and then select Marshaling Rack 101-MR-DCS-001.
2. Right-click the selected marshaling rack and then on the shortcut menu, click **Actions** > **Cross Wiring**.
3. Under **Primary terminal strip,** make sure that terminal strip 101-MR-DCS-001, FT-1 is selected.
4. Under **Primary auto cross-wiring side**, select **Right**.

5. Activate the primary terminal strip by clicking on the title bar of the **Primary Terminal Strip** pane and click to find the terminals that can be cross-wired and to show the potential cross-wiring connection by a dotted line.

Tip

- The software has found matching terminals 2in, 3-v, 4+v, 5in, and 6-v in Terminal Strip TP-1 belonging to Marshaling Rack 101-MR-DCS-001 and Terminals 1+, 1-, and 1SH in Terminal Strip TP-2 belonging to the same marshaling rack.

6. Click to cross-wire the selected terminals.

All the terminals have been cross-wired and the upper part of the **Primary Terminal Strip** pane should appear as shown.

7. Click ☒ to close the **Cross Wiring** window.
8. In the **Domain Explorer**, right-click Marshaling Rack 101-MR-DCS-001 and then on the shortcut menu, click **Actions** > **Connection**.
9. In the **Connection** window, select one of the wires connected to Terminal 3+ and click to open the Point-to-Point Wiring Diagram.
10. When prompted to preview the report, click **Yes**.

    The diagram is displayed as shown.

As you can see, the wiring discontinuity has disappeared and the wiring continues uninterrupted from the field device to the DCS card.

11. Print out the Point-to-Point Wiring Diagram if needed and then click ☒ to close it.
12. Click ☒ to close the **Connection** window.

Task 11 — Adding a New Instrument to the Existing Wiring

1. In the **Domain Explorer**, expand the **Panels by Category -> Junction Boxes** hierarchy and then select Junction Box 101-JB-DCS-001.
2. Right-click the selected junction box and then on the shortcut menu, click **Actions** > **Connection**.
3. Make sure that Terminal Strip 101-JB-DCS-001, TS-1 is selected in the **Terminal strip** list (there is only one terminal strip on the list).
4. In the **Cables** folder of **Domain Explorer**, select Cable 101-LT-201 and drag it to the right side of Terminal 6.
5. On the **Cable Connection Definition** dialog box, make your selections as follows:
   a) Under **Cable end to connect**, select **End 2**.
   b) Select the connection type as **3 in a row** and select the cable set in the right data window.
6. Click **Connect** to make the connection.
7. After the cable has been connected, open the Point-to-Point Wiring Diagram for the wire connected to Terminal 6+ and view the connection. It should go straight to Terminal Strip FT-1 of the marshaling rack.
8. Print out the Point-to-Point Wiring Diagram if needed and then click ☒ to close it.
9. Click ☒ to close the **Connection** window.
10. Now select 101-DCS-001 from the **DCS Panels** folder in the **Domain Explorer**.
11. Right-click the selected panel, then on the shortcut menu, click **Actions** > **I/O Assignment**.
12. On the module toolbar, click to open the **I/O Assignment Filter** dialog box.
13. Select the first three filtering criteria and click **OK**.
14. In the **I/O Assignment** window, select tag 101-LT-201 and drag it to channel 3 in the right side pane.
15. On the **Control System Tag Properties** dialog box, type **BLI201** and click **OK**.
16. Now close the **I/O Assignment** window and select Marshaling Rack 101-MR-DCS-001.
17. Right-click the selected panel, then on the shortcut menu, click **Actions** > **Connection**.
18. In the **Terminal strip** list, select 101-MR-DCS-001, TP-1.

    Note that your tag wiring has been propagated from the DCS to the marshaling rack terminals.
19. Close the **Connection** window.

20. Right-click the same marshaling rack again, then on the shortcut menu, click **Actions** > **Cross Wiring**.
21. Click to find matching terminals for automatic cross wiring.

    The software finds matching terminals for automatic cross-wiring and marks them with a dotted line. You can view them by scrolling down in the **Primary Terminal Strip** pane.
22. Click to effect the automatic cross wiring.
23. Close the **Cross Wiring** window.

    You have successfully added a new tag to your wiring!

View and Print Wiring and I/O Assignment Reports

The software allows you to generate an array of wiring reports. These include various cable, panel, and I/O reports.

You will now generate the standard system wiring reports to understand their functions.

1. Open the Wiring module open and on the **Reports** menu, select the required report.
2. Where a dialog box for the report opens, enter the required criteria and click **OK**.
3. When prompted to preview the report, click **Yes**.

Task 12 — Create a New Tag with no Associated Profile and then Add a Device Panel and a Cable to the Tag

What if you have forgotten to set a device panel, a cable, and a connection type in the instrument profile?

1. Click on the main toolbar to open the Wiring module.

2. On the module toolbar, click to open the **Device Panels** window.

3. On the module toolbar, click to add a new tag.
4. On the **New Tag Number** dialog box, under **Tag class**, select **Conventional**.
5. Under **Tag number**, type **101-FT-555** as the new tag name and click **OK**.

Tip

- If the **Select Instrument Type** dialog box opens, select **FT – D/P TYPE FLOW TRANSMITTER** and click **OK**.

6. On the **Loop Name** dialog box, accept **101-F-555** and click **OK**.
7. On the **Loop Number Properties** dialog box, accept all the definitions and click **OK**.
8. Click **OK** on the **Tag Number Properties** dialog box.
9. With the **Device Panels** window open, click to filter the display of data.

10. On the **Filter** dialog box, filter the data as follows and then click **OK**.

11. In the **Device Panels** window, select Tag Number **101-FT-555** and click on the module toolbar.

12. On the **Create Device Panel and Cable** dialog box, from the **Reference device panel** list, select **DEFAULT FIELD DEVICE 3-WIRE**.

13. In the **Conventional connections** group box, click **New**.
14. On the **Conventional Connection Properties** dialog box, enter data as shown below, and then click **OK**.

15. On the **Create Device Panel and Cable** dialog box, click **OK**.
16. Click to close the **Device Panels** window.
17. In the **Domain Explorer**, click , and find the new device panel and cable you just created.

Task 13 — Wiring Design that Includes a Barrier

You will now learn how to create a barrier and include it in your wiring design. You will do it by adding a barrier terminal strip in Marshaling Rack 101-MR-DCS-001. To create a barrier you will use the **Apparatus** feature in the Wiring module. **Apparatus** pertains to I/O devices of different kinds. These devices can be used for various purposes, for example, devices that accept wires and signals, barriers, opto-couplers, relays, repeaters, isolators, and so forth.

Adding a new apparatus to your wiring starts with the configuration of a new apparatus strip — setting the number of apparatuses required, defining the apparatus terminal numbering, pattern, signal propagation, and color of the terminals. You create a new apparatus after completing the new apparatus configuration (you do this exactly the same way you create a terminal strip). You can also create a new apparatus based on any existing configuration. You can modify or delete an apparatus configuration. Once a new apparatus has been added, it is displayed in the **Domain Explorer** showing the terminals belonging to the apparatus. Note that you can have regular terminals and apparatus terminals on the same strip.

Define a New Apparatus Configuration

You need to create an apparatus configuration before you can add an apparatus to a selected strip. When configuring an apparatus, you define the apparatus profile (name, description, manufacturer, and model), the number of apparatuses, and the apparatus terminal configuration.

1. In the **Domain Explorer**, expand the **Panels by Category** folder and then the **Marshaling Racks** folder.
2. Right-click Marshaling Rack **101-MR-DCS-001** and on the shortcut menu click **New** > **Apparatus Group**.
3. On the **Apparatus** dialog box, click **New** to create a new apparatus configuration.
4. In the **Configuration name** data field, type **BARRIER**.
5. Enter the description **ANALOG INPUT BARRIER** in the **Description** data field.
6. From the **Manufacturer** list, select **MTL**. If this value is not available from the list, click to add it to the list, then select it from the list.

7. From the **Model** list, select **702**. If this value is not available from the list, click ... to add it to the list, then select it from the list.
8. In the **Number of apparatuses** spin box, select **4** to have a block of four apparatuses to be added by default when adding a new apparatus to a panel.
9. In the **Apparatus position numbering** group box, select **Numbered**.
10. To define the apparatus terminal configuration, do the following in the **Apparatus configuration** group box:
    a) From the **Orientation** list, select **Left / Right** to represent an apparatus that has two sides — an input and an output.
    b) Under **Terminal Color and Numbering**, click in the left text box and type **3+** as the terminal name.
    c) From the left **Terminal Color and Numbering** list, select **Blue** as the color of the left terminal side.
    d) Click in the right **Terminal Color and Numbering** text box and type **1+** as the terminal name.
    e) From the right **Terminal Color and Numbering** list, select **Gray** as the color of the right terminal side.

11. Click **Add Terminal** to add another terminal to this apparatus configuration.
12. Define the second terminal as follows:
    a) From the **Orientation** list, select **Left / Right**.
    b) Click in the left **Terminal Color and Numbering** text box and type **4**– as the terminal name.
    c) From the left **Terminal Color and Numbering** list, select Blue as the color of the left terminal side.
    d) Click in the right **Terminal Color and Numbering** text box and type **2** – as the terminal name.
    e) From the right **Terminal Color and Numbering** list, select **Gray** as the color of the right terminal side.

This completes the new apparatus configuration. Make sure that your dialog box looks like the one shown here before you proceed with the apparatus creation.

13. Click **Save** to save the new apparatus configuration.

Create a New Apparatus and Make All the Connections

Having defined a new apparatus configuration, you will now add a new apparatus to Marshaling Rack 101-MR-DCS-001. When adding a new apparatus, you will also create a new strip where this apparatus will be located. It is also possible to add an apparatus to an existing strip if required.

1. On the **Apparatus** dialog box, click **Create**.
2. On the **Terminal Strip Properties** dialog box, under **Terminal strip**, type **ST-BR-1** and click **OK**.

You have successfully created a new terminal strip with four barriers in Marshaling Rack 101-MR-DCS-001. Expanding this marshaling rack in the **Domain Explorer** will show you the new terminal strip and its four barriers.

Next you will view the connection details of Marshaling Rack 101-MR-DCS-001.

3. In the **Domain Explorer**, right-click Terminal Strip ST-BR-1, then on the shortcut menu, click **Actions** > **Connection**.

   The **Connection** window opens displaying the information as shown.

4. Close the **Connection** window and then in the **Domain Explorer** select Terminal Strip FT-1 under Marshaling Rack 101-MR-DCS-001.
5. Open the **Connection** window and disconnect Pr 12 on the left side of the marshaling rack as follows:

   a) Select the cable set so that all the wires are selected.

b) Click on the **Connection** window toolbar to disconnect the selected wires.

6. Close the **Connection** window.
7. In the **Domain Explorer**, expand Marshaling Rack 101-MR-DCS-001 and right-click Terminal Strip ST-BR1. On the shortcut menu, click **Actions** > **Connection**.
8. Reconnect Pr 12 on the left (marshaling) side of the terminal strip as follows:

   a) In the **Domain Explorer**, expand the **Cables** folder, scroll down to Cable C-101-JB-DCS-001/1 and click it to display its cable sets.

   b) Click Cable Set Pr 12 to display its wires.

   c) Drag the upper 101-PT-201 wire to the first (3+) terminal.

   d) On the **Wire Connection Options** dialog box, accept the **End 2** option and click **OK**.

   e) Drag the lower 101-PT-201 wire to the first (4-) terminal.

   f) On the **Wire Connection Options** dialog box, accept the **End 2** option and click **OK**.

   Now the **Connection** window for 101-MR-DCS-001 should appear as shown.

9. Close the **Connection** window and in the **Domain Explorer** expand 101-DCS-001.

   The signal has already been propagated from Terminal Strip TS 8 CHANNELS in DCS 101-DCS-001 to Terminal Strip FT1 in Marshaling Rack 101-MR-DCS-001.

Now all that remains to complete the signal propagation is to perform cross wiring in the marshaling rack between the appropriate terminals on Strips ST-BR-1 and FT-1.

10. Right-click Marshaling Rack 101-MR-DCS-001 and on the shortcut menu, click **Actions** > **Cross Wiring**.
11. In the **Cross Wiring** window, select Primary Strip 101-MR-DCS-001, ST-BR-1 and Secondary Strip 101-MR-DCS-001, FT-1.
12. Click to find the terminals on Terminal Strip FT-1 that can be cross-wired with Terminal Strip ST-BR-1.
13. Accept the suggested defaults and click to complete the cross wiring.

    At the end of the cross wiring procedure, the **Primary Strip** pane should appear as shown.

This completes the wiring design for Loop P-201. You can now display the Point-to-Point Wiring Diagram and view the new connections.

Task 14 — Wiring Design for Other Loop Types

In this section, you will learn how to create wiring for loops that include local indicators, jumpers, and temperature transmitters and view the related SmartLoops that have been created. We assume that you have already become familiar with the wiring procedures required for this task. Refer to the sections wherever indicated to refresh your memory.

Design the Wiring for Loop T-202

You will now create all the required wiring that constitutes Loop T-202.

1. Press **F8** to open the **Reference Explorer**.
2. Expand the **Panels** folder and create a new device panel named TEMPERATURE TRANSMITTER.
3. Create a strip for this device panel using the following definitions:
   a) On the **Strip Terminal Configuration** dialog box, define a new configuration with 3 terminals in the pattern.
   b) Name the configuration TEMP. TRANSMITTER and make definitions as shown.

   c) Create the terminal strip based on this configuration and name it TT-OUT.
4. Duplicate Terminal Strip TT-OUT to create Terminal Strip TT-IN. (See page 135 for further details of how to duplicate a terminal strip.)

On expanding the terminal strips in the **Reference Explorer**, the terminals should be numbered as shown.

5. Open the Instrument Index module. You can find details of instrument type creation in the Instrument Index module tutorial.
6. Create an instrument type TT (NEW TEMPERATURE TRANSMITTER) and define the profile. Make sure that your definitions on the **General** tab are the same as those shown.

7. On the **Wiring and Control System** tab, select the **Include wiring** and **Control system** check boxes.
8. From the **Reference device panel** list, select **TEMPERATURE TRANSMITTER**.
9. In the **Conventional connections** group box, click **New**, and then complete the **Conventional Connection Properties** dialog box as shown.

10. Click **OK** to close the **Conventional Connection Properties** dialog box.

11. Examine the definitions on the **Wiring and Control Systems** tab of the **Instrument Tag Profile** dialog box and click **OK**.

12. Click **OK** in the **Instrument Types** dialog box.
13. Press **F7** to open the **Domain Explorer**.
14. Expand the hierarchy **New Refinery > Crude Area > Crude Unit 1**.
15. Right click the **Loops** folder and then on the shortcut menu, click **New** > **Loop**.

16. Create a new loop T-202, using the definitions shown.

17. When prompted to create new tag numbers associated with the new loop, click **Yes** and create the following tag numbers:
    - TE-202 - Use the TE-THERMOCOUPLE instrument type. After defining the new tag number, click **New** on the **Tag Number Properties** dialog box to create another tag number.
    - TT-202 - Use the TT-NEW TEMPERATURE TRANSMITTER instrument type.
    - TY-202 - Use the TY-I/P TRANSDUCER instrument type.

    You can find details of loop and tag number creation in the Instrument Index module tutorial.
18. In the **Domain Explorer**, expand the **Panels by Category** folder and then the **Device Panels** folder.
19. Right-click Device Panel 101-TT-202 and then on the shortcut menu, click **Actions** > **Connection**.
20. Make the device panel connections as follows:
    a) In the **Connection** window, select Terminal Strip TT-IN from the **Primary terminal strip** list.
    b) In the **Domain Explorer**, expand the **Cables** folder and drag Cable C-101-TE-202 to the left side of the input terminal strip (TT-IN) using **2 in a row** as connection type. See Task 8 on page 154 to learn how to make connections.

c) Select Terminal Strip TT-OUT and disconnect the Shield wire from Terminal 3SH.

d) Close the **Connection** window.

21. Make the junction box connections as follows:

    a) Select Junction Box 101-JB-DCS-001and open the **Connection** window.

    b) Connect Cable C-101-TT-202 at the left side of Strip TS-1, starting at Terminal 13+ with **3 in a row** as the connection type.

    Observe that the tag number signals have been propagated to the right side of the terminals.

22. Select the wire connected to the right of Terminal 13+ and click to display the connection at the other end of the wire to Marshaling Rack 101-MR-DCS-001, Strip FT-1. (This connection was already made in a previous wiring task in this tutorial.)

23. Close the **Connection** window for each panel.

    You have now completed the wiring up to the marshaling rack. Next, you will define the DCS connections and do the cross wiring in the marshaling rack.

24. In the **Domain Explorer**, expand the **Panels by Category** folder and then the **DCS Panels** folder.

25. Right-click 101-DCS-001 and then on the shortcut menu, click **Actions** > **I/O Assignment**.

26. Assign TT-202 to Terminal Strip TS 8 CHANNELS, Channel 6 and rename the control system tag **BTT202**. (If no tags are displayed on the left area of the screen, click and in the **Criteria** section, select the **Uncoupled Tags** check box.)

27. Display the **Connection** window for Terminal Strip TS 8 CHANNELS and make sure that Cable C-101-MR-DCS-001, Set Pr 6 is connected to Channel 6, Terminals 16+v, 17in, and 18-v. (The signal 101-TT-202 should be propagated to the left in the wires connected to these terminals.)

28. Select the wire connected to the left of Terminal 16+v and click to display the connection at the other end of the wire to Marshaling Rack 101-MR-DCS-001, Strip TP-1. (This connection was already made in a previous wiring task in this tutorial.)

29. Close the **Connection** window for each panel.

30. In the **Domain Explorer**, select Marshaling Rack 101-MR-DCS-001 and cross-wire Terminal Strip FT-1, Terminals 13+ , 13-, and 13SH (right side) and Terminal Strip TP- 1, Terminals 16+v, 17in, and 18-v (left side) respectively. (See Task 10 on page 171 to learn how to do it.).

When done, the cross-wiring connections should be completed as shown.

31. Display the Point-to-Point Wiring Diagram for the loop as follows:
    a) Display the **Connection** window for Device Panel 101-TT-202 and select any one wire.
    b) On the **Reports** menu, select **Loop Point to Point Diagram**.
    c) When prompted to preview the report, click **Yes**.

The **Point to Point Wiring Diagram** print preview for all the wiring in the loop is displayed. Your connections for Tag Number 101-TT-202 should be the same as shown.

Design the Wiring for Loop F-201

You will now create all the required wiring that constitutes Loop F-201. The wiring for this loop is a little more complicated than for the previous example. In particular, you will learn how to create jumpers and how to adjust a signal path definition manually.

You will now create all the required wiring that constitutes Loop T-202.

1. In the **Domain Explorer**, create a new Junction Box 101-JB-PLC-001 by duplicating 101-JB-DCS-001.
2. In the **Domain Explorer**, create a new Multi-Cable C-101-JB-PLC-001 by duplicating Cable C-101-JB-DCS-001/1.
3. In the **Domain Explorer**, create a new PLC and name it **101-PLC-001**.
4. Right-click the new PLC and on the shortcut menu, click **New** > **Rack**.
5. On the **Rack Properties** dialog box, name the rack **R-1** and click **OK**.
6. Right-click the new rack and on the shortcut menu, click **New** > **Slot**.
7. On the **Slot Properties** dialog box, under **Slot**, type **1** and click **OK**.
8. Right-click the rack again, and on the shortcut menu, click **New** > **Wiring Equipment**.

9. On the **New Wiring Equipment** dialog box, select **I/O Card** from the **Category** list and under **Name**, type 1/3 (8 A/I) Card and click **OK**.
10. On the **Wiring Equipment Properties – I/O Card (New)** dialog box, click the **Control System** tab and from the **System I/O type** list, select **AI**.
11. Click **OK** on the **Wiring Equipment Properties – I/O Card (New)** dialog box.
12. In the **Domain Explorer**, right-click the new I/O card and then on the shortcut menu, click **New** > **Terminal Strip**.
13. Create the new terminal strip according to the following definitions:
    a) On the **Strip Terminal Configuration** dialog box, configure the terminal strip to have 8 channels with 3 terminals per channel.
    b) Name the new terminal strip TS 1/3 (8A/I).

    When expanding the terminal strip in the **Domain Explorer**, the terminals should be numbered as shown.

14. In the **Domain Explorer**, right-click the new I/O Card 1/3 (8 A/I) Card, then on the shortcut menu, click **Actions** and click **I/O Assignment**.
15. In the **I/O Assignment** window, assign Tag 101-FT-201 to I/O card 1/3 (8A/I), Channel #1.

    Tip

    - If Tag 101-FT-201 is not on the list in the **Tag List** pane of the **I/O Assignment** window, click and select the **Uncoupled tags** check box.
16. Create a new marshaling rack and name it 101-IR-002.

17. Add a new terminal strip for the field side of 101-IR-002 using the following definitions:
    a) Using the **2 TERMINALS+SHIELD** configuration, configure the strip to have 16 blocks of terminals with 3 terminals per block.
    b) Name the new strip FT-JB-001.
18. Add a new terminal strip for the PLC side of 101-IR-002 using the following definitions:
    a) Configure the terminal strip to have 8 blocks of terminals with 2 terminals per block.
    b) Create the configuration shown.

    c) Name the new terminal strip TP-AI-1/3.
    d) Delete the superfluous terminals that you created by using the 16-pair configuration.

When expanding the terminal strip in the **Domain Explorer**, the terminals should be numbered as shown.

19. Make the following connections in Junction Box 101-JB-PLC-001, Strip TS-1:
    a) Connect Cable C-101-FT-201 starting at Terminal 6+ at the left side of the terminal strip using **3 in a row** as the connection type.
    b) Connect Cable C-101-FI-201 starting at Terminal 7+ at the left side of the terminal strip using **3 in a row** as the connection type.
    c) Connect Multi-Cable C-101-JB-PLC-001 starting at Terminal 1+ at the right side of the terminal strip. Connect all the cable sets on the JB cable side using **3 in a row** as the connection type.
20. Scroll down to view the connections at Terminals 6+ to 7- and note the signal propagation in the wires at the right side of the terminal.
21. Select Cable Set Pr 7 with wires connected on the right side of the strip and disconnect this set.

22. Select the wire connected to the right side of Terminal 6- and drag it to the right side of Terminal 7-.

Note that the signal propagation changes for this wire after reconnection. You will later change the signal definition for this wire manually.

23. Add a jumper between Terminals 6- and 7+ on the right side of the terminal strip as follows:

    a) Click .

    b) On the **Jumper** dialog box, enter the values as shown.

    c) Click **OK**.

The connections should appear as shown.

24. Change the signal definition as follows for the wire you have moved:
    a) Double-click a screw head on Terminal 7- to open the **Terminal Connection** dialog box.
    b) From the **Signal** list on the right side, select **101-FT-201**.

    c) From the **Signal Level** list on the right side, select 2.
    d) From the **Sequence** list on the right side, select 3.
    e) Click **Save**.

The dialog box should appear as shown.

f) Click **Close** on the **Terminal Connection** dialog box.

25. Close the **Connection** window and then reopen it for marshaling rack 101-IR-002, Terminal Strip FT-JB-001.
26. Connect the other end of Cable C-101-JB-PLC-001 to Terminal 1+ at the left side. Connect all the cable sets on the MARSHALING cable side using **3 in a row** as the connection type.
27. On the **Connect Options** dialog box, click **OK**.
28. Scroll down to view the connections at Terminals 6+ to 7SH.

29. When done, close the **Connection** window.
30. In the **Domain Explorer**, create Cable C-101-IR-PLC-003 by duplicating Cable C-101-MR-DCS-001. Rename **End 1 Location** as Instrument Rack and **End 2 Location** as PLC.
31. Add an overall shield to the cable as follows:

    a) Select the cable and add a new cable set to it named **Overall Shield**.

    b) Select the new cable set and add a new wire to it.

    c) In the **Wire** field, type Overall Shield and under **Polarity**, select Overall Shield.

32. Connect Cable C-101-IR-PLC-003 between Marshaling Rack 101-IR-002, Terminal Strip TP-AI 1/3 and PLC 101-PLC-001, Terminal Strip 1/3 (8A/I) with connection definitions as follows:

    a) On the marshaling rack side, connect all the cable sets **except** for OVERALL SHIELD at the right of the terminal strip, using **2 in a row** as the connection type.

    b) On the PLC side, connect all the cable sets **except** for OVERALL SHIELD at the left of the terminal strip, using **3 in a row** as the connection type.

    Note that a signal for Tag 101-FT-201 appears for the first two wires because of the I/O assignment you made in step 15.

    c) Disconnect the Shield wire connected to Terminal 3-v.

33. When done, close the **Connection** window.

34. Select Marshaling Rack 101-IR-002 and cross-wire Terminal Strip FT-JB-001, Terminals 6+ and 6- (right side) and Terminal Strip TP-AI-1/3, Terminals +1 and -1 (left side) respectively.

    When done, the cross-wiring connections should be completed as shown.

This completes the wiring design for Loop FI-201.

This completes all the wiring required for this tutorial. We would like to draw your attention to the fact that some functions of the Wiring module were omitted because the examples in the Tutorial do not use them; nevertheless, you are encouraged to familiarize yourself with the following functions:

- **Connection Type** can be defined by clicking . Simple and complex versions of connection types can be created in a plant related list and reused as required for further automation of the wiring design. For details, see the relevant section of the Wiring module documentation.
- **General Signals and Tag Signals** allow the creation of signals that do not originate from field devices. **General Signal** refers to a signal that may be associated with several tags, for example, a common power supply. **Tag Signal** is a unique signal, for example, the digital signal from a cabinet mounted trip amplifier. It will require this device to be listed in the Instrument Index and its profile will have the wiring checkbox selected, however, without defining a device panel or an instrument cable.

Generating Loop Drawings

This section explains how to generate loop drawings using an external CAD engine such as SmartSketch, AutoCAD, or MicroStation, or using the Enhanced Report Utility that is available with your SmartPlant Instrumentation installation.

Generating Enhanced SmartLoop Reports

You can generate Enhanced SmartLoop reports using the data that you saved to the database when you designed your wiring. To do so, you must first install the Enhanced Report Utility from the SmartPlant Instrumentation CD Browser (click **Enhanced Report Utility Installation**).

Task 1 — Setting Preferences

Set Preferences for Enhanced SmartLoop Reports

1. On the **File** menu, click **Preferences**.
2. In the tree, navigate to **Loop Drawings** > **General**.
3. Under **Default generation method**, select **Enhanced SmartLoop**.

   Tip

   - You can override the default generation method for each loop individually prior to generating the loop drawing or report.
4. In the tree, navigate to **Enhanced Reports** > **Enhanced SmartLoop**.
5. Expand the tree and click **View**.
6. Under **Display options**, select the desired settings.

   Tip

   - Click **Help** to display the Online Help with an explanation of each setting.
7. In the tree, click **Drawing**.
8. Under **Generation method**, click **By loop**.
9. In the tree, click **Title Block**.
10. Ensure that the settings appear as shown.

11. Click **OK**.

Set Preferences for Individual Loops

1. Under **Domain Explorer**, expand the plant hierarchy **New Refinery** > **Crude Area** > **Crude Unit 1** > **Loops**, and select loop **101-F –100**.

   Tip

   - For details of how to locate an item using the **Search** command, see page 64, steps 1 and 2.

2. Right-click the loop, and on the shortcut menu, click **Apply Generation Method** > **Enhanced SmartLoop** > **By Loop**.

   The symbol **E** appears to the left of the loop icon for loop **101-F-100**.

3. Now select and then right-click loop **101-F-102**.
4. On the shortcut menu, click **Apply Generation Method** > **Enhanced SmartLoop** > **By Signal**.

   The symbol **ES** appears to the left of the loop icon for loop **101-F-102**.

5. Now select and then right-click loop **101-F-201**.
6. On the shortcut menu, click **Apply Generation Method** > **CAD**.

   The symbol **C** appears to the left of the loop icon for loop **101-F-201**.

Tip

- You can apply the same indication to several loops by holding down **Shift** or **Ctrl** while selecting the required loops in the **Domain Explorer** and then selecting the appropriate generation type.

The **Domain Explorer** should appear as shown.

Task 2 — Generating an Enhanced SmartLoop Report

1. In the **Domain Explorer**, select and then right-click loop **101-F-100**.
2. On the shortcut menu, click **Reports** > **Generate Loop Drawings**.
3. On the **Generate Loop Drawings** dialog box, make sure that the **Generation method** list displays **Enhanced SmartLoop**, and then click **OK**.
4. When prompted to preview the report, click **Yes**.

   The Enhanced Report Utility opens with the report displayed.

5. On the **Main** toolbar, click the **Zoom In** command and drag over portions of the drawing to zoom in and display more details.

Change the Loop Generation Type to Show Individual Signals

1. In the **Domain Explorer**, select and then right-click loop **101-F-100**.
2. On the shortcut menu, click **Apply Generation Method** > **Enhanced SmartLoop** > **By Signal**.

   The symbol **ES** appears to the left of the loop icon for loop 101-F-100.
3. In the Enhanced Report Utility, on the **View** toolbar, click **Refresh**.

The Enhanced Report Utility refreshes the report and displays each signal separately as shown.

4. Change the generation method for loop 101-F-100 back to **Enhanced SmartLoop** > **By Loop**.

Task 3 — Modifying an Enhanced SmartLoop Drawing by Changing Data Directly from the Enhanced Report Utility

Modify Properties of Entities from the Enhanced Report Utility

1. In the Enhanced Report Utility, select the upper wire that is connected to the device panel for tag **101-FT-100**, as shown.

2. Do one of the following:
   - On the **Actions** toolbar, click .
   - On the **Actions** menu, click **Entity** > **Properties**.
3. On the **Wire Properties** dialog box, from the **Color** list, select **Yellow**.

4. Click **OK** to return to the Enhanced SmartLoop report.

   Tip

   - The wire color value is not updated at this stage.

5. Now position the cursor over cable **C-101-FT -100** and repeat the previous steps to change the cable name to **DEMO CABLE**.

6. To update the modified data values, on the **Actions** toolbar, click **Refresh** .

   Tip

   - If the **Refresh** command is not enabled, click on a blank region of the drawing sheet first.

The Enhanced Report Utility regenerates the loop drawing with the updated data values.

7. Restore the properties that you changed to their original values and refresh the report again.

Access Tag Documents

1. On the currently open drawing page of the Enhanced Report Utility, select the **TAG: 101-FT -100** label.
2. On the **Actions** toolbar, click **Tag Documents** .
3. On the **Documents** window that opens in SmartPlant Instrumentation, click **Specification** to open the flow transmitter specification sheet.
4. Click to print out the specification sheet if needed.
5. Click to close the specification sheet.

Modify the Wiring Connection from the Enhanced Report Utility

Suppose that in loop 101-F-100, you want to move the connections in junction box 101-JB-DCS-001 from terminals 1+ and 1- to terminals 5+ and 5-.

1. In the **Domain Explorer**, select and then right-click loop **101-F-100**.
2. On the shortcut menu, click **Reports** > **Generate Loop Drawings**.
3. On the **Generate Loop Drawings** dialog box, make sure that the **Generation method** list displays **Enhanced SmartLoop**, and then click **OK**.
4. At the print preview prompt, click **Yes**.
5. Click and zoom in on junction box 101-JB-DCS-001.
6. Position the cursor over terminal strip TS-1 of junction box 101-JB-DCS-001 and select the label **TS-1**.
7. On the **Actions** toolbar, click **Connection Window** .
8. In the **Connection** window, select cable 101-FT-100 (on the left side of the terminal strip) and drag it to terminal 5+.

Tip

- The window scrolls automatically if you drag the cable to the lower edge.

9. On the **Cable Connection Definition** dialog box, make your cable connection definitions as shown.

10. Click **Connect** to make the connection.

    The cable moves to its new position (with wires on terminals 5+, 5-, and 5sh) as shown.

11. Click ☒ to close the **Connection** window.
12. Return to the Enhanced Report Utility.
13. On the **Actions** toolbar, click **Refresh** to refresh the report and display the new connections.

Tip

- If the **Refresh** command is not enabled, click on a blank region of the drawing sheet first.

Tip

- If the data fails to update, close and reopen the Enhanced Report Utility.

14. On the **File** menu, click **Save As**, and on the **Save As** dialog box, do the following:
    a) Type a name for the file and navigate to the folder where you want to save the file.
    b) In the **Save as type** list, select AutoCAD (*.dxf) and click **Save**. You can import the file saved in this format into SmartSketch, AutoCAD or MicroStation.
15. Move the cable back to its original position and regenerate the Enhanced SmartLoop Report.
16. Click ☒ to close the Enhanced Report Utility.

Task 4 — Modifying the Appearance of a SmartLoop Drawing by Changing the Enhanced Report Layout Properties

When you generate an Enhanced SmartLoop report, the software automatically calculates the optimal layout based on the number of points in the loop and the selected template size. If the loop changes, by the addition or deletion of elements, the page layout changes accordingly. In this task, you will configure some aspects of the appearance of the report manually by changing the values of some of the report layout properties.

Define an Enhanced Report Layout and Assign the Layout to a Loop

1. In the **Domain Explorer**, select and then right-click loop **101-F-100**.
2. On the shortcut menu, click **Actions** > **Enhanced Report Layouts**.
3. On the **Enhanced Report Layouts** dialog box, click **New**.
4. On the **Layout Properties** dialog box, on the **General** tab, do the following:
   a) In the **Layout** box, type **Template A4 Wide**.
   b) In the **Description** box, type **A4 Landscape**.
   c) Beside **Template file name and path**, click **Browse**, navigate to the location <SmartPlant Instrumentation home folder>\RAD\Template\ and select the file Awide.sma.
   d) Beside **Title block file name and path**, click **Browse**, navigate to the location <SmartPlant Instrumentation home folder>\RAD\Template\Types\loop\ and select the file Loop_Wide.sym.
   e) In both lists, **Wire representation** and **Cross-wire representation**, select **Diagonal**.

   The dialog box should appear as shown.

Tip

- The file paths may be different to the ones shown, depending on the folder where you installed SmartPlant Instrumentation.

5. Click **OK** to close the dialog box and return to the **Enhanced Report Layouts** dialog box.

   The new layout appears as a highlighted row as shown.

6. Click **Assign** to assign the **Template A4 Wide** layout to loop 101-F-100.
7. Click **OK** to close the dialog box.
8. At the prompt to save data changes, click **Yes**.
9. Generate the Enhanced SmartLoop report for loop 101-F-100.
10. On the **Main** toolbar, click the **Fit** command to display the entire report.

The report should appear as shown.

Change the Title Block Location

1. In the **Domain Explorer**, select and then right-click loop **101-F-100**.
2. On the shortcut menu, click **Actions** > **Enhanced Report Layouts**.
3. Select layout **Template A4 Wide** and click **Properties**.
4. On the **Layout Properties** dialog box, click the **Title Block** tab and in the **Vertical** list, select **Top**.

The dialog box should appear as shown.

5. Click **OK** to save the settings and return to the **Enhanced Report Layouts** dialog box.
6. Click **OK** to close the **Enhanced Report Layouts** dialog box and at the prompt to save the changes, click **Yes**.
7. Generate the report for loop 101-F-100.

The report should appear as shown.

Note

- The software calculates the title block location from the origin of the title block symbol file, which is at the bottom left corner for shipped title blocks. In this example, the layout defines the title block location at the top left of the report, thus the title block is positioned as shown. In the following exercise, you will adjust the position of the title block by redefining the working area margins of the report.

Define the Working Area Margins of an Enhanced Report

In the previous exercise, you can see that the title block was positioned above the drawing area. For this example, you can locate the title block within the drawing area by changing the defined working area margins.

1. In the **Domain Explorer**, select and then right-click loop **101-F-100**.
2. On the shortcut menu, click **Actions** > **Enhanced Report Layouts**.
3. Select layout **Template A4 Wide** and click **Properties**.

4. On the **Layout Properties** dialog box, click the **Working Area** tab and do the following:

   a) In the **Units** list, select **in**.

   b) In the **Top** box, type **1.7** and click outside the box.

   The dialog box should appear as shown.

5. Click **OK** to save the settings and return to the **Enhanced Report Layouts** dialog box.
6. Click **OK** to close the **Enhanced Report Layouts** dialog box and at the prompt to save the changes, click **Yes**.
7. Generate the report for loop 101-F-100.

The report should appear as shown.

Tip

- The software starts drawing 1.7 inches from the top of the drawing sheet, as specified by the layout, and that this displacement includes the title block.

8. After generating the report for loop 101-F-100, select layout **Template A4 Wide** and click **Properties**.
9. On the **Layout Properties** dialog box, on both the **Title Block** and **Working Area** tabs, click **Reset**.

Change the Relative Position of the Drawing Area to the Title Block

You are now going to specify a larger template area for the report and modify the position of the drawing area so that it is to the right of the title block.

1. In the **Domain Explorer**, select and then right-click loop **101-F-100**.
2. On the shortcut menu, click **Actions** > **Enhanced Report Layouts**.
3. Select layout **Template A4 Wide** and click **Duplicate**.
4. On the **Layout Properties** dialog box, on the **General** tab, change the following values:
   a) In the **Layout** box, type **Template B Wide**.
   b) In the **Description** box, type a meaningful description for the layout.

c) Beside **Template file name and path**, click **Browse**, navigate to the location <SmartPlant Instrumentation home folder>\RAD\Template\ and select the file Bwide.sma.

5. Click the **Title Block** tab and in the **Drawing area relative to title block** list, select **To right**.

   The dialog box tab should appear as shown.

6. Click **OK** to save the settings and return to the **Enhanced Report Layouts** dialog box.
7. Click **Assign** to assign the **Template B Wide** layout to loop 101-F-100.
8. Click **OK** to close the **Enhanced Report Layouts** dialog box and at the prompt to save the changes, click **Yes**.
9. Generate the report for loop 101-F-100.

   The report should appear as shown.

Tip

- The software calculates the start of the drawing from the right edge of the title block symbol. That is why the drawing items are pushed off to the right of the drawing area.

10. Click ☒ to close the Enhanced Report Utility.

Assign an Enhanced Report Layout to Multiple Loops

Once you have defined a layout, you have the advantage of being able to assign it to a number of loops with similar drawing elements. In this way, you do not have to specify the layout for each loop individually.

1. In the **Domain Explorer**, select and then right-click loop **101-F-102**.
2. On the shortcut menu, click **Actions** > **Enhanced Report Layouts** and note that the **DEFAULT** layout is assigned to this loop.
3. Generate the report for loop 101-F-102, making sure that you select **Enhanced SmartLoop** as the generation method.

The report should appear as shown.

4. Return to the **Domain Explorer** and click the **Loops** folder.
5. In the **List** pane, select loops **101-F-100** and **101-F-102**.
6. Right-click, and on the shortcut menu, click **Actions** > **Enhanced Report Layouts**.
7. At the prompt informing you that the highlighted layout is the one assigned to the first entity in the selection, click **OK**.
8. On the **Enhanced Report Layouts** dialog box, select the layout **Template A4 Wide** and click **Assign**.
9. At the prompt, click **OK**.
10. Click **OK** to close the **Enhanced Report Layouts** dialog box.

    Tip

    - If you select loop 101-F-102 and open the **Enhanced Report Layouts** dialog box, you can see that the layout **Template A4 Wide** is now assigned to this loop.

11. In the **Domain Explorer**, in the **List** pane, select loops **101-F-100** and **101-F-102**.
12. Right-click, and on the shortcut menu, click **Reports** > **Generate Loop Drawings** to generate the reports for both loops.
13. In the Enhanced Report Utility, view each report by selecting it from the **Window** menu.

Loop 101-F-100 report:

Loop 101-F-102 report:

Task 5 — Adding Annotations to an Enhanced SmartLoop Report

During the process of creating control loops, you may want to revise loop parameters, correct faulty wiring, or indicate special operating conditions. You can annotate SmartLoop drawings by including a watermark and by adding redlining.

Include a Watermark

1. In the Enhanced Report Utility, generate the report for loop 101-F-102.
2. On the **Actions** toolbar, click **New SmartText** .
3. On the **SmartText Properties** dialog box, on the **General** tab, enter the following information:
   a) In the **Caption** field, type **Watermark as SmartText**.
   b) Under **Watermark**, select the **Set as watermark** check box.
4. Click the **Text** tab and enter the following information:
   a) Click **Font** to open the **Font** dialog box.
   b) In the **Font style** field, select **Bold**.
   c) In the **Size** field, select **72**.
   d) In the **Color** field, select **Silver**.
5. Click **OK** to return to the **SmartText Properties** dialog box.
6. Under **Alignment**, for both **Vertical** and **Horizontal** fields, select **Center**.
7. In the **Rotation** field, type **30** (degrees).

   The dialog box tab should appear as shown.

8. Click **OK** to return to the drawing sheet.
9. Place the cross-hair cursor at the center of the blank area of the sheet below the loop elements.

   The SmartLoop drawing is displayed with the watermark as shown.

10. Click to select the watermark and drag it on the drawing sheet as you desire.
11. On the **Main** toolbar, click **Save Custom Changes** .
12. On the **Save Custom Changes** dialog box, click **This drawing only**, and then click **OK**.

Tip

- Implementing the **Save Custom Changes** command ensures that whenever you generate the report for the loop, it appears with the watermark and any other annotations that you have added.

Add Redlining

In this objective, you will mark the wires on the loop and add a note to show that the wire colors should be switched.

1. In the Enhanced Report Utility, generate the report for loop 101-F-102.
2. On the **Main** toolbar, click the **Zoom Area** command and drag the zoom area over the field device and adjacent connector on the drawing to display more details.

3. On the **Draw** toolbar, click , move the cross-hair cursor in the drawing area to the position where you want to place the center of the circle, and click the cursor once at that position.
4. Release the mouse button and drag the mouse until the circle is of the desired size, then click to place the circle.

5. Position and size the circle as shown by doing the following:
   a) Click and then click anywhere on the circumference of the circle.
   b) Select the handle at the center of the circle and drag to move the circle.
   c) Select the handle at the edge of the circle and drag to resize the circle.

By following the above steps, try to position and size the circle approximately as shown.

6. Next, you will draw a line next to the circle as follows:
   a) Click to start the drawing line mode and move the cursor to the drawing area.
   b) Click once below and to the right of the circle about two inches away to define the start of the line.
   c) Click again at the lower edge of the circle to define the end of the line.
   d) Click to exit the drawing line mode.

   The line should appear approximately as shown.

7. Finally you will add some text beside the line as follows:
   a) On the **Actions** toolbar, click **New SmartText** .
   b) On the **SmartText Properties** dialog box, in the **Caption** field, type **Swap terminals**.
   c) Click the **Text** tab and click **Font** to open the **Font** dialog box.
   d) In the **Size** field, select **10**.

e) In the **Color** field, select **Red**.

f) Click **OK** to return to the **SmartText Properties** dialog box.

g) Under **Alignment**, for both **Vertical** and **Horizontal** fields, select **Center**.

h) Accept the remaining values and click **OK** to close the **SmartText Properties** dialog box and return to the drawing sheet.

i) Click the cross-hair cursor at the lower right end of the line you drew.

j) Drag the text so that it appears as shown.

8. Click **File** > **Save Custom Changes**.

Attach Redlining Items to a Layout

A powerful feature of the Enhanced Report Utility is the ability to attach redlining items per layout so that the software displays those items for all loops to which that layout is assigned.

1. Click the circle to select it.
2. On the **Actions** toolbar, click **Attach Redlining** .
3. On the **Attach Redlining** dialog box, click **For all drawings assigned to the current layout**.

4. Repeat the above steps for the straight line.
5. On the **Main** toolbar, click **Save Custom Changes** .
6. On the **Save Custom Changes** dialog box, click **All drawings assigned to the current layout**, and then click **OK**.
7. Close the Enhanced Report Utility and regenerate the reports for loops 101-F-100 and 101-F-102.

Note

- The redlining items (circle and line) should appear in both reports; however, the software saves SmartText per loop only. To save text at the layout level, you must add it as a macro caption.

Add a Macro Caption

When you add SmartText to a drawing sheet and save to the database, the software displays the text every time you generate an Enhanced SmartLoop report for the specified loop. In this exercise, you will add text as a macro caption so that the text appears in all loop drawings that belong to the same enhanced report layout.

1. Display the report for loop 101-F-102.
2. Click on a blank area of the drawing sheet, and on the **Actions** toolbar, click **New Macro** .
3. On the **Macro Properties** dialog box, in the **Caption** box, type **Swap terminals (Caption)**.
4. Select the **Display caption only** check box.

5. Click the **Caption Text** tab and do the following to set the values for the text formatting options:
   a) Click **Font** to open the **Font** dialog box.
   b) In the **Size** field, select **10**.
   c) Click **OK** to return to the **Macro Properties** dialog box.
   d) Under **Alignment**, for both **Vertical** and **Horizontal** fields, select **Center**.
   e) Accept the remaining values and click **OK** to close the **Macro Properties** dialog box and return to the drawing sheet.
   f) Click the cross-hair cursor on the drawing sheet at the position where you want to insert the text.
   g) Drag this text immediately below the existing **Swap terminals** label that you added as SmartText.
6. On the **Main** toolbar, click **Save Custom Changes**.
7. On the **Save Custom Changes** dialog box, click **All drawings assigned to the current layout**, and then click **OK**.

8. Close the Enhanced Report Utility and regenerate the reports for loops 101-F-100 and 101-F-102.

   The redlining and text should now appear in the reports as shown.

   Loop 101-F-102:

   Loop 101-F-100:

This completes the set of objectives for Enhanced SmartLoop Reports.

Generating CAD Loop Drawings

Conventional loop drawings are generated from drawing blocks on which SmartPlant Instrumentation data is superimposed. The great advantage of this method is that from a small number of blocks, different combinations of instrument, wiring, and control panel arrangements can be obtained, and many hundreds, or thousands, of loop drawings, each with unique data, may be generated from a single pattern. Before starting the module, make sure that the following files are located in the path SMARTPLANT\INSTRUMENTATION\CAD\BLOCKS.

Block	SmartSketch File	AutoCAD File	MicroStation File
Logo	logo_dem.sym	logo_dem.dwg	logoig.cel
Border	bord_def.sym	bord_def.dwg	border.cel
Orifice Plate	fe_orif.sym	fe_orif.dwg	fe_ori.cel
Control Valve	cv.sym	cv.dwg	cv.cel
Analog Input	ai.sym	ai.dwg	ai.cel
Analog Output	ao.sym	ao.dwg	ao.cel

Note

- The purpose of this tutorial is not to instruct you how to define blocks or cells but rather to demonstrate how to generate loop drawings using a CAD application. At this stage, we have provided the necessary blocks to use in this Tutorial.

Task 1 — Setting Preferences

Set Preferences for your CAD Application

9. On the **File** menu, click **Preferences**.
10. In the tree, navigate to **Loop Drawings** > **General**.
11. Under **Default generation method**, select one of the following CAD applications:
    - **SmartSketch**
    - **AutoCAD**
    - **MicroStation**
12. Set other options on this page as appropriate for the specified application.
13. In the tree, navigate to **Loop Drawings** > **CAD File Locations**.

14. Set the folder locations for each group of files as appropriate for the specified CAD application as shown in the following example.

Important

- You must specify the folder locations for the software be able to create drawing blocks and generate loop drawings using the CAD application that you selected.
- On the **Preferences**, **General** page ensure that you have defined a value for **Temporary folder path**.

15. Click **OK**.

Note

- If the Loop Drawings module is already open when you set the preferences, you must close the module and then re-open it for the settings to take effect.

Task 2 — Defining Block Types and Drawing Blocks

Classification of drawing blocks according to block type is useful for filtering the blocks. It is recommended that each block type contain drawing blocks that serve the same general function. This way, it will be easy to find a particular drawing block when you want to use it.

The CAD session is based on SmartSketch. The procedures for AutoCAD and MicroStation are very similar.

Define Block Types

1. Open the Loop Drawings module by doing one of the following:
   - On the main toolbar, click .
   - On the **Modules** menu, click **Loop Drawings**.
2. Do one of the following:
   - On the module toolbar, click .
   - On the **Tables** menu, click **Block Types**.
3. On the **Block Types** dialog box, click **New**.
4. On the **Block Type Properties** dialog box, type data as shown.

5. Click **OK**.
6. On the **Block Types** dialog box, click **OK** to save the data.

Define Blocks

Within each block type, you now need to define the drawing blocks and select for each block the file that contains the required drawing.

1. Do one of the following:
   - On the module toolbar, click .
   - On the **Tables** menu, click **Blocks**.
2. On the **Blocks** dialog box, from the **Block type** list, select **INSTRUMENTS**.
3. Click **New**.
4. On the **Block Properties** dialog box, enter data as shown.

Tip

- If you need to search for the appropriate file, click **Browse**.

Tip

- If you are using AutoCAD or MicroStation as your CAD application, blocks of the appropriate file type should be available for selection.

5. Click **OK** to display the new block on the **Blocks** dialog box.

6. Repeat steps 3 – 5 so that the blocks in the **INSTRUMENTS** block type are as shown in the following table.

SmartSketch File	Block Name	Description
fe_orif.sym	Orifice Plate SmartSketch	FLOW ELEMENT
cv.sym	Control Valve SmartSketch	CONTROL VALVE
ai.sym	Analog Input SmartSketch	INPUT LOOP
ao.sym	Analog Output SmartSketch	OUTPUT LOOP

7. Click **OK** to save data and close the dialog box.

Task 3 — Assigning Blocks to Instrument Types

Assign Blocks to Instrument Types

When you assign a block to a specific instrument type, the software associates this block automatically with any instrument tags of that instrument type. Such blocks appear green in the **Loop Explorer**. Only one block may be assigned to each instrument type.

1. Do one of the following:
   - Click .
   - On the **Actions** menu, click **Block - Instrument Type Assignment**.
2. On the **Block – Instrument Type Assignment** dialog box, scroll down the list of instrument types and select **FE – D/P TYPE FLOW ELEMENT** as shown.

Tip

- This selection will only create associations with those instruments of type FE which are described as D/P TYPE FLOW ELEMENT, and not with other FE descriptions.

3. Click **Assign**.
4. On the **Assign Block To Instrument Type** dialog box, from the **Block type** list, select **INSTRUMENTS**, and then in the data window select **Orifice Plate SmartSketch**.

5. Click **OK** to assign this block to the FE D/P TYPE FLOW ELEMENT instrument type. The **Block – Instrument Type Assignment** dialog box should appear as shown.

6. Make other block – instrument type assignments so that all the assignments are as shown in the following table.

Instrument Type	Description	Block Name
FE	D/P TYPE FLOW ELEMENT	Orifice Plate SmartSketch
FT	D/P TYPE FLOW TRANSMITTTER	Analog Input SmartSketch
FV	CONTROL VALVE	Control Valve SmartSketch
FY	I/P TRANSDUCER	Analog Output SmartSketch

7. When done, click **Close**.

Task 4 — Defining the General Blocks

Using the following procedure, you can define your own border and logo using SmartSketch blocks.

Define New Border and Logo Blocks

1. Do one of the following:
   - Click .
   - On the **Tables** menu, click **Blocks**.

2. Define a new border block as follows:
   a) On the **Blocks** dialog box, from the **Block type** list, select **BORDER**.
   b) Click **New**.
   c) On the **Block Properties** dialog box, name the block **Demo Border SmartSketch**.
   d) Beside **File name**, click **Browse** and navigate to the file **bord_def.sym**.
   e) Click **View** to view the block in SmartSketch.
   f) Click **OK** to close the **Block Properties** dialog box.
3. Define a new logo block as follows:
   a) On the **Blocks** dialog box, from the **Block type** list, select **LOGO**.
   b) Click **New**.
   c) On the **Block Properties** dialog box, name the block **Demo Logo SmartSketch**.
   d) Beside **File name**, click **Browse** and navigate to the file **logo_dem.sym**.
   e) Click **View** to view the block in SmartSketch.
   f) Click **OK** to close the **Block Properties** dialog box.
4. Click **OK** to close the **Blocks** dialog box.

Assign Your New Border and Logo Blocks as Default

1. On the **Actions** menu, click **Default General Blocks**.
2. From the **Border block** list, select **Demo Border SmartSketch**.
3. From the **Logo block** list, select **Demo Logo SmartSketch**.
4. Click **OK**.

Note

- Unless you redefine the default border, the border you select in this dialog box will be used from now on for all the loop drawings you generate.

Task 5 — Editing the Title Block for a Specific Loop

Define Drawing Title Block Information

1. In the **Domain Explorer**, select loop **101-F-100**.
2. Right-click the loop and on the shortcut menu, click **Actions** > **Loop Drawing List**.
3. On the **Loop Drawing List** dialog box, click **References** to view the drawing reference used.

   Tip

   - On the **Document References** dialog box, the P&ID drawing reference appears automatically.

4. Click **New** to add another document reference.
5. On the **Document Reference Properties** dialog box, type data as shown.

6. Click **OK** to close the dialog box and return to the **Document References** dialog box.

7. Click **OK** to close the **Document References** dialog box.

Edit Approvals

1. On the **Loop Drawing List** dialog box, click **Approvals**.
2. On the **Approvals** dialog box, type data similar to that shown.

3. When done, click **OK**.

Edit Revisions

1. On the **Loop Drawing List** dialog box, click **Revisions**.
2. On the **Revisions** dialog box, add a revision as follows:
   a) In the **Revision method** list, select **0, 1, 2 …** and click **New**.
   b) Enter data similar to that shown.

3. Click **OK** to close the **Revisions** dialog box.

Task 6 — Understanding Macros

Macros are required to extract specific data from a given loop or instrument.

Get Acquainted with the Macro Language

1. Do one of the following to open the **Macro Definitions** dialog box:
   - On the module toolbar, click .
   - On the **Tables** menu, click **Macro Definitions**.

2. Navigate to the instrument tag macros as follows:.
   a) In the **Macro group** list, select **INSTRUMENT**.
   b) In the **Macro type** list, select **INSTRUMENT**.
   c) In the **Macro attribute** list, select **NAME**.
3. In the **Macro definitions** data window, observe the macro names and description of each macro to learn what they mean.
4. Browse through other macros to see how they are organized and what the macro names mean.
5. Click **Close** to close the **Macro Definitions** dialog box.

Note

- Wiring macros include group level and wire sequence identifiers. For a fuller explanation of macros, see the *SmartPlant Instrumentation User's Guide*, *Loop Drawings Module*, *SmartPlant Instrumentation Macros*.

Task 7 — String Manipulation for Loop Drawings

To perform string manipulation on the text to be displayed by the macro, the software enables you to define functions which you can attach to the macros and then extract the necessary string attributes.

Define User Functions

1. On the **Tables** menu, click **User-Defined Macro Functions**.
2. On the **User-Defined Macro Functions** dialog box, click **New** to open the **User-Defined Macro Function Properties** dialog box.
3. Do the following:
   a) In the **Macro function** field, type **S11**.
   b) In the **Abbreviation** field, type **S11**.
   c) In the **Description** field, type **Substring takes 1 character from 1st pos**.
4. Click **Insert** to open the **Standard Macro Functions** dialog box.
5. Select the following standard function as shown.

6. Click **OK** to return to the **User-Defined Macro Function Properties** dialog box.
7. In the **Parameters** section, define the parameters as follows:
   a) In the **Start position** field, use the spin box to select the value **1**.
   b) In the **Length** field, use the spin box to select the value **1**.

   The dialog box should now appear as shown.

8. Click **OK**.
9. Add another user-defined function **S13**, making the following definitions on the **User-Defined Macro Function Properties** dialog box:
   a) In the **Macro function** field, type **S13**.
   b) In the **Abbreviation** field, type **S13**.
   c) In the **Description** field, type **First three characters**.
   d) Click **Insert**.
   e) On the **Standard Macro Functions** dialog box, select the **Substr** function.
   f) Click **OK** to return to the **User-Defined Macro Function Properties** dialog box.

g) In the **Start position** field, enter **1**.

h) In the **Length** field, enter **3**.

The **User-Defined Macro Functions** dialog box should appear as shown.

10. When done, click **OK** to close the dialog box.

For further details of how to use these functions in your drawing macros, see the *SmartPlant Instrumentation User's Guide*, *Loop Drawings Module*, *Creating User-Defined Macro Functions*.

Task 8 — Generating CAD Loop Drawings

Select an Element (Loop, Instrument, or Block) and View Drawings

1. Do one of the following:
   - On the module toolbar, click .
   - On the **Actions** menu, click **Loop Explorer**.
2. In the **Loop Explorer**, expand loop 101-F-100 by double-clicking its icon.

   The software displays all the tag numbers associated with this loop.

3. Double-click tag 101-FE-100 to display the drawing blocks associated with it.

The block is displayed in green, indicating that it was associated with the tag through the instrument type.

4. Right-click the block, and on the shortcut menu, click **Reports** > **View Block in CAD Application**.

   The drawing block for the selected tag is displayed in your CAD application, including the macros. (The example shows how the drawing appears in SmartSketch – what you see may be slightly different depending on the CAD application you are using).

5. Expand each of the tags in loop 101-F-100 and observe all the blocks that you associated previously.

   Tip

   - The view obtained is in accordance with the hierarchy level selected. If you select a loop, the software displays all the drawing blocks for that loop. If you select a tag number, the software displays the blocks attached to that tag number; selection of a block displays that block only. When viewing drawing blocks before generation, the software displays the blocks with macros only.

6. In the **Loop Explorer**, select loop 101-F-100.
7. Right-click the loop, and on the shortcut menu, click **Reports** > **View in CAD Application** to view the drawing blocks attached to the loop.

Associate Blocks with Tag Numbers Manually

The block assignments made by the initialization process were created automatically. You can achieve the same goal by associating blocks with tag numbers manually.

1. In the **Loop Explorer**, double-click loop 101-F-100 to display the tag numbers.
2. Double-click tag 101-FT-100 to display its associated block.

3. Right-click the **Analog Input SmartSketch** block.
4. On the shortcut menu, click **Actions** > **Dissociate Blocks from Tag Number** to dissociate the block.
5. At the prompt, click **Yes**.
6. Right-click the **101-FT-100** tag.
7. On the shortcut menu, click **Actions** > **Associate Blocks with Tag Numbers**.
8. On the **Associate Blocks with Tag Numbers** dialog box, select block **Analog Input SmartSketch** as shown.

9. Click **OK**.

In the **Loop Explorer**, you now have one red block associated with tag 101-FT-100. Blocks are shown in red to indicate that they have been associated manually.

10. Right-click the **Analog Input SmartSketch** block attached to tag 101-FT-100.
11. On the shortcut menu, click **Reports** > **View Block in CAD Application** to display the block drawing in your CAD application.
12. Check that the wiring macros show the correct group level and sequence for the wires, as in the following example.

Change the Block Association Method from “Block – Instrument Assignment” to “Manual Association”

1. In the **Loop Explorer**, double-click tag 101-FE-100 to display the associated block.

   The block is displayed in green because you associated this block with the tag number using the block-instrument type assignment.

2. Right-click the tag, and on the shortcut menu, click **Actions** > **Change Block Association Method**.
3. At the prompt, click **Yes**.

   The software changes the association method to manual. In the **Loop Explorer** the block is now displayed in red.

4. Repeat the same procedure for tags 101-FY-100 and 101-FV-100.

Generate a CAD Loop Drawing

1. In the **Loop Explorer**, select and then right-click loop 101-F-100.
2. On the shortcut menu, click **Reports** > **Generate Loop Drawings**.
3. On the **Generate Loop Drawings** dialog box, do the following:
   a) In the **Generation method** list, select **CAD**.
   b) Select the **Save drawing automatically** check box, and click **Browse** to navigate to the desired output folder.
4. Click **OK** to generate the loop drawing.
5. Inspect the results and verify that the loop drawing output reflects the wiring data as you executed in the session on the Wiring module.

Add a New Revision to a Loop Drawing and Generate the Loop

1. In the **Loop Explorer**, select and then right-click loop 101-F-100.
2. On the shortcut menu, click **Actions** > **Loop Drawing List**.
3. Click **Revisions**.
4. On the **Revisions** dialog box, click **New** to add a new a revision for the loop.
5. Regenerate the loop and view the new revision data in the loop drawing.

Note

- The following screen captures show how the loop drawing for loop 101-F-100 appears before and after generation. Note how the macros are replaced by SmartPlant Instrumentation data after the generation.

Loop Drawing with Macros Before Generation

Loop Drawing After Generation

Generating Hook-Up Drawings

You can create, edit, and view the physical installation detail drawings of instruments and the fittings that are required to connect them to the piping. You can also list the parts and generate a Bill of Material.

Hook-Up drawings instruct the contractor how to install the instruments and what type of fittings and accessories should be used. You use the Hook-Ups module to perform the following operations:

- Create the required hook-up types (categories)
- Create appropriate hook-ups within hook-up types
- Associate actual CAD drawings with SmartPlant Instrumentation hook-ups
- Build a fittings library and associate the fittings with the appropriate hook-ups
- Generate a Bill of Material

Note

- Before starting the module, make sure the hook-up drawings that you require are located in the path <SmartPlant Instrumentation home folder>\Hook-up\Standard.

- To open the Hook-Ups module, do one of the following:
  - On the main toolbar, click .
  - On the **Module** menu, click **Hook-Ups**.

Task 1 — Setting Preferences

Set Preferences for your CAD Application

1. On the **File** menu, click **Preferences**.
2. In the tree, navigate to **Hook_ups** > **File Locations**.
3. Set the folder locations for each group of files as appropriate for the specified CAD application as shown in the following example.

Important

- You must specify the folder locations for the software be able to create drawing blocks and generate hook-up drawings using the CAD application that you selected.
- On the **Preferences**, **General** page ensure that you have defined a value for **Temporary folder path**.

4. Click **OK**.

Note

- If the Hook-Ups module is already open when you set the preferences, you must close the module and then re-open it for the settings to take effect.

Task 2 — Creating Hook-Up Items

Create a Library

1. On the **Tables** menu, click **Item Libraries**.
2. On the **Item Libraries** dialog box, click **New** to open the **Item Library Properties** dialog box.
3. In the **Item library** field, type **DEMO HOOK-UP LIBRARY**.

4. In the **Description** field, type **DEMO HOOK-UP ITEMS LIBRARY** and then click **OK**.
5. On the **Item Libraries** dialog box, select the library you just created, select the **Set as current item library** check box, and click **OK**.

Add Items to a Library

1. On the module toolbar, click .
2. On the **Item List for Current Item Library** dialog box, click **New**.
3. On the **Item Properties** dialog box, enter data as shown.

Tip

- To add manufacturers, click the ellipsis button ... beside the **Manufacturer** property and on the **Item Manufacturers** dialog box, add new manufacturers **SWAGELOCK** and **WORCESTER**.

4. Click **Apply**.
5. Click **Close** to close the **Item Properties** dialog box.
6. On the **Item List for Current Item Library** dialog box, click **New** and then enter the next set of data as shown.

7. Continue to add items with properties as shown.

Item Number	Storage Number	Description	Size	Manufacturer	Model	Material	Rating	Item Cost	Unit of Measure
32	990006	ELBOW	1/2"	SWAGELOCK	1234575	CS	3000#	12.	Items
45	990007	PLUG	1/2"	SWAGELOCK	1234576	gcs	-	5.	Items
102	25-1365	CONNECTOR MALE	1/2" OD x 1/4" NPT	SWAGELOCK	CS		-	.	Items
149	990001	BALL VALVE	1/2"	WORCESTER	1234567	SS	-	.2	Items
182		PIPE	1"			GCS	80	.	meter

8. Click **OK** to close the **Item List for Current Item Library** dialog box.

Task 3 — Creating Hook-Ups and Attaching Them to Tag Numbers

Create a Hook-Up Type and a Hook-Up

1. Do one of the following:
   - On the module toolbar, click .
   - On the **Associations** menu, click **Hook-Up Explorer**.
2. In the **Hook-Up Explorer**, right-click the **Hook-Up Explorer** folder.
3. On the shortcut menu, click **New** > **Hook-Up Type**.
4. On the **Hook-Up Type Properties** dialog box, type data as shown.

5. Click **OK** to display the new hook-up type in the **Hook-Up Explorer**.
6. In the **Hook-Up Explorer**, right-click the hook-up type you just created.
7. On the shortcut menu, click **New** > **Hook-Up**.
8. On the **Hook-Up Properties** dialog box, type the hook-up name as shown and navigate to the drawing block file in the path <SmartPlant Instrumentation home folder>\CAD\Blocks\H_646MT.sym.

   Tip

   - The path on your system may be slightly different from the one shown.

9. Repeat steps 1 – 4 to complete the creation of another hook-up type with the following information:
   a) Hook-up type name — **VALVES**.
   b) Hook-up type description — **ALL TYPES**.
10. Repeat steps 5 – 7 to complete the creation of another hook-up under the **VALVES** hook-up type with the following information:
   a) Hook-up name — **CV**.
   b) Hook-up description — **CONTROL VALVE**.
   c) Drawing block file name — <SmartPlant Instrumentation home folder>\CAD\Blocks\H_646MT.sym.

   On completion, the hook-up types and hook-ups should appear in the **Hook-Up Explorer** as shown.

Note

- SmartSketch hook-up drawing files have extension .sym, AutoCAD hook-up drawing files have extension .dwg, and MicroStation hook-up drawing files have extension .cel.

Attach Hook-Up Types to Instrument Types

1. Click .
2. On the **Hook-Up Type - Instrument Type Assignment** dialog box, in the **Find instrument type** box, type **FE**.
3. Select instrument type **FE – D/P TYPE FLOW ELEMENT** as shown.

4. Click **Assign**.
5. On the **Assign Hook-Up Type to an Instrument Type** dialog box, select **FLOW** as the hook-up type.
6. Click **OK**.

   The selected hook-up type is now assigned to the highlighted instrument type.

7. In the same way, assign instrument type **FT – D/P TYPE FLOW TRANSMITTER** to hook-up type **FLOW** and instrument type **FV – CONTROL VALVE** to hook-up type **VALVES**.
8. When done, click **Close**.

Note

- It is possible to assign more than one hook-up type to a particular instrument type, for example, FT will have an installation hook-up drawing and a typical cable installation drawing. One hook-up type can also have more than one instrument type assigned to it, as you did in the example.

Assign Instrument Tags to Hook-Ups

1. In the **Hook-Up Explorer**, right-click the hook-up **FLOW INSTR. BELOW RUN-LIQUID**.
2. On the shortcut menu, click **Actions** > **Hook-Up Tag Numbers**.
3. Click **Assign**.
4. On the **Assign Tag Numbers to Hook-Up** dialog box, in the **Instrument Type** data window, first select instrument type **FE – D/P TYPE FLOW ELEMENT** and then select tags **101-FE-100** and **101-FE-102** to be installed as shown.

Tip

- Use the **Ctrl** or **Shift** keys to select more than one tag number.

5. Click **Assign** to assign the tag numbers.

Tip

- When you do this, the tag names no longer appear in the **Tag** data window.

6. Repeat steps 2 – 5 for **FT – D/P TYPE FLOW TRANSMITTER** and choose Tags **101-FT-100** and **101-FT-102**.
7. Click **OK** to close the **Assign Tag Numbers to Hook-Up** dialog box.
8. Click **Close** to close the **Hook-Up Tag Numbers** dialog box.
9. View the assigned tags in the **Hook-up Explorer**.

Task 4 — Assigning Items to the Hook-Up Drawing

Assign Items from the Current Library to a Hook-Up

1. In the **Hook-Up Explorer**, right-click the hook-up **FLOW INSTR. BELOW RUN - LIQUID**.
2. On the shortcut menu, click **Actions** > **Hook-Up Items**.
3. On the **Hook-Up Items** dialog box, click **Assign**.
4. On the **Assign Sub-Library Items to Hook-Up** dialog box, hold down **Ctrl** and select item numbers **32**, **45**, **102**, and **182**.
5. Click **OK** to display the selected items in the item list.

Item Number	Order	Quantity	Unit of Measure	Storage Number	Description
32	1	4.	Items	990006	ELBOW
45	2	2.	Items	990007	PLUG
102	3	4.	Items	25-1365	CONNECTOR MALE
182	4	25.	meter		PIPE

Tip

- You can edit the quantity (number of items or length) of each item by typing the appropriate number in the **Quantity** column.

6. When done, click **OK** to close the dialog box.
7. In the **Hook-Up Explorer**, expand the hook-up **FLOW INSTR. BELOW RUN - LIQUID** and view the hook-up items as shown.

Task 5 — Generating a Bill of Material (BOM)

Generate a Bill of Material

1. In the **Hook-Up Explorer**, select a hook-up type or hook-up.
2. Right-click, and on the shortcut menu, click **Reports** > **Bill of Material**.
3. When prompted to preview the report, click **Yes**.
4. Select the **Edit mode** check box to enable entering of figures in the **Total Qty For Order** column (the last column in the report).
5. Select the **Print tag list** check box and then click .

Viewing and Modifying Data for Multiple Records

SmartPlant Instrumentation allows you to view, modify and print out your data for various entity categories using the Browser module. You can open multiple windows that present you with information.

You access and manipulate your data by defining and using browser views. You can customize each browser view to present the data that is important for you – this includes specifying which properties to display, determining the order of the property columns, and sorting or filtering the data records according to criteria that you specify.

For example

The Control Valve Browser (available after expanding the Process Data and Calculation group) allows you to view and manipulate control valve data (process and calculation results data) of the current plant. The Loop Browser (available after expanding the Loop Drawings group) allows you to view and manipulate the loop data (Loop Form) of the current plant, and so on.

- To open the Browser module, do one of the following:
  - On the main toolbar, click .
  - On the **File** menu, click **Browser Manager**.

  The **Browser Manager** window opens.

Task 1 — Viewing and Editing Tag Number Data from a Browser View

There is a special browser known as the **Instrument Index Standard Browser**, which in addition to allowing you to view and edit tag number data, also provides a means of performing a number of other tasks related to tag numbers, such as generating specification sheets, associating tag numbers with loops, and perfuming I/O assignments.

View and Edit Data for a Single Tag Number in the Instrument Index Standard Browser

1. Open the Instrument Index module by doing one of the following:
   - On the main toolbar, click .
   - On the **Modules** menu, click **Instrument Index**.
2. Open the **Browser View** window for the **Instrument Index Standard Browser** by doing one of the following:
   - On the module toolbar, click .
   - On the **Actions** menu, click **Browse Index**.
3. To edit a single data value, click in the cell containing that value.

   Tips
   - To move to the next data value in the same row, press the **Tab** key.
   - Click **Ctrl** + **Home** to move to the first editable value at the beginning of the tag list or **Ctrl** + **End** to move to the end of the tag list.
   - Non-editable data appears with a yellow background.

Notes

- Some fields only enable you to select from a list. If you want to add to or delete some of the options in a list, you can access the required list from the **Tables** menu in the **Instrument Index Module** window.
- The software automatically saves to the database changes you make to a record when you proceed to another record or when you close the **Browser View** window.

Edit the Browser View Window Records – Multi-Row Editing

Now you will learn how to select a number of tag numbers in the **Instrument Index Standard Browser** for multi-row editing. When selecting the multi-edit option, a dialog box opens where all the available **Browser View** window fields for the selected tag are available. In addition to this multi-editing feature, you are also able to access the appropriate supporting tables if you need to update the available options in the lists.

1. In the Instrument Index module, open the **Browser View** window for the **Instrument Index Standard Browser** by doing one of the following:
   - On the module toolbar, click .
   - On the **Actions** menu, click **Browse Index**.
2. Select multiple tags by doing one of the following:

- To select a group of consecutive tags, select the first tag in the group and then press and hold the **Shift** key, and then click the last tag in the group.
- To select a group of non-consecutive tags, click the first tag that you want to select, then press and hold the **Ctrl** key as you click the other tags that you want to select. (You can also click **Ctrl** + **Home** to move to the beginning of the tag list or **Ctrl** + **End** to move to the end of the tag list.)

3. Do one of the following:
   - On the **Browse** toolbar, click .
   - On the **Edit** menu, click **Properties**.

The **Tag Number Properties** dialog box opens (in this example, the selected tag is 101-FT-100).

4. Edit the data as desired.

   Tip

   - To modify data in a supporting table, click the ellipsis button ... beside the desired property.

5. Click **Apply** to save all the data for the currently selected tag.
6. Click **Next** to display the next tag to be edited (available only if you selected multiple tags).
7. After making changes, click **Apply** again.
8. When done, click **OK** to return to the **Browser View** window.

Edit Data Using the Browser View Buffer

The **Browser View Buffer** data window provides a very convenient way to copy complete tag data from one tag to another. You can copy all the data of an existing tag to a newly created tag or an existing tag.

You can use the **Browser View Buffer** data window to update multiple tags in a batch.

1. In the **Browser View** window, select tag **101-FT-100**.
2. On the module toolbar, click .

   The software copies all the tag data to the **Browse Buffer** data window.
3. Select tag **101-FT-201** which has the same instrument type as tag 101-FT-100.
4. On the module toolbar, click .

The software copies the data from tag 101-FT-100 to tag 101-FT-201.

Task 2 — Finding and Sorting Data in a Browser View Window

Each **Browser View** window offers convenient sort and search tools that you can use to find a specific value and sort the data as you require. You can search for any record that exists in any of the **Browser View** window columns. For example, in the **Instrument Index Standard Browser**, you can find a specific tag number, an instrument type, or an I/O type, and so forth. The following search options are available:

- Find a record by entering the whole value, for example, **101-FV 100** (the full tag number)
  or **CONTROL VALVE** (the full instrument type value).
- Find a record by entering a string which is a part of the value, for example, **FV** (or just **V**)
  or **CONTROL VALVE** (the full instrument type value).
- Search for a record as you type in the value.

Find a Tag Number in the Browser View Window

This feature enables you to find a tag number in a long list of tags on the fly.

1. In the **Browser View** window, select the **Tag Number** column header.
2. On the module toolbar, click .
3. On the **Find** dialog box, select the **As typed** option button and in the **Find what** box, type **101-FT-201**.

Tip

- Use the **Tab** key to skip to the first character of each tag segment.

Watch how the software takes you to the tag number you are looking for in the **Browse** window.

4. Close the **Find** dialog box.

Find a Particular Value in the Browse Window

This feature enables you to find a value in any column in the **Browser View** window. You select the column that you want to search and then use the **Find** command to find the required value in that column. Let's say that you want to find a value in the **Service** column. Do the following:

1. In the **Browser View** window, click the **Service** column header.

   The software selects all the records in this column.

2. On the module toolbar, click .
3. On the **Find** dialog box, select the **String** option button and in the **Find what** box, type **Feed to**.

   Tip

   - Ensure that you have cleared the **Case sensitive** check box.

The dialog box should appear as follows.

4. Click **Find** to highlight the first occurrence of the value you typed in.
5. Click **Find** again to find the next occurrence of the value you typed in.
6. Close the dialog box.

Now find all the occurrences of the following values:

- **Fisher** (in the **Manufacturer Name** column)
- **AI** (in the **I/O Type Name** column)

Sort the Data in the Browser View Window

This option enables you to sort the displayed tag numbers in the **Browser View** window according to the existing columns.

1. Double-click the **Instrument Type** column header to sort the tag numbers according to their instrument type. (If you double-click this column header one more time, the software reverses the sort sequence.)

   Tip

   - If you are searching for a value in a very long list, sort the data according to that column before starting the search.

2. Now sort the tag numbers in the **Browser View** window according to their I/O type.

Task 3 — Using More Features of the Standard Instrument Index Browser

There are several other features that are specific to the **Standard Instrument Index Browser**. We will now show you some of these useful features that will make your work easier and faster.

Use the Browse Window Shortcut Menus

Right-clicking anywhere in the **Browser View** window opens a shortcut menu that allows you to perform the most common actions available in the Instrument Index module. Try it now and see what options are available.

Open a Document Associated with a Tag Number

This feature enables you to access the documents associated with the selected tag number.

1. In the **Browser View** window, select tag number **101-FT-100**.
2. On the module toolbar, click to open the **Documents** pop-up window.

Tips

- Available documents appear in **bold**. The software created some of these documents automatically based on your definitions for the instrument type profile associated with the tag when you created it.
- Disabled options signify that no documents of this type have yet been created for the selected tag.

3. Click on each available option to access the associated documents.

Tip

- You can view the documents, print them out, and save them as external files if necessary.

4. Click ☒ to close the **Document** pop-up window.

Associate an External Document with a Tag Number

This option allows you to associate an external document (a file not created in SmartPlant Instrumentation) with a selected tag. After making this association, you will be able to open this document from the **Browser View** window.

1. Select the tag with which you want to associate an external document.
2. Right-click the selected tag, and on the shortcut menu, click **Tag Number Activities** > **Associate Documents**.

   The **Associated Documents** dialog box for the selected tag opens.
3. Click **Associate** to open the **Associate External Documents** dialog box.
4. Navigate to the folder in the path <SmartPlant Instrumentation home folder>\CAD\Blocks\ where the CADBlocksLib.xls file is located.

   Tip

   - The actual location of the home folder on your computer depends on where you installed SmartPlant Instrumentation.

5. Click **Open**.

   A new data line is added to the **Associated Documents** dialog box displaying the information about this external document.

   The dialog box should appear as shown (your file location may be different from the location shown in the example).

6. Click in the **Description** field and type a description as desired.
7. Click **OK**.
8. Now open this file by doing the following:
   a) Select the tag with which you just associated the file.
   b) Click to open the **Documents** pop-up window.
   c) Scroll down and click **Other** to open the **Associated Documents** dialog box.
   d) Select the document that you want to view and click **Open**.

Note

- You can associate any external document in any file format supported by the Windows version installed on your computer. Make sure you have the appropriate application needed to open the external file.

Task 4 — Creating 'Junction Box Assignment per Tag' Browser

You will now create a new browser called **Junction Box Assignment per Tag**, containing a list of all tag numbers with their associated junction box numbers and connection terminals. To create a new browser, you need to create your customized **style**, **sort**, **filter**, and then a **view** that becomes your new customized browser.

Create Your Customized Style

1. Open the Browser module by doing one of the following:
   - On the main toolbar, click .
   - On the **Modules** menu, click **Browser**.
2. In the **Browser Manager** window, expand the **Instrument Index** browser group by clicking the ⊞ icon.

3. Right-click **JB - Index Browser**.
4. On the shortcut menu, click **Add View**.
5. In the **View profile** data window, click **Edit** to enable editing.

Tip

- The **Edit** label changes to **Save**.

6. Enter data as shown.

7. Click **Save** to save the new view profile.
8. In the **Browser groups** data window, expand **JB-Tag Index** and click **Style** to open the **Style** section.

9. Click **Edit** to enable editing.

Tip

- The **Edit** label changes to **Save**.

10. In the **Name** field, select the displayed name and overtype this name with a new **style** name — **Junction Box Style**.

Tip

- You can modify the field names to suit your needs, for example, in the **Field Name** column, select **Panel Name/DCS/Cabinet** and type in **Cabinet** instead.

11. Select the **View** check box for the following fields that are to be included in the new style (the color of the selected field changes to red):
    - **Tag Number** (selected by default and must be included)
    - **Cable Name**
    - **Panel Name**
    - **Service**
    - **Terminals**
    - **Location Desc**
    - **Instrument Type**

    The window should appear as shown.

12. Click **Customize** to open the **Customize Field Layout** dialog box.

13. Customize the layout as follows:
    a) To rearrange the field sequence, drag field headers to the location where you want them to appear.
    b) To modify field size, drag field borders to the left or to the right.

14. Click **OK** to generate the customized layout and return to the **Browser Manager**.
15. Click **Save** when done (if **Edit** is displayed, click it first then click **Save**).

    All the fields you selected now appear in the **Style Field List**.

Style field list - Junction Box Style

Field Name	Length	AutoSize	Sum	Avg.
Tag Number	50	No	No	No
Cable Name	50	No	No	No
Panel Name	50	No	No	No
Service	52	No	No	No
Terminals	20	No	No	No
Location Desc	10	No	No	No
Instrument Type	6	No	No	No

Create a Customized Sorting Sequence

1. Make sure that the hierarchy tree of the **JB-Tag Index** browser is expanded. If it is not, double-click it so that it appears as shown.

2. In the **Browser groups** data window, click **Sort** to display the **Sort** group box.
3. Click **Edit** to open the fields for editing.
4. In the **Name** field, type **Ascending by Tag**.
5. Click in the highlighted **Field Name** column and from the list, select **Tag Number**.
6. In the **Order** column, select the **Ascending** check box.

7. Click **Save** to save your sort definition.

   Your new sort definition now appears in the **Sort Field List** data window.

Create a Customized Filter

The Browser module enables you to define a filter for your new browser. However, the browser you are creating now does not require a filter.

1. Make sure that the hierarchy tree of the **JB-Tag Index** browser is expanded. If it is not, double-click it.
2. In the **Browser groups** data window, click **Filter** to display the **Filter** group box.
3. Click **Edit**.
4. In the **Name** field, type **All Instruments**.
5. Define the filter criteria as follows:
   a) In the **Field Name** column, select **Tag Number**.
   b) In the **Operator** column, select **like**.
   c) In the **Value** column, type **%F%**.

   Tip

   - The '%' symbols behaves as a wildcard, and in this example will display all tag numbers that contain the string 'F'.

6. Click **Save**.

Open Your New Customized Browser

Now that you have completed defining your new browser, you can open it and use it.

1. In the **Browser groups** data window, select the **JB-Tag Index** browser view.
2. On the **Browser** toolbar, click **Open View** to open your newly created browser view.

3. If desired, modify the layout as follows:
   a) To change the column sequence, drag column headers to the new positions where you want them to appear.
   b) To change the width of a column, position the cursor on the border between two column headers and drag the border to the right or to the left.

Note

- If you change the layout from the browser view, to retain the modified column sequence and widths the next time you open the browser view, right-click anywhere inside the browser view, and on the shortcut menu, click **Save Field Layout**.

Task 5 — Creating a New Customized Process Data Browser

This task involves the creation of a new browser based on the process data available in SmartPlant Instrumentation.

Create a New Browser — 'Flow Meters'

You are now going to create a new browser in the **Process Data and Calculation** browser group called **Flow Meters**. Use the procedure given for the **JB-Tag Index** browser.

1. In the **Browser Manager**, expand the **Process Data & Calculation** browser group tree hierarchy.
2. Select **Flow Instrument Browser** and add a new view.
3. In the **View name** box, type **Flow Meters**.
4. From the **Data level** list, select **Unit**.
5. Click **Save**.
6. In the **Browser groups** data window, expand the view that you created and click **Style** to display the **Style** data window.

   Tip

   - Before making your selection of the fields to include for the style, click the **Field Name** column to arrange the field names in alphabetical order.
7. Select the following fields to include in the style:
   - **Beta Ratio**
   - **Bleed/Vent Hole Diameter**

- **Flow Maximum**
- **Flow Minimum**
- **Flow Normal**
- **Fluid Name**
- **Fluid Phase** (PD_FLUID_STAT)
- **Full Scale Flow**
- **Line Number**
- **Orifice Diameter**
- **Pipe Material**
- **Service**
- **Tag Number**

8. Sort by tag number, as previously.
9. Define a filter as shown (to display only tags that include the string 'FE').

10. Save all your settings.
11. Open the new browser to see the results, which should appear similar to the one shown.

Note

- The data displayed depends upon the number of flow elements you have created.

Task 6 — Customizing the Printout Structure

You will now learn how to customize the layout structure for a printed out browser view. You can select the fields to be included in the printout and determine the following:

- Column sequence
- Column size
- Field layout: single field per column or multiple fields per column (up to five fields per column)
- Text and header font size
- Column and header height

Create a New Line Browser and Customize Its Printout Structure

1. In the **Browser Manager** window, expand the **Instrument Index** browser group.
2. Select **Line Browser** and click to add a new browser.
3. Define the view profile of the new browser as shown.

4. When defining the style, select the following fields:
   - **Line Internal Diameter**
   - **Line Number**
   - **Line Schedule**
   - **Line Size**

- **Line Type Name**
- **Line Unit**
- **Pipe Material**

5. Click **Layout** to open the **Layout** dialog box (if this button is not active, click **Edit**).

6. Select labels from the **Column Header** data window and drag them to the columns where they are to appear in the print-out as shown.

Tips

- Your printout layout can take the form of single fields in each column or multiple (no more than 5) fields in a single column.
- The fields that you do not select from the **Column Header** data window will not show up in the printout even if they exist in the **Browser View** window for the current browser.

7. Click **Font** to open the **Select Font Size** dialog box where you customize the column and header font size as well as the heights of the rows and the header.

8. If the values are different from the font sizes shown, use the spin boxes to set the values

Tips

- You must define the column height before generating the printout structure, otherwise the **Generate Printout Structure** dialog box will not display any data.
- The font size and the height properties are measured in points. 1 point = 1/100 inch.

9. Click **OK** to accept your settings and return to the **Layout** dialog box.
10. Click **Generate** to open the **Generate Printout Structure** dialog box where you customize the column sequence and size.

11. Set the printout structure as follows:
    a) Set the column sequence by dragging the column header to the required position.
    b) Set the column size by dragging the edge of the column either to left or to the right.

c) Click **OK** to accept the settings and return to the **Layout** dialog box.

12. Click **Save** to save all the settings and close the **Layout** dialog box.

Tip

- The multiple field layout is shown in the printout only.

13. On the module toolbar, click to display your customized browser.
14. Click to print out the new browser view.
15. At the prompt, click **Yes** to display the print preview.

    Observe how the **Browser View** fields are arranged in the generated print preview.

Task 7 — Using the Sum and Avg. Features in Printed Reports

The **Sum** and **Avg.** features in the **Styles Settings** of the **Browser Manager** window enable you to display the total and average values of all the numeric values in the rows contained in each group delimited by a group separator in the print preview.

Create a New Cable List Browser and Print Out the View Report

You are now going to create a new browser called **Cable List** for which the style definitions contain sum and average settings. You will then generate the browser view report for this browser and see how these settings affect that report.

1. In the **Browser Manager** window, expand the **Wiring** browser group.
2. Select **Cable Browser** and click to add a new browser view.
3. Define the view profile of the new browser as shown.

4. Click **Save**.
5. In the right pane of the **Browser Manager**, double-click **Style** to define the style settings for the new browser.
6. Name the style **Default Cable List Style**.
7. When defining the style, select the following fields:
   - **Cable Name**
   - **Cable Description**
   - **End 1 Location**
   - **End 2 Location**
   - **Length**
   - **Units**
   - **Jacket Color**
   - **Cable Type**
   - **Manufacturer**
   - **Model** (selected automatically when you select **Manufacturer**)
8. For the **Length** field, select the **Sum** and **Avg.** check boxes as shown.

9. Click **Customize** to modify the width and sequence of the selected fields and determine how they appear in the **Browser View** window.
10. Click **Layout** to set the printout structure of the new browser view. Customize the printout structure as needed.

    Tip

    - You cannot use the multiple field layout if you want to display the sum and average values in the printout.
11. Click to set a sorting sequence. Click **Edit** and then select the **Cable Description** field for sorting the data by. This allows you to group all the fields in the printout according to this field.
12. After saving the settings, click to display your customized browser.

    Tip

    - In the **Browser View** window, make sure that all the fields in the **Length** column contain values and that the unit of measure in the **Unit** column is **meter**.
13. Click to print out the new browser view.
14. At the prompt, click **Yes** to display the print preview.
15. In the generated Cable List Report, do the following:

    a) In the **Group separator** group box, select the **Insert a separating line** option button.

    b) Select the check box beside **Cable Description** as the group separator and click **OK**.

The software regenerates the report displaying the total and average length values for each group having the same value of **Cable Description**.

Cable Name	Cable Description	End 1 Location	End 2 Location	Length	Units
C-101-JB-DCS-001/1		JB	MARSHALLING	220	mete
C-101-JB-DCS-001/2		JB	MARSHALLING	220	mete
C-101-JB-PLC-001		JB	PLC Cabinet	250	mete
C-101-MR-DCS-001		DCS	MARSHALLING	15	mete
C-101-MR-DCS-002		DCS	Marshalling Cabinet	15	mete
C-101-MR-DCS-003		DCS	Marshalling Cabinet	15	mete
Sum:				735	
Avg:				122.5	
C-101-FT -905	1 Pr,Awg#20,Bk/Wh,Indiv S			20	mete
C-101-FT -906	1 Pr,Awg#20,Bk/Wh,Indiv S			20	mete
C-101-FT -907	1 Pr,Awg#20,Bk/Wh,Indiv S			20	mete
C-101-FT -908	1 Pr,Awg#20,Bk/Wh,Indiv S			20	mete
C-101-FT -909	1 Pr,Awg#20,Bk/Wh,Indiv S			20	mete
C-101-FT -910	1 Pr,Awg#20,Bk/Wh,Indiv S			20	mete
C-101-FT -919	1 Pr,Awg#20,Bk/Wh,Indiv S			20	mete
C-101-FY -905	1 Pr,Awg#20,Bk/Wh,Indiv S			20	mete
C-101-FY -906	1 Pr,Awg#20,Bk/Wh,Indiv S			20	mete
C-101-FY -907	1 Pr,Awg#20,Bk/Wh,Indiv S			20	mete
C-101-FY -908	1 Pr,Awg#20,Bk/Wh,Indiv S			20	mete

Tip

- Your report output may differ from the one shown according to the cables that exist in the database.

16. Save this report as an external file and print it out if desired.

Demo P&ID

REVISIONS			
No.	DESCRIPTION	BY	APP'D	DATE
P0	FOR TUTORIAL	MS		01.11.04
0	UPDATED	MS		10.10.04
1	ADD FI-201	MS		15.08.99

REFERENCE DRAWINGS

DWG No.	SUBJECT
	TUTORIAL BOOK

INTERGRAPH

DESIGNER			
		PROP'O.	BY	DATE
		DES.		
		DRAWN.		
		CHECK.		
		APP.		
		CHECK.		
		INS ENG		
		APP.		

DESCRIPTION : P&ID TUTORIAL FOR DEMO PROJECT

INTERGRAPH

DWG. NO.: 101-PID01-001

REV 1

FILE: PID.DWG D:\MYFOLDER1\TUTORIAL

Wiring Block Diagram

FILE: pid_wir.dwg