SmartPlant P&ID
Creating a P&ID Course Guide

Process, Power & Marine

INTERGRAPH

Table of Contents

Preface

This document is a course guide for the various SmartPlant P&ID User Guides. The content is the similar as the online Help delivered as part of the software with the exception of the Labs.

If You Need Assistance

Intergraph Process Power and Marine Online

Our web site brings you fast, convenient, up-to-the-minute information about Intergraph's products, services, and direction. Our web address is:

http://www.intergraph.com/

Intergraph Process, Power and Marine Support

You may log a service request via the web at:

http://www.intergraph.com/support/

In the United States, call Intergraph's standard support number at   1-800-766-7701 from 7:30 AM to 5:30 PM CST, Monday through Friday, except holidays.

If you are outside the United States, please call your local Intergraph office. The most up-to-date list of international offices and distributors is available on the web at

http://www.intergraph.com/contact/worldwide.asp

Please have the following information available when you call or log a Service Request

- Your Name and Telephone Number

- The Product's name

- A description of the Problem

- The version of the product including the service pack number.

Intergraph Process, Power and Marine Directory

Intergraph General Information

- U.S.

- Canada

- All Countries – 1-256 –730-2000

Customer Response Center (U.S. & Canada Only)

- Customer Support – 1-800-766-7701

- U.S. Customer Care Center

- Fax – 1-256-730-3830

Training Registration (U.S. Only)

- 1-800-766-7701

- http://www.intergraph.com/training/

Mailing Address

300 Intergraph Way
Madison, Alabama 35758
U.S.A.

Getting HELP in SmartPlant

The software's user assistance supplies command information as you perform tasks. You can access different kinds of information any time you are running the software. This information could include reference topics, narrative descriptions, or instructional material.

In addition, the software provides several learning tools that you can activate from the online **Help** menu.

The software offers the following important user-assistance features:

Online Help

• Complete command descriptions give you more information. When you click **Help** or press **Shift** + **F1**, the pointer changes to a northwest arrow with a question mark. You can then get context-sensitive Help for any command by clicking the toolbar button or by clicking the menu command at the top of the window.

• A structured table of contents, an index, and full-text search capabilities provide easy access to Help topics. Press **F1** any time you need online Help during a design session. When a command is active, the Help topic for that command appears. If no command is active, then the table of contents for the Help topics appears.

You can also access the table of contents by clicking **SmartPlant P&ID Help** on the **Help** menu.

Learning Tools

• You can use **About SmartPlant P&ID** on the **Help** menu to see your software version and license information.

User Interface Features

• ToolTips helps you find command names. When you pause the pointer on the command button on the toolbar, a yellow label displays the command's name.

• Brief command descriptions show you the basic function of a command. When you point the pointer at the command button on the toolbar, the description appears in the Status Bar at the bottom of the window. Also, messages that explain your actions for each step of the command appear in the Status Bar.

SmartPlant P&ID Help Command (Help Menu)

Displays the table of contents for the SmartPlant P&ID Help topics, which includes step-by-step instructions for using the software, reference information, examples of features, and technical support information. It also provides access to the Help index and full-text search.

About Command (Help Menu)

Displays information about your copy of the software, including the version number and the copyright, legal, and licensing notices.

Printable Guides Command (Help Menu)

Opens a page in your Internet browser that includes the links to the User's Guides in portable document format. Click on a link and the guide is opened in the appropriate application, from which you can print it. The User's Guides contain the same information as the online Help.

Upon installing SmartPlant Engineering Manager or P&ID, the printable guides (.pdf) may also be found on your disk at ~ \Program Files\SmartPlant\Engineering Manager\Program\resdlls\0009 or ~\Program Files\SmartPlant\P&ID Workstation\Program\resdlls\0009.

List of Printable Guides	
SmartPlant Engineering Managers User's Guide	SmartPlant Workshare Configuration and Reference Guide
SmartPlant Catalog Manager Users's Guide	SmartPlant P&ID User's Guide
SmartPlant Data Dictionary Manager Users's Guide	SmartPlant P&ID Drawing Manager User's Guide
SmartPlant Data Dictionary Template Comparison Utility Guide	SmartPlant P&ID Options Manager User's Guide
SmartPlant Filter Manager Users's Guide	SmartPlant P&ID Rule Manager User's Guide
SmartPlant Format Manager User's Guide	SmartPlant P&ID Insulation Specification Manager User's Guide
SmartPlant Line Style Editor Users's Guide	SmartPlant P&ID Symbol Libraries Reference Guide
SmartPlant Reference Data Synchronization Manager Guide	SmartPlant P&ID to PDS Piping Data Transfer Configuration and Reference Guide
SmartPlant Projects Configuration and Reference Guide	SmartPlant P&ID Installation & Upgrade Guide

Introduction

The Intergraph® SmartPlant® family of process industry solutions is an open line of discipline-specific software tools that provide an integrated solution for the entire plant life cycle. Knowledge-based, intuitive, easy-to-use, accessible, flexible, and data-driven, SmartPlant supports global workflows. The software enables users to create logical and physical definitions of the plant model and enables access to plant data from conceptual design to decommissioning.
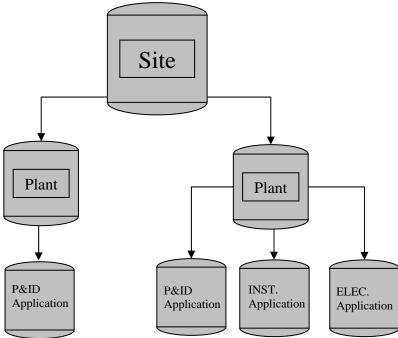
SmartPlant is the fulfillment of the Intergraph vision to speed and improve the creation of information and to provide this data to multiple users at any moment in the appropriate form. Workflows are compressed, reducing production time, lowering costs, enhancing global execution, and extending the life and usability of plant information.

The successor to the Intergraph Plant Design System (PDS™), SmartPlant includes expanded functionality for front-end engineering and design (FEED), construction, operation, and maintenance phases.

Introducing SmartPlant Engineering Manager

SmartPlant Engineering Manager provides all the tools you need to effectively set up and manage your SmartPlant work. SmartPlant Engineering Manager takes advantage of a client/server design that greatly enhances performance and lowers the cost of ownership. Because it is built on few Microsoft dependencies and is not web-based, SmartPlant Engineering Manager requires no web server. The intuitive user interface design, with its streamlined layout, allows you to easily manage user access and to share plant data.

Supporting SmartPlant P&ID, SmartPlant Electrical, and SmartPlant Instrumentation, SmartPlant Engineering Manager manages the plant structures while the applications themselves are responsible for manipulating the actual data (including creating, deleting, modifying, launching, and so forth). SmartPlant Engineering Manager allows you to view not only the data related to the whole site but also data related to individual plants and projects. You can create and maintain SmartPlant Engineering sites, plant structures and projects, in addition to adding plant group types, modifying plant attributes, creating and modifying hierarchies, and associating SmartPlant applications.

Introducing SmartPlant P&ID

SmartPlant P&ID creates intelligent P&IDs by populating the database with relevant plant data. This method provides valuable information throughout the plant life cycle. As a data-centric, rule-based solution for the P&ID life cycle, SmartPlant P&ID helps users improve design quality, data consistency, and standards compliance. With quick access to supporting engineering data, SmartPlant P&ID significantly cuts design and modification time and increases accuracy with its exclusive data-centric approach and use of design rules, automatic checks, and drag-and-drop capabilities.

SmartPlant P&ID is vastly different from graphic-driven P&ID solutions of today. All data from the P&ID is stored in the plant database and adheres to plant standards. The graphical representation of the P&ID is a view or a report of the data. The strong data import and export facilities of SmartPlant P&ID allow users to populate the system with relevant plant data, such as process data from process simulation databases based on Aspen Zyqad from Aspen Technologies, Inc. or equipment and line lists. You can then use this information in the SmartPlant P&ID Stockpile to design the P&ID.

The rule-based and automation capabilities of SmartPlant P&ID also differentiate it from other P&ID systems. SmartPlant P&ID features a comprehensive, user-definable rule-based system that assists the engineer during the design phase of the plant and subsequent life cycle phases. Data is entered directly into the database; rules are executed; and feedback is immediate. The design rule-base confirms data consistency and compliance with plant and engineering standards, allowing faster, more efficient design with less iteration.

SmartPlant P&ID incorporates the latest Microsoft technologies, such as OLE automation, to provide integration with existing data and other systems. Running on Microsoft Windows XP and Windows 2000, SmartPlant P&ID does not require a traditional, expensive CAD engine for P&ID creation. The open architecture of SmartPlant P&ID permits integration with other systems, such as Intergraph PDS, SmartPlant Instrumentation, and Aspen Zyqad, all of which allow users to share data with third-party software.

The Interactive Graphic commands supplied by the SP P&ID software support a wide range of schematic and 2D physical applications, which include:

- o Equipment commands

- o Piping commands

- o Instrumentation commands

- o Design Commands

- o Assemblies

The reference database (RDB) includes files such as symbols, report files, and templates, which support actions performed during the design creation task. The RDB supports the international engineering standards of the American National Standards Institute (ANSI), Desutsche Institut fur Normung (DIN), Process Industry Practices (PIP), ISO Plus (International Organization for Standardization)

With **SmartPlant P&ID**, you can easily perform the required tasks to create a detailed plant model. You can place components such as equipment, piping, and instrumentation from the **Catalog Explorer** or the **Stockpile** into graphical representations of your plant model in the drawing.

A key feature of SmartPlant P&ID is using catalogs to create instances of components within the plant model. Within this environment, you can drag and drop items from the catalog into appropriate model views.

SmartPlant P&ID uses a stockpile concept to store data that does not currently reside in the drawing. Examples of items that are stored in the **Stockpile** are items deleted from the drawing in which the Tag Required Flag = True has been set in Catalog Manager and Item Type has been added to Options Manager > Settings > Stockpileitems. Off-Page Connector mates, Loops, Packages, Test System, Safety Class and imported data are additional items that are placed in the stockpile vs. the Plant Model.

In the **Properties** window, you can add values for various properties of each item after you place it in your drawing. You can also annotate your drawing with labels as you design the drawing.

If you repeatedly place particular groups of items, you can save the group of items as an assembly. Assemblies appear as symbols in the **Catalog Explorer** list view so that you can place them exactly as you place other symbols in your drawing.

As you work on your drawing, you can monitor any inconsistencies in your design by reviewing the inconsistency indicators. These indicators appear in your drawing as soon as an inconsistency occurs in the design. You can resolve inconsistencies using hints from the **Consistency Check** dialog box.

At any time during the P&ID creation workflow, you can generate reports to help you keep track of information in the P&ID. After you finish the P&ID, you can generate MTOs for the components in the drawing, for example.

You can also print the drawing.

SmartPlant Engineering Manager Program Group

The SmartPlant Engineering Manager program group provides several utilities for managing your plant data.

SmartPlant Engineering Manager allows you to create the SmartPlant site and plants. You can create plant structures, plant groups, hierarchy templates, as well as define the access to plant data on many levels.

Catalog Manager allows you to create and modify symbols and labels.

Data Dictionary Manager allows you to add properties to SmartPlant database tables, define external programs, view relationships, and create and modify select lists.

Data Dictionary Template Comparison Utility allows you to compare two data dictionary templates at a time.

Filter Manager allows you to create and modify filters to discriminate on database data. Filters are used for displaying data in symbology, gapping, graphical views, reports, rules, and so forth.

Format Manager defines available formats for units of measure properties.

Reference Data Synchronization Manager allows you to synchronize reference data (including data dictionaries, rules, options, and symbols) between plants.

Upgrade Utility guides you through upgrading your SmartPlant Engineering data.

Refresh Site Roles Utility allows you to automatically refresh the roles in a site on a scheduled basis.

SmartPlant P&ID Program Group

SmartPlant P&ID provides multiple views of a central, unified data structure that represents the plant model. A view is a visual presentation of the data that composes the plant model and can be a schematic drawing or a table. The plant model is the computer representation of the conceptual design in its entirety, including all plant components and their relationships. By manipulating model views, you can organize the information within the plant model to better understand and maintain the data. SmartPlant has several programs and utilities for running and managing your plant data.

SmartPlant P&ID provides the design environment for SmartPlant.

Drawing Manager allows you to create and delete P&IDs and drawing versions and print multiple drawings. Drawing Manager allows you to perform Workshare- and project-specific commands.

Insulation Specification Manager allows you to create and modify lookup tables for insulation specifications and thicknesses.

Options Manager defines plant-wide graphic standards for symbology, gapping, heat tracing, and formats. Options Manager also defines paths to SmartPlant files and directories.

Rule Manager defines rules for placement and property copying on placement.

Programming Help provides comprehensive information for customizing the software by using the Automation layer of the SmartPlant P&ID application. You should be familiar with SmartPlant P&ID interactively and understand the basic concepts of programming using Automation

SmartPlant Engineering Manager performs higher-level data management tasks, such as specifying user permissions, designing plant hierarchies, and so forth. For more information about SmartPlant Engineering Manager, see *SmartPlant Engineering Manager Help*, *SmartPlant P&ID Installation Guide*, or the *SmartPlant P&ID Upgrade Guide*.
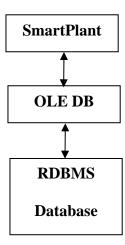
The SmartPlant Database

A database is a body of information, or data, specific to an application such as SmartPlant P&ID. The data may or may not be related, and it is usually stored in one or more data files or disk partitions in such a way that it can be recalled at any time in a variety of ways.

The software that performs the organization, storage, and manipulation of data in a database is called a Database Management System (DBMS). One DBMS may differ from another in many ways, the most important of which is in the way the data is organized. One of the most popular DBMSs is the Relational Database Management System (RDBMS).

The RDBMS controls access to a database through a concept known as the user. A database user is similar to an operating system user. That is, database users are recognized by their username and a password. Database users also have ownership of and access privileges to the data in the database.

A simplified model of SmartPlant and the database communication is shown in the following illustration:

```
┌─────────────────┐
│   SmartPlant    │
└─────────────────┘
         ↕
┌─────────────────┐
│     OLE DB      │
└─────────────────┘
         ↕
┌─────────────────┐
│     RDBMS       │
│                 │
│    Database     │
└─────────────────┘
```

The SmartPlant data model is contained in the Data Dictionary, which is stored in the database. The data dictionary contains the structure for tables that are maintained in the database. Data Dictionary Manager displays the database tables and properties. An administrator can also add properties to tables and create or modify select lists (properties with pre-determined values) with **Data Dictionary Manager**.

SmartPlant P&ID Reference Data

The SmartPlant product is delivered with a set of Symbols (.sym format), Rules.rul, Format.txt, ProjectStyles.spp, InsulationSpec.isl, Templates (.pid format), Borders (.igr format), Report files (.xls format), ExportLayer.xls, Engineering Framework files that can serve as a starting point for P&ID creation. This example reference data is located in the **~\SmartPlant\P&ID Reference Data** directory. This data can be fully customized to meet a project's standards and should be modified with the appropriate Program Group, for example the rules.rul file is edited through Rule Manager.

Reference the printable quide of SmartPlant P&ID Symbol Libraries Reference Guide or the ~Program Files\SmartPlant\P&ID Workstation\Program\resdlls\0009\SymbolLibrariesRefGuide.pdf for a book of the delivered symbols.
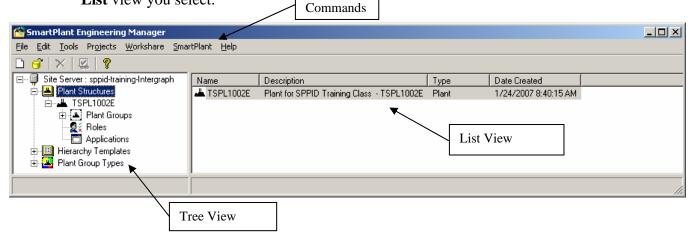
RDB supporting the engineering standards of Process Industry Practices (PIP) and ISOPlus are also available.

SPEM and SPPID Overview

The SmartPlant Engineering Manager user interface consists of three main areas. The **Menu**, **Tree** view on the left displays the plant hierarchy in a tree format. The **List** view on the right displays property data for the nodes in the selected branch in the tree.

The commands in SmartPlant Engineering Manager are node-specific, meaning that the availability of a command depends on which node in the **Tree** view or item in the **List** view you select.

Commands

List View

Tree View

Site Server Root
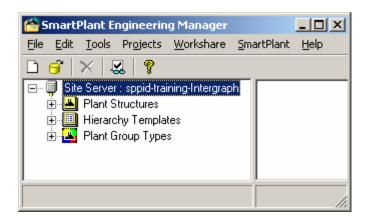
The **Site Server** node is the root directory for each site when opened in SmartPlant Engineering Manager. A site is a logical unit of data that is normally used to model a collection of physical plants. Every plant within a site has a unique identity.

In SmartPlant, you access a site by opening the smartplantv4.ini file, which contains the database type, connection alias, and the schema information for the site and the site data dictionary. The site schema basically keeps track of the plants in the site. You can place this .ini file in any location on any workstation and share it out to other users. Therefore, a site server or site is simply any workstation where the SmartPlantV4.ini file is stored.

For each plant structure, the **Site Server** node contains a **Plant Structure** node that contains its related **Plant Groups**, **Roles**, **Applications**, **Satellites**, and **Projects** nodes.

The three main root nodes are listed under the **Site Server** root

- Plant Structures
- Hierarchy Templates
- Plant Group Types

The **Site Server** root also contains the **Hierarchy Templates** and **Plant Group Types** nodes, for use by members of the Site Administrators user access group.

🔷 Important

- While you can connect to only one site at a time, you can have more than one site on any given computer.

The **Site Server Options** dialog box displays when you run SmartPlant Engineering Manager for the first time. This dialog box will display during future sessions only when SmartPlant

Engineering Manager cannot find an active site server or the site connection information in the **SmartPlantV4.ini** is invalid.
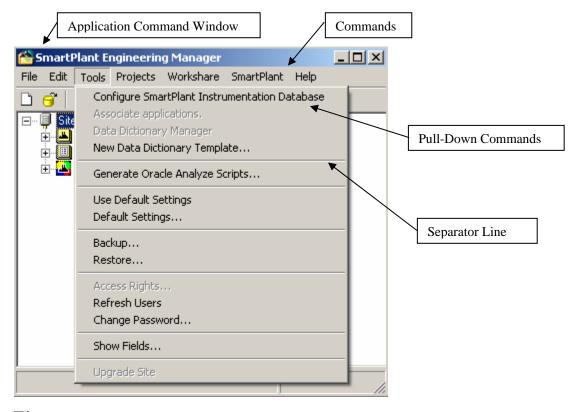
- Create a new site - Starts the New Site Server wizard, which steps you through creating the site server.

- Connect to an existing site - Allows you to browse to the location of an existing SmartPlantV4.ini file.

- Restore site from backup - Starts the Restore Site Server wizard.

✍ Notes: You cannot create a new site unless you log in using a domain account that has administrator privileges on the computer where you are running SmartPlant Engineering Manager.

Before you can create a site server, you must create a database instance in Oracle and the database must be started. Select Start > Settings > Control Panel > Services to verify that the database is running.

Commands

Commands are your main source to interaction with the software application. Each application has its own command window. It is from this command window that you access all other pieces of the application's graphical interface.

Application Command Window

Commands

Pull-Down Commands

Separator Line

✍ Notes: During certain operations, the system dims some of the commands. For example, if a particular command is not applicable at a certain selection that command is disabled until it is applicable. This feature has been applied to commands to clarify the user interface.
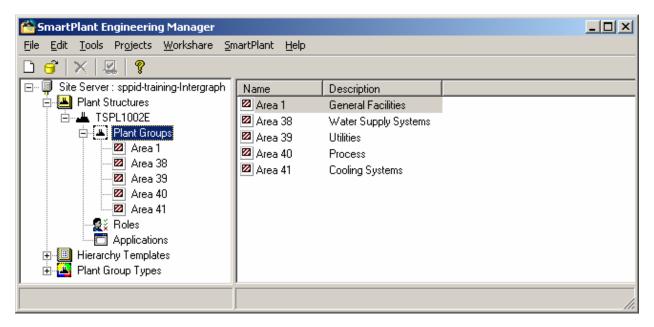
Tree View

The Tree view displays plant groups in their plant structures. You define both the plant structure and the plant group types. At the base of the tree is the Site Server

root. The Site Server root was created when the site administrator completed the **Create Site Schema Wizard.**
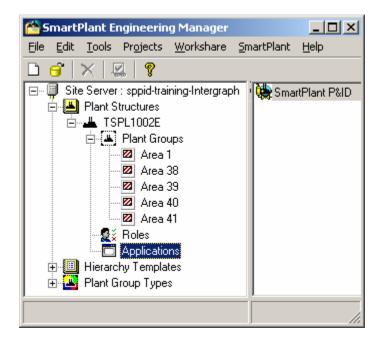
List View

The **List** view displays property data for the children of the selected node in the tree. The **List** view sorts by the column heading that you click. You can use the **Tools** > **Show Fields** command to control which properties appear in the **List** view and the order in which they appear.

Plant Structures Root

The Plant Structure Root contains all of the plants created in the site. Each plant structure represents the physical hierarchy (plant breakdown structure) of your plant. A hierarchy is used as a template to create a plant structure, defining the available plant groups for that particular plant. The hierarchy is essentially a set of rules that the plant structure must follow. Under each plant structure are its related Plant Groups, Roles, Applications, Projects and Satellites nodes.

Plant Groups - Displays the plant breakdown structure (physical hierarchy) of the plant.

Roles - Displays the user access roles defined for the plant.
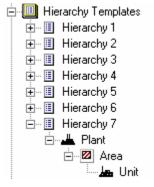
Applications - Displays the applications associated with the plant.

Projects - Displays the projects associated with the plant. This node appears only when the plant is enabled for projects. The project structures themselves are displayed at the plant structure level under the Plant.

Satellites - Displays the satellite slots created in the plant. This node appears only when the plant is enabled for Workshare.

Hierarchy Templates Root

The Hierarchy Template Root displays the various hierarchy templates that can be used to generate the structure of a plant. A hierarchy is made up of a set of two or more plant group types that are arranged in a tree structure.

♦ **Important**

- Drawings are not allowed in the top-most root item in a hierarchy. Therefore, **Allow P&ID Drawings** is not available at the root item level in a hierarchy.

- If you use a hierarchy in which **Allow P&ID Drawings** is not turned on for at least one level, the plant cannot be used in SmartPlant P&ID, but can be used in SmartPlant Electrical.

The plant structure hierarchy does not reference the hierarchy template after the plant structure is created. SmartPlant Engineering Manager writes a copy of the hierarchy template to the plant database when the new plant is created. This reduces the dependency on the site for the hierarchy definition and allows you to modify the hierarchy template independent of whether any plant structures used it during their creation. You cannot modify a plant structure hierarchy after the plant structure has been created.
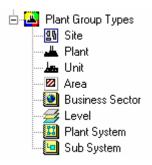
Using Custom Hierarchies

SmartPlant integration supports custom hierarchies, as long as they contain a minimum of three levels. By default, the delivered SPEMdatamap.xml file is compatible with the standard SmartPlant Plant > Area > Unit hierarchy.

- After registering, SmartPlant Engineering Manager cannot retrieve the PBS document if the plant and SmartPlant hierarchies are not compatible. To be compatible with the SmartPlant hierarchy, your plant hierarchy can contain less than or equal, but not more than the number of levels in the SmartPlant hierarchy.

- SmartPlant Engineering Manager retrieves from the SmartPlant hierarchy only the hierarchy levels it needs. For example, if your plant hierarchy contains 4 levels and the SmartPlant hierarchy contains 8 levels, only the top 4 levels of the SmartPlant hierarchy are retrieved.

- Hierarchy item names at the same level do not have to match. Hierarchies are mapped by depth (level), not by name.
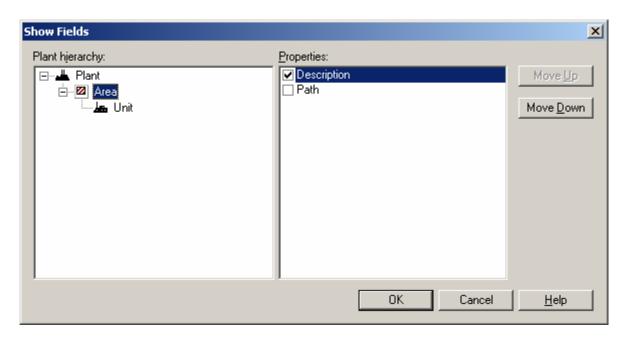
Plant Group Types Root

The Plant Group Types Root displays the building blocks used to create plant breakdown structures (hierarchies). Eight plant group types are delivered by default. You can also create custom plant group types.

Tools > Show Fields Command

The **Tools > Show Fields** command allows you to specify which properties display in the **List** view and the order in which they appear. The **List** view columns are the properties of the items in the **Tree** view.

Items - Lists the items available for display in the **List** view. Select an item from this list to display its properties in the **Properties** list.

Properties - Lists all available properties for the selected item. Properties that that are checked will display in the **List** view in SmartPlant Engineering Manager.

Move Up - Moves the selected property up in the list. The higher a property is in the list, the farther to the left the property appears in the **List** view.

Move Down - Moves the selected property down in the list. The lower a property is in the list, the farther to the right the property appears in the **List** view.

 Notes:

- To enable the **Show Fields** command, select either the **Site Server** node, a **Plant** node or a **Project** node.

- **Show Fields** settings for the site server are stored in the SiteShowFields.cfg file in the C:\Documents and Settings*user profile* folder. This file contains all **Show Fields** settings except for the plant hierarchy information, which is stored in the plant schema.

- Projects use the **Show Fields** definitions of the parent plant.

User Access Rights: An Overview

Users can have access privileges that vary from one plant to another in the same site. These rights are defined by categories. The following table details the categories and rights exposed through SmartPlant Engineering Manager for controlling user access
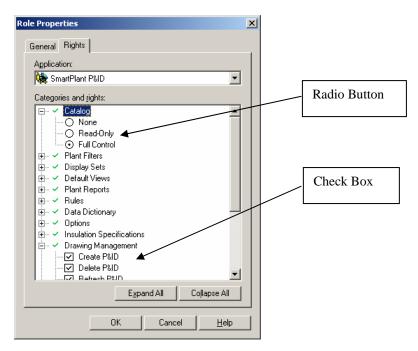
privileges on a per-plant basis. Categories with radio-button options indicate that the rights contained within are mutually exclusive, meaning you can choose only one right in that category to apply to the role. In other categories, you can choose multiple rights, as denoted by check boxes.

Mutually-Exclusive Rights

- **None** - The user is not allowed to execute the application or utility for this plant structure.

- **Read-Only** - The user can execute the application or utility for this plant structure to view the data held within it.

- **Modify Settings** - The user can execute the application or utility for this plant structure to view the data held within it and to modify any custom settings.

- **Full Control** - The user can execute the application or utility for this plant structure and perform all commands and modifications.

Important
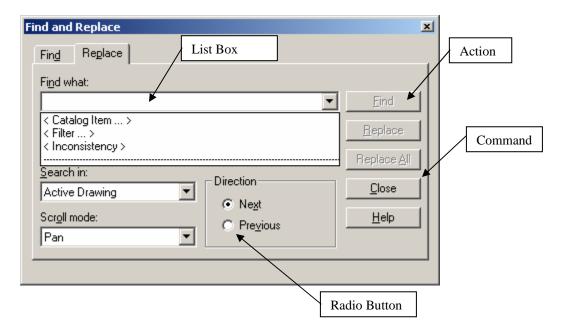
The **Full Control** right is not available to a satellite site when operating in the Workshare mode because the reference data must be controlled by the host site.

Dialog Boxes

Dialog boxes are another type of menu available and represent the most detailed method of interaction between you and the software application.

The below dialog box is from the Edit > Replace command within the SmartPlant P&ID environment.

Ribbons
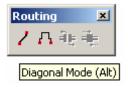
Ribbons help you control various settings for the active command and are commonly found within the SmartPlant P&ID program. You can use the same ribbon for creating an item or modifying it. For example, if you place a pipe run or select an existing pipe run to modify it, the same ribbon is active. The options on a ribbon work like options on dialog boxes. Any options that you set affect the active command.

The below Line Routing Ribbon appears when you select a line element from the **Catalog Explorer** list view or the **Engineering Data Editor** or when you select an existing line in a drawing. You can dock the **Line Routing** ribbon in the main toolbar; it remains there for the current design session unless you move it again.
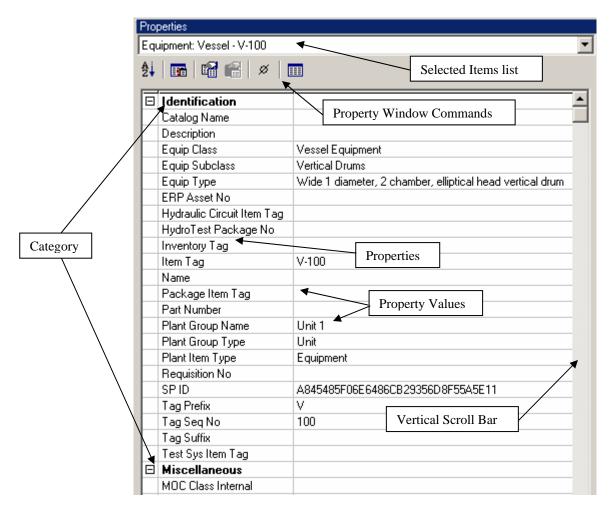
Tool Tips

Tool Tips help you find toolbar command names. When you point to a command button on the toolbar, a yellow label displays the command's name.

Properties Window

When you select an item in a plant model, you can enter, view, and modify the properties associated with that item in the **Properties** window. Your system administrator can create custom properties and their default values for each property in the **Data Dictionary Manager.**

The **Properties** window display is a two-column table that provides the common properties of the current selection in the active **Design** window. The active items always determine the content of the **Properties** window. You can only have one **Properties** window open. If only one component is selected, or all components are of the same type, all the properties of that type of component appear. If your select set contains components of different types, then only the common properties appear. If the selected items have different values for their common properties, the value box is blank.

You can modify the values for properties in the **Properties** window. You can display and edit relationships that involve components, component types, or assemblies.

At the top of the **Properties** window is the **Selected Items** list, which itemizes all of the selected objects individually and as a select set. You can display item properties by locating the appropriate item in the **Selected Items** list, as well as by selecting an item in the **Design** window. If more than one item is selected in the **Design** window, then the **Selected Items** list contains the choice **Select Set** as well as individual items in the set.

You can use the **Copy Bulk Properties** and **Paste Bulk Properties** buttons to copy properties from one item to another. This action can be particularly useful when correcting inconsistencies, for instance, or when you must apply changes to more than one item.

Catalog Explorer

Catalog Explorer is an interface to view and manipulate an electronic catalog containing drawing symbols and their associated properties. Conceptually, the **Catalog Explorer** resembles a manufacturer catalog with pictures of parts to build a plant. The paper catalog contains many items arranged in sections; you can search for items or browse through the catalog and bookmark certain pages and sections. **Catalog Explorer** replaces the paper catalog with an electronic equivalent. You can browse the catalog tree view for drawing items that you need and create bookmarks for important categories.

Catalog Explorer consists of two main views: the **Tree** view and the **List** view. In the **Tree** view, you can view the nodes of the catalog file system and open or close the nodes to see the contents. In the **List** view, you see the contents of the opened node. You also use the **List** view to select catalog items before placing them in a drawing or in a stockpile. You can create a navigation bar in the region between the tree and list views for shortcuts to places throughout the catalog.

Toolbar buttons carry out commands and allow you to customize the tree and list views to suit your work session. These toolbar buttons are located at the top of the tree view.

Catalog

Toolbar Buttons

My Catalog
Symbols
  Assemblies
    Equipment
    Instrumentation
    Piping
  Design
  Equipment
  Equipment Components
  Implied Components
  Instrumentation
  Piping
    Fire & Safety
    Fittings
    Gaps
    Labels - Piping Component
    Labels - Piping Segments
    Macrocomponents
    Piping OPC's
    Routing
    Segment Breaks
    Specialty Components
    Valves
      2 Way Common

Nodes

Expansion Icon

Vertical Scroll Bar

Collapse Icon

Ball Valve
Butterfly Valve
Check Valve
Diaphragm Valve
Gate Valve
Globe Valve
Needle Valve
Plug Valve

Horizontal Scroll Bar

Catalog Items

Engineering Data Editor

The **Engineering Data Editor (EDE)** uses a grid or tabular format to display the query results for the database. The **Engineering Data Editor** is automatically displayed in the lower left of the main window, and you can also open the **Engineering Data Editor** to appear in the **Design** window.

> 📝 **Notes:**
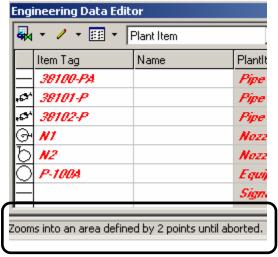
- The maximum number of items that can be displayed in the **Engineering Data Editor** is 65,537. If your database query results in more than this number, the **Query Results** dialog box opens. You can then choose to refine your query in order to generate fewer items or to enter report-only mode and run a report on your query results.

- The **EDE** is where you view items in the Stockpile, a drawing stockpile, the active drawing or other drawings.

| | Toolbar | | Item Type | Stockpile | Asking Filter | Count of Items |

Engineering Data Editor

Plant Item — Items found: 32 of 32

	Item Tag	Name	PlantItem Type	Parent Item Tag	Plant Group Name
	N1		Nozzle		
	N2		Nozzle		
	P-101A		Equipment		Unit 1
	UC1100-P		Pipe Run		Unit 1
	UC1101-P		Pipe Run		Unit 1
	UC1102-P		Pipe Run		Unit 1
	UC1103-P		Pipe Run		Unit 1
	UC1104-P		Pipe Run		Unit 1

Property values of Items in Database based on Item Type selected and Stockpile command(s)

Prompt/Status Bar

The Prompt/Status bar displays at the bottom of the main window. When you point at the command button on the toolbar, the description appears in the prompt/status bar at the bottom of the window.

The below image displays the prompt/status bar when you have pointed at the Window Area command button.
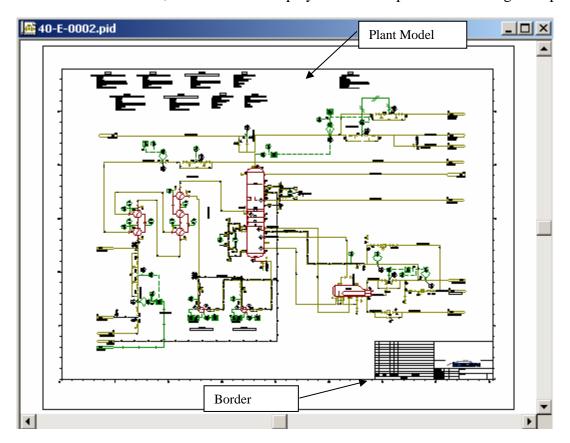
Also, prompts that explain what to do at each step of the command appear in the status bar, and messages about the operations the software is performing appear in the status bar.

The below image displays the prompt bar when you have selected a valve form the Catalog Explorer to place in the Design Window.

Design Window

The **Design** window displays Drawing views and the **Engineering Data Editor**. Each of these views provides a way for you to enter information for the plant model into the database. The two views also provide a way for you to view information in the database in different ways. In the Drawing view, you can see the traditional diagram of a plant containing equipment, instrumentation, and piping in a graphical representation. The **Engineering Data Editor** allows you to see the same information in a tabular format, and it can also display items in the plant and drawing stockpiles.

The **Design** window itself consists of several parts. The **Design** window, or Modeler, displays the model information for a design in a **Drawing** or **Engineering Data Editor** view. You can open as many windows as needed to support a design. Each window has its own independent set of properties defining view range, display properties, and so forth.

You create your drawing In the **Drawing** view, using vertical and horizontal scroll bars as well as **Zoom** and **Pan** commands to change the center point for the **Drawing** view. The title bar identifies the drawing name.

You can define one or more **Engineering Data Editors** for a drawing, too. The **Engineering Data Editor** displays drawing information in a table, much like a

Properties window. In the **Engineering Data Editor**, you can specify filters and layouts to define the table information and even edit some item properties, too.

By using commands on the **Window** menu, you can cascade or tile views to more easily navigate among them.

SmartPlant P&ID provides multiple views of a central, unified data structure that represents the plant model. A view is a visual presentation of the data that composes the plant model and can be a schematic drawing or a table. The plant model is the computer representation of the conceptual design in its entirety, including all plant components and their relationships. By manipulating model views, you can organize the information within the plant model to better understand and maintain the data.

Notes:

- The software continually updates the database as you edit a drawing; however, the software does not update the drawing file until you actually save the file (when you click **File > Save** or when you quit the program). Occasionally, the software can end in an abnormal way, for example, due to power outage. As a result, the database is up-to-date with changes that you posted to the drawing, but the drawing file reflects the status when you last saved the drawing. When you re-open the drawing, the software recognizes that the drawing file is different from the database and displays a message: **One or more items in this drawing are inconsistent with the database**. Click **OK** to re-create the drawing from the database.

Creating Plant Groups

Plant Groups Node: An Overview

The **Plant Groups** node displays the actual physical components of the plant structure itself. This node shows each item created in the plant structure and displays its attributes in the **List** view.

During the creation of the plant structure we utilized Hierarchy Template 7, which has a definition of Plant Area Unit. Thus when we create a new Plant Group within this plant structure we are prompted to create a new Area and then a new Unit. Once a plant group has been created the **List** view will display the properties values.

Create a New Item in a Plant Group

1. Select the Plant Groups node from the tree view.

2. Right-click the Plant Groups node, and select the New command or select File > New. The actual command name changes based on the hierarchy defined for your plant.

3. Provide the information requested on the New dialog box.

4. Select OK.

Notes:

- An asterisk (*) at the end of an item name indicates a value is required for that item.

- The name length is limited to 240 characters. A folder with this name cannot already exist in the plant structure folder.

- The length of the description is limited to 240 characters.

- The unit code is used as an alpha-numeric identification field in all tags in all drawings created in the unit. The length of the unit code is limited to 40 characters. This limit is 3 characters if you plan to use the Piping Data Transfer to PDS 3D process. This code cannot be changed after the new unit is created.

- The path displays the location where files in this plant group are stored. The software automatically appends the value in the **Name** box to this path and creates a folder using this name in your plant storage location. This field is limited to 255 characters.

- You can use a space character in the unit name but not any of the following characters: ~ ! @ # $ % ^ & * ( ) - + = < > . , ? / \ | [ ] { } ' " : ;

- If a plant group, in the tree view, has no drawings or plant items that belong to it, you can delete that item in SmartPlant Engineering Manager.

- The **Path** field cannot contain any spaces if you plan to use the **Piping Data Transfer to PDS 3D** process.

- In a Workshare collaboration, new plant groups cannot be created by standalone satellite sites or by the satellite sites in a project.

Revise Plant Group Item – Description Property

You can also display or edit the properties for the selected plant group item in the plant by selecting the item in the Tree view, then right-clicking and selecting Properties. The title of this dialog box and the fields displayed change based on the hierarchy defined for your plant and the level at which you are viewing the properties.

1. Select the **Plant Group Item** under the **Plant Group** node from the tree view.

2. Right-click the item selected and select **Properties** command or select **Edit** > **Properties**

3. Type or change the value for the **Description** property. Notice the other properties may be read only and thus greyed out

4. Select **OK**.

Revise Plant Group Item – Name Property
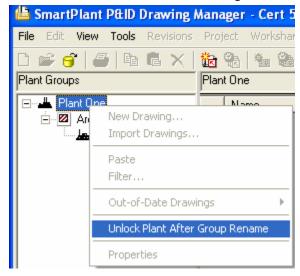
- Changes to plant groups will be seen the next time **Drawing Manager** is opened. If you do not re-initialize **Drawing Manager** and open a drawing for which a Plant Group has been modified the drawing will be recreated.

- P&IDs will have to be updated via **Global Validation** command in **Drawing Manager** to update Item Tags, including any labels which utilize the Item Tag property. After running **Global Validation** the drawing will be out-of-date for **Model Items**, requirement is to **Update Drawings** after **Global Validation** has been run.

  - Utilize **Update Drawings** to update any existing labels, which do not include the Item Tag property, that currently utilizes the **Name** property. NOTE: Requires opening the symbol(s) in **Catalog Manager** to force the drawing Out-of-Date.

    OR

- Utilize the **Update Labels** (*UpdateLabelsCmd.dll*) to update any existing labels that currently utilizes the **Name** property.

- After a rename of plant group in SPEM, all P&ID users are locked from the drawings either in **Drawing Manager** or **SmartPlant P&ID**

- Only permissions of Full Control with Options Manager will be able to unlock drawings with the **Unlock Plant After Group Rename** command on the shortcut menu.

- **Connected Workshare** – First, the rename can occur at either the Host or a Satellite. The rename requires ownership of the plant group, therefore any Plant Group that a satellite has been given ownership to may be renamed by that satellite.

  Caution: the rename will lock the plant structure for all locations in a connected Workshare plant. While this will cause some consternation in some situations the answer is simply 'communication' between workshare sites. Clearly if a satellite site had a reason to rename the plant group they have ownership to, it should be a reason that is known to more than just that satellite since in the end all sites work from the same plant structure.

When the plant structure is renamed it will be the sole responsibility of the Host site to Unlock the plant after Group Rename (the name of the command within Drawing Manager that requires someone with Full Control access to Options Manager). Since Full Control is required and since, by design, satellite sites may not have any Options Manager access right above 'Modify Settings', no user at a satellite will be permitted to unlock the plant structure. Again, communication amongst the sites should keep this being a problem.

Minor item note. When renaming via the properties dialog of the plant group, a site without ownership of the plant group will be given the appearance that the rename is occurring since they will see the path field change to reflect the changes they are entering in the Name field. When they select the OK button on the properties dialog the changes are not saved if they do not have ownership.

- **Standalone Workshare** – As with other aspects of standalone workshare there are some differences and some will be seen as restrictions over connected.

  First, standalone satellites do not have the ability to own a plant group therefore they do not have any ability to rename any plant groups.

  Second, since the communication of plant structure changes between the host and satellite is via the Synchronize Reference Data, this will also be the mechanism used for locking and unlocking the plant structure after renames. If a plant group is renamed at the host it will immediately lock the plant structure of the P&IDs at the host but, without the dblink, standalone satellites will be unaware and unaffected.

  If the Host site generates a 'Publish Synchronize Package' after a rename but before the plant structure has been unlocked at the host, the Synchronize Package will contain the 'lock plant structure' command and when retrieved at the satellite site their plant drawings will be locked.

  If the Host generates the Synchronize Package after the plant structure has been unlocked in Drawing Manager, when the Satellite site retrieves that package their plant groups will renamed AND their drawings will be unlocked. This means the satellite site administrator for standalone workshare will not have a period between the rename and the unlocking in which to update the drawings with any item tag validation. Again, communication between the sites should be seen as the best way to avoid any problems.

- **Projects** – rename of Plant Groups is permitted only at the As-Built and locks the As-Built and all Projects.

Delete a Plant Group Item

1. In the **Tree** view, select the plant group you want to delete.

2. Right-click and select the **Delete** command or select **Edit > Delete** or select ☒ from the toolbar

Important

- If the plant group you want to delete resides in the Plant and is shared with a project, the plant group must be deleted from the project before you can delete it from the Plant.

- To delete a plant group that resides in a plant or a plant/project environment, you must remove all drawings from the plant group before it can be deleted. You cannot delete a plant group that contains any drawings that are fetched, checked out, or residing in the project.

- If, after deleting the drawings in the plant group, you still cannot delete the plant group, check the plant stockpile for items still related to the plant group. These items must also be deleted or their association with the plant group removed use SmartPlant P&ID before you can delete the plant group.

- Deleting a plant group cannot be undone. If you have backed up your plant structure, you can use the **Restore** command to retrieve the backed up version of the plant structure.

Drawing Manager
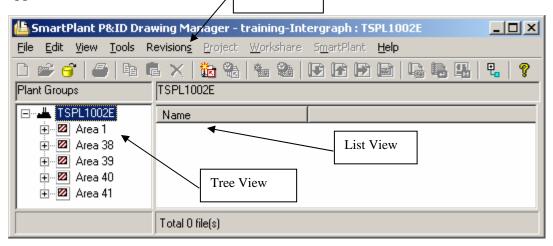
SmartPlant® P&ID Drawing Manager handles functionality having to do with the drawing files in SmartPlant P&ID. You do not modify the designs themselves in Drawing Manager, but you do create, open, modify, and delete drawings and drawing properties. Drawing Manager is also the interface for printing multiple drawings and for upgrading drawings to the current version of SmartPlant P&ID.

Drawing Manager includes versioning tools for creating, comparing, and recovering deleted drawing versions. These operations are carried out with commands on the Versions menu.

The Workshare capability of SmartPlant P&ID is carried out in Drawing Manager. These commands are on the Workshare menu in Drawing Manager. The actual host and satellite sites are specified and configured in SmartPlant Engineering Manager.

Navigating in Drawing Manager: An Overview

The main Drawing Manager interface is split into two main panes: the **Tree** view, where plant groups are arranged in nodes, and the **List** view, where lists of drawings appear.

Commands

List View

Tree View

You can control the display in the **List** view in many ways: add a filter so that only drawings that match specific criteria are displayed, display all drawings that reside under the selected node in the **Tree** view, and specify which and in what order drawing properties appear in the **List** view. The following commands are needed to carry out these activities.

Open Database - Used to open a site so that you can access a plant or project and its related drawings.

Customize Current View - Allows you to specify the drawing properties that you want to appear in the **List** view.

Include Subnodes - Controls the depth of display in the **Tree** view.

Select All - Selects all of the drawings in the **List** view.

Filter - Allows you to filter out drawings that you do not want displayed in the **List** view.

Clear Filter - Used to clear any filters applied to the **List** view.

Properties - Displays properties related to the selected plant group or drawing.

Exit - Used to close Drawing Manager.

Notes:
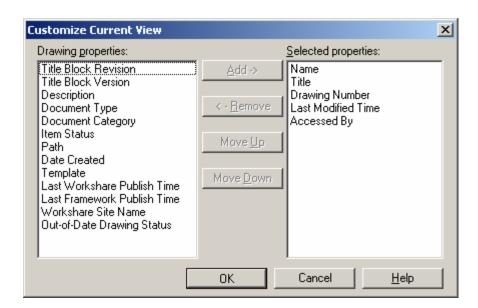
- You can drag-and-drop drawings from one plant group to another plant group, providing the P&IDs are in the same plant structure and the **Allow P&ID Drawings** option has been enabled in SmartPlant Engineering Manager.

Modifying User Interface Properties

You can control the display in the list view many ways. You can add a filter so that only drawings that match specific criteria are displayed. You can display all drawings that reside under the selected node and customize what drawing properties are displayed in the list view and in what order.

View > Customize Current View

The **View** > **Customize Current View** command sets options for the information that is displayed in the **List** view of Drawing Manager. This dialog box opens when you click **View > Customize Current View** on the main command bar

Drawing properties - Lists the drawing properties that are available for display in the **List** view. Select a property from this list and click **Add** in order to move it to the **Selected properties** list.

Notes:

- The Drawing properties property, Accessed By, requires you to have Administrator or Power Users privileges on the machine where the drawing files reside. If you do not have Administrator or Power Users privileges (in other words, you are a member of the Users group), the Accessed By field will not populate.

Selected properties - List the properties that will display in the **List** view.

Add - Moves the property into the **Selected properties** list so that the selected information is displayed in the **List** view.
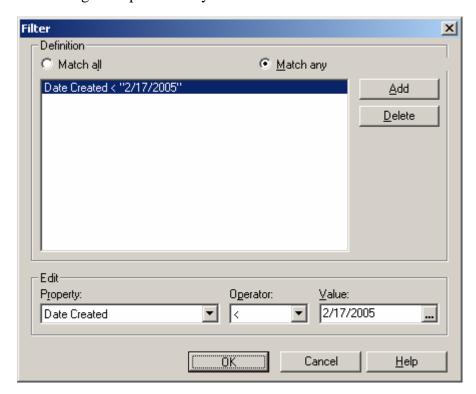
Remove - Moves the selected property back into the **Drawing properties** list. That information will no longer display in the **List** view.

Move Up - Moves the property you select in the **Selected properties** list up one position. Use this button to further customize the way the columns in the **List** view are displayed.

Move Down - Moves the property that you select in the **Selected Properties** list down one position. Use this button to further customize the way the columns in the **List** view are displayed.

View > Filter

The **View** > **Filter** command sets options for the display of drawings in the **List** view. This dialog box opens when you click **View > Filter**.

Definition - Displays all defined criteria associated with a filter. To add to or modify the definition list, you must select a line in the list and then define or edit the property in the **Edit** group.

    Match all - Specifies that items matching ALL of the filtering criteria pass through the filter.

    Match any - (Default) Specifies that items matching any one or more of the filtering criteria pass through the filter.

    Add - Places a new entry at the end of the existing definition list and enables the options in the **Edit** group so that you can edit the new entry.

    Delete - Removes the selected criterion from the definition list. This button is available only when you select a criterion in the definition list.

Edit - Allows you to define or edit a single line of filter definition criteria.

    Property - Displays a list of all properties for a certain item type. Examples of properties include revision number and name. You define or modify filtering criteria by selecting a property, an operator, and a value.

Operator - Specifies the relationship between the property and its value. These relationships include greater than, >; equal to, =; not equal to, <>; and so forth.

Value - Lists appropriate values for the property specified in the **Property** list. If a list of attributes is not already associated with the **Value** box, you must choose null or type a value, which can be free text. You can type a percent sign, **%**, as a wildcard character to find multiple characters, or type a question mark, **?**, as a wildcard character for a single character. Do not use an asterisk, *****, in the **Value** box.

📝 **Notes:**

- View filters in Drawing Manager are strictly established only for the specific case at hand; you cannot save a filter setting.

View > Clear Filter

The **View** > **Clear Filter** command removes a filter from the view and displays all drawings. This command is available only if you have applied a filter to the view.

View > Include Subnodes

The **View > Include Subnodes** command displays in the **List** view all of the drawings that reside under the selected plant in the **Tree** view, not only those drawings that reside directly under the selected node.

📝 **Notes:**

- If you select a plant in the **Tree** view and then click **View > Include Subnodes**, the entire plant hierarchy is displayed in the **List** view. This command provides an easy way of viewing the plant hierarchy without having to open every Plant Group.

Working with Drawings: An Overview

Drawing Manager enables you to manage your process and instrumentation drawings by allowing you to create new drawings in your plant structure, open and view drawings, and delete drawings. The following commands are needed to carry out these activities.

New - Used to create a new drawing.

Open - Opens a drawing in SmartPlant P&ID. Drawing Manager checks to make sure that you have the appropriate permissions for opening or modifying drawings.

Copy - Used to copy one or more drawings from within the same plant or same project. The copied drawing(s) can then be placed using the Paste command. To copy a drawing from one plant to another plant, refer to the Import Drawing command. Any graphics that have been band-aided should be deleted and replaced prior to using this command.

Paste - Places a copy of a drawing in the selected plant or project. Any graphics that have been band-aided should be deleted and replaced prior to using this command.

Show History - Displays the version history of a drawing, provides access to the **Compare** and **Compare With** commands for viewing changes between drawing versions, and the **View** command, which allows you to view a drawing as read-only without opening SmartPlant P&ID.

Delete - Used to remove drawings from the plant.

File > New Drawing

The **File** > **New Drawing** command opens the **New Drawing** dialog box where you specify properties for a new drawing. The drawing is created under the currently selected plant group node in the **Tree** view.
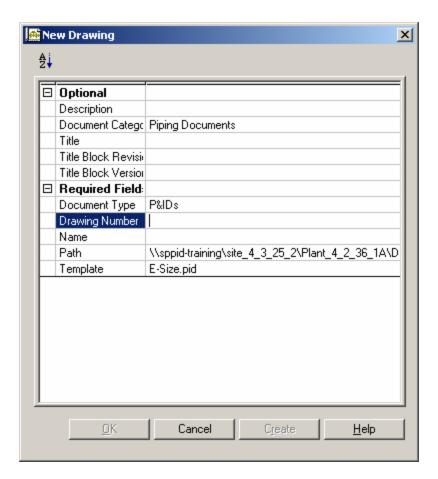
📝 Notes:

- You must have the appropriate permissions, specified in SmartPlant Engineering Manager, in order to be able to create drawings.

The new drawing dialog box sets property values for the new drawing being created. The drawing will be created under the plant group node that you select in the **Tree** view. This dialog box opens when you click **File > New Drawing**.

📝 Notes:

- The **Drawing Number**, **Name**, & **Template** properties on a drawing are always considered as Read/Write by **Drawing Manager** even when set to Read Only within the **Data Dictionary Manager**. This is because these properties must be set when creating a drawing.

- You can govern which properties appear within the **Optional** or **Required** category through **Options Manager** > **Settings**.

  o Drawing Properties – Optional

  o Drawing Properties - Required

Alphabetized Button - Displays drawing properties alphabetically. This button acts as a toggle and is available when properties are displayed categorically.

Categorized Button - Displays drawing properties in categories. This button acts as a toggle and is available when properties are displayed alphabetically.

OK - Creates the new drawing, adds it to the selected plant node, and closes the **New Drawing** dialog box. After entering the Drawing Number of Name you must select or tab to another field in order for the OK command to be enabled.

Cancel - Closes the **New Drawing** dialog box without creating a new drawing.

Create - Creates a new drawing and adds it to the plant node. The **New Drawing** dialog box will remain open. This button is available only after you enter values for the **Required Fields** in the list of drawing properties. After entering the Drawing Number of Name you must select or tab to another field in order for the OK command to be enabled.

Edit > Copy

The **Edit > Copy** command is used to copy one or more drawings from within the same plant or same project. The copied drawing(s) can then be placed using the **Edit > Paste** command. To copy a drawing from one plant to another plant, refer to the Import Drawing command.

 Notes:

- You must have the appropriate permissions, specified in SmartPlant Engineering Manager, in order to copy drawings.

- You can hold **Ctrl**, select a drawing, and drag it to a new location to make a copy. You can also drag it to the current list view to make a copy. If you drag and drop the drawing to another unit, you will need to select the contents of the drawing using SmartPlant P&ID and set the **Plant Group Name** value in **Property** grid.

- A multi-rep model item is created at the target only once if the drawings that contain all the representations for it are selected for copy in one session. If the drawings are copied in separate sessions, the model item is re-created at the target for that session.

- Paired OPCs in a drawing that are not copied (for example, not in a select set) are placed in the plant stockpile. Paired OPCs in a copied drawing have their relationships maintained by the copy. Paired OPCs are not moved from the plant stockpile to a drawing by a subsequent copy session.

- A plant group item is created at the target only once if the drawings that contains all its members are selected for copy in one session. If the member drawings are copied in separate sessions, the plant item group is re-created at the target for each session.

Edit > Paste

The **Edit > Paste** command pastes the selected drawing(s) in Drawing Manager.

 Notes:
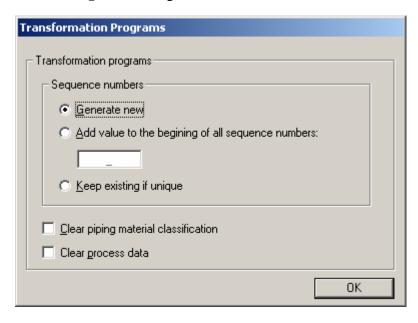
You can change the name of a pasted drawing by selecting the drawing and then select **Edit > Properties**.
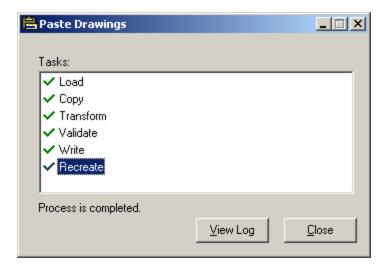
Paste a Drawing

1. In the **Tree** view, select the node where the copy of the drawing will reside.

2. In the **List** view, select the location that you want to paste the copy of the drawing.

3. Click **Paste** on the main toolbar. The system displays the **Transformation Programs** dialog box.

4. Click to select the **Generate new sequence numbers** if you would like new sequence numbers to be created. Selecting the option causes existing tag sequence numbers to be set to null for any item that has the property **TagSequenceNo**. The null sequence number triggers the creation of new sequence numbers during normal item tag validation. An exception to this is that the tag sequence number for an instrument item is not set to null. Item tag validation does not generate a new sequence number for an instrument. Instrument tags will be duplicated at the target.

5. Click to select the **Add value to the beginning of all sequence numbers** to use this option. Click in the provided field and enter the values you want to display at the beginning of all your sequence numbers.

6. Click to select **Keep exiting if unique** to use this option.

7. Click to place a check mark in the **Clear piping material classification** check box to use this option. Selecting this option removes any defined piping material class.

8. Click to place a check mark in the **Clear process data** check box to turn on this option. Any values in the Process category will be removed.

9. Click **OK**. The system displays the **Paste Drawings** dialog box.

10. When the processes complete, click **View Log** to view the report or click **Close** to dismiss the Paste Drawings dialog box.

📝 **Notes:**

- You must have the appropriate permissions, specified in SmartPlant Engineering Manager, in order to paste drawings.

- For more information about creating a transformation program, see *Customizing the Sample Projects* in the *SmartPlant P&ID Programmer's Guide*

Edit > Properties

The **Edit > Properties** command opens the **Properties** dialog box, which displays the plant group properties or drawing properties based on the item selected in the **Tree** or **List** view.

Plant group properties are read-only. You can modify some of the drawing properties, such as the name, version, title, description, and so forth.

Modify Drawing Properties

1. In the list view, select the drawing whose properties you want to modify.

2. Click **Edit > Properties**.

3. On the **Drawing Properties** dialog box, change or add drawing properties.

4. Click **OK**.

Notes:

- You can rename and renumber a drawing, but keep in mind possible naming conflicts that can arise when you retrieve a drawing under an older name. In particular, a drawing that has been renamed appears in the list of deleted drawings when you use the **Tools > Recovery > Retrieve Deleted Drawings** command even though it has not actually been deleted.

- You cannot modify many of the drawing properties after the drawing is created.

Move a Drawing

1. In the **Tree** view, select the node in which the drawing resides.

2. In the **List** view, select the drawing that you want to move.

3. Drag and drop the selected drawing to the new location.

Notes:

- If you drag and drop the drawing, which contains data, to another plant group, you will need to change the **Plant Group Name** for those items utilizing the **Engineering Data Editor** or the **Property Grid** in SmartPlant P&ID.

Edit > Delete

The **Edit > Delete** allows you to delete drawings from the **List** view and from the plant structure. Because a new version of the drawing is automatically created when you delete a drawing, you can recover the drawing after you have deleted it, but there are many drawing recovery ramifications to consider, see the Recovering Drawings: An Overview for more information on ramifications.

Of course, you must have the appropriate permissions, specified in SmartPlant Engineering Manager, in order to be able to delete a drawing.

Delete a Drawing

1. In the **List** view, select the drawing that you want to delete.

2. Click **Delete**.

3. To confirm the drawing deletion, click **Yes** on the message box.

4. Click **View Log** on the **Deleting Drawings** dialog box to review notes from the drawing deletion process.

🖉 **Notes:**

- If you have a saved version of a deleted drawing, you can retrieve the drawing as it was when you saved it. For more information, see Recover a Version of a Deleted Drawing.

- If a plant group has no drawings or plant items belonging to it, you can delete that hierarchy item in SmartPlant Engineering Manager. Keep in mind, though, that if you have associated a plant item with a hierarchy item by using the **Properties** window in SmartPlant P&ID, then even though it can look as if no drawings are associated with that plant group, you cannot delete that hierarchy item in SmartPlant Engineering Manager.

- You can also delete saved versions of a drawing, without deleting the drawing itself. For more information, see Show the History of a Drawing and Its Versions.

Opening Drawings

You can open drawings from either Drawing Manager or from SmartPlant P&ID.

File > Open Drawing

The **File** > **Open Drawing** command opens the selected drawings in SmartPlant P&ID.

Open a Drawing from Drawing Manager

1. In the tree view, select the node where the drawing resides.

2. In the list view, click the drawing you want to open.

3. Click **File** > **Open Drawing**.

4. View or modify your drawing in SmartPlant P&ID.

🖉 **Notes:**

- You may also use the **Open Drawing** command on the right-click menu. Or you can double-click a drawing in the list view.

- You must have the appropriate permissions, specified in SmartPlant Engineering Manager, in order to view or modify drawings.

Open a Drawing in a Different Site from Drawing manager

1. On the pull-down menu, click **File** > **Open Database**.

2. On the **Open Plant Structure** dialog box, click **Site Server**.

3. On the **Open Site Server** dialog box, select the correct SmartPlant Initialization file and click **Open**.

4. Select the correct plant on the **Open Plant Structure** dialog box and click **Open**.

5. Navigate to the correct drawing in the list view and select **File** > **Open Drawing**.

Open a Drawing in a Different Plant, same site from Drawing manager

1. On the pull-down menu, click **File** > **Open Database**.

2. On the **Open Plant Structure** dialog box, select the correct plant and select **Open**.

3. Navigate to the correct drawing in the list view and select **File** > **Open Drawing**.

Open Drawings from SmartPlant P&ID

1. Open SmartPlant P&ID by clicking **Start** > **Programs** > **Intergraph SmartPlant P&ID** > **SmartPlant P&ID**.

2. Click **File** > **Open**. The **Open** dialog box appears.

3. In the list, click a plant group to display the drawings it contains.

4. Click a drawing in the list, and then click **OK**. SmartPlant P&ID opens and displays the selected drawing.

Open a Drawing in a Different Site from SmartPlant P&ID

1. On the pull-down menu, click **File** > **Open**.

2. On the **Open** dialog box, click the **Open Database** button.

3. On the **Open Plant Structure** dialog box, click **Site Server**.

4. On the **Open Site Server** dialog box, select the correct SmartPlant initialization file and click **Open**.

5. Select the correct plant on the **Open Plant Structure** dialog box and click **Open**.

6. Navigate to the correct drawing in the tree view and select **OK**.

Open a Drawing in a Different Plant, same site from SmartPlant P&ID

1. On the pull-down menu, click **File** > **Open**.

2. On the **Open** dialog box, click the **Open Database** button.

3. On the **Open Plant Structure** dialog box, select the correct plant and select **Open**.

4. Navigate to the correct drawing in the tree view and select **OK**.

Customizing the SmartPlant P&ID Drawing Environment

SmartPlant P&ID's unique components work together to help make creating and modifying P&IDs an easy task. The design window supplies a schematic representation of the drawing. The **Stockpile** contains items that do not exist in the P&ID but exist in the model. The **Catalog Explorer** provides the items that you can insert in the drawing or model. The **Properties** window organizes the characteristics of the current select set in spreadsheet style. And toolbars contain shortcuts to the most commonly used commands.

As you work in SmartPlant P&ID, you can set various options for each of these components to streamline your workflow. You can add buttons to toolbars, move toolbars, and customize various windows in SmartPlant P&ID to meet your needs. These customization features allow you to have your favorite, most-used options available for immediate selection.

Arranging Work Space

- You can arrange the workspace in several ways:

- Move windows around on the screen.

- Display, hide, or customize toolbars using the Toolbars command on the View menu or the buttons on the main toolbar.

- Move toolbars by dragging them with your mouse.

- Move a ribbon to the top or bottom of the Drawing view.

- Set viewing options for the window by clicking Options on the Tools menu and setting the options you want on the dialog box.

Using the View Menu

On the **View** menu, you can select a variety of commands for zooming in and out on items in the **Drawing** view. You can also right-click anywhere in the **Drawing** view to access many of these commands.

View > Previous

The **View** > **Previous command** restores a Drawing View.
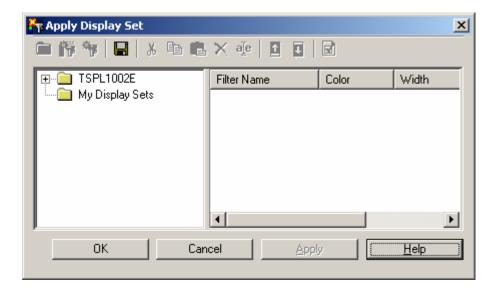
Notes:

- To stop the repainting of items in the window, press **Esc**. This action is convenient for large drawings.

- To refresh the window, press **F5**.

View > Apply Display Set

The **View** > **Apply Display Set** dialog box specifies which symbols display in the Drawing View. You can create filters that define which symbols you want to display. Click **View > Apply Display Set** to open this dialog box. You can save these filters and share them with other users working in the same Plant. You can also define colors and widths for display items. If a display set is currently active, its name displays in the lower right-hand corner of the product.

To remove the results displayed by selecting the **Apply Display Set** command, select **View > Clear Display Set**.

Add Folder- Adds an empty folder to the Tree. You can use **Rename** to define a name for the folder. These folders can be used to organize display sets.

Add Display Set - Creates a new display set. You can use **Rename** to define a name for the new display set.

Add Filter - Displays the **Select Filter** dialog box. Select any displayed filter to add it to the current display set.

Save - Saves the selected display set.

Cut - Removes the selected display set and places it in memory.

Copy - Copies the selected display set into memory.

Paste - Pastes any values that are currently stored in memory.

Delete - Deletes the selected item.

Rename - Allows you to click on an item in the Tree and rename it.

Move Up - Allows you to move the selected filter name up in the list.

Move Down - Allows you to move the selected filter name down in the list.

Properties - Displays the properties of the filter.

Filter Name - Displays a list of filter names. These filters are defined using the Add Filter command.

Color - Displays the color of the selected item for your display set. You can click in the Color field to display a color palette and then select any color to define a new color for any selected item(s). The default color square is defined with the crosshatch pattern.

Width - Displays the selected display width of items defined in the selected filter. A wider display width would cause the item to be more visible.

View > Zoom Area

Allows you to zoom in on the Drawing view using the pointer to fence the area that you want to zoom in on. The selection then fills the view.

View > Zoom In

Allows you to zoom in on the Drawing view using the pointer. The selection then fills the view.

View > Zoom Out

Allows you to zoom out from an area on the Drawing view using the pointer that reduces the display of items around a specified point in the active window.

View > Fit

Fits all visible items in the active view.

View > Pan

Allows you to move the display in any direction from a specific point in a drawing to see other areas of the drawing by dragging the pointer across the view.

View > Show Grid

Displays a grid so that you can place items with precision. The grid is not considered part of the drawing and does not print.

Notes: You can change the grid display from static to dynamic by setting options on the **Grid** tab of the **View Properties** dialog box, which opens when you click **View > Properties**.

View > Snap Grid

Aligns items with the grid. The grid is a set of lines in the drawing background that helps you align items. When you set the **Snap grid** option, items always align with the grid lines or nearest intersection of the grid lines. Grid lines do not print.

✎ **Notes:** You can change the grid display by setting options on the Grid tab of the View Properties dialog box, which opens when you click View > Properties. To view grid lines at a finer level, set the style to Dynamic and adjust the grid line width with the Density control.

View > Show Inconsistencies

When **View > Show Inconsistencies** is selected, inconsistent relationship indicators appear in the drawing.   If you do not want the inconsistencies to appear, click **View > Show Inconsistencies** again to clear the selection of the option.

✎ **Notes:** Also, you can right-click in an empty portion of the drawing in order to display a shortcut menu. Select **Show > Inconsistencies** to toggle the display of inconsistencies off and on.

View > Show Claims

When View > Show Claims is selected item designated as claimed will be displayed. Select **View > Show Claims** again to turn off the display of claim status of drawing items.

✎ **Notes:**

- This command is only available when Projects have been enabled.

- You define the line color and weight used to designate claim status on the Claims tab of the View Properties dialog box.

- If you turn on the display of claim status in the drawing, then when you print the drawing, the claim status will be plotted.

View > Display

The **View > Display** command allows you to specify the windows, such as the **Catalog Explorer, Property Window and Engineering Data Editor**; you want to see in your design session.

✎ **Notes:**

- Another method for displaying the **Catalog Explorer, Property Window or Engineering Data Editor** window is too right-click the empty area in the main toolbar and then select from the shortcut menu.

View > Toolbars

The **View** > **Toolbars command** sets options for displaying, hiding, or changing toolbars. Open this dialog box by clicking **View > Toolbars** on the main menu bar.

Toolbars - Lists the available toolbars. You can select the box next to the toolbar that you want to display, hide, or change.

Toolbar Name - Displays the name of the toolbar that you have selected.

New - Accesses the **New Toolbar** dialog box.

Customize - Adds buttons to or removes buttons from built-in toolbars with the **Customize** dialog box.

Reset - Returns the toolbar to the state that it was in at the beginning of your current design session.

Color Buttons - Adds color to the toolbar buttons. Clearing this option causes toolbar buttons to appear in black and white.

Large Buttons - Enlarges toolbar buttons so that they are easier to see.
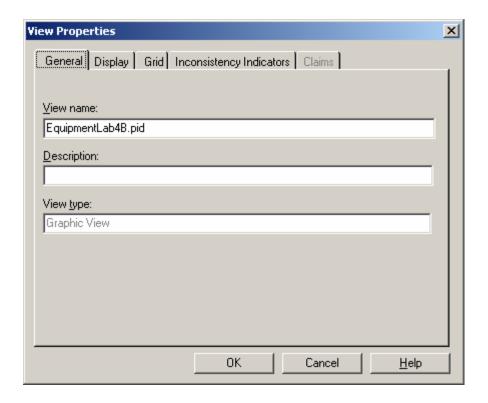
Show ToolTips - Displays on-screen descriptions of a toolbar button when the pointer pauses over one of them.

View > Properties

The **View Properties** dialog box controls the display of information in the Drawing view. By clicking **View > Properties** on the main menu bar the below dialog box will appear.

 Notes:

- You can access the **View Properties** dialog box by right-clicking in space in your drawing and selecting **Properties** from the shortcut menu.

General Tab (View Properties Dialog Box)

The **General** tab from the **View** > **Properties** command will displays miscellaneous information about a Drawing view, including the view name, description, and view type.

View name - Allows you to name the Drawing view.

Description - Allows you to enter a description of the Drawing view.

View type - Displays the type of view. This information is read-only.

Display Tab (View Properties Dialog Box)

Controls the display of items in a Drawing view. Items that you can display include labels, drawing borders, grids, notes, and inconsistency indicators. When you select the check box, the display toggles on for that item.

Do not show labels for filtered items - Prevents the display of labels in filtered views of the design.

Show - Lists options for objects you want to appear in your Drawing view.

Labels - Toggles the display of all labels on or off, in any view. If the display of labels is turned off here, then the **Do not show labels** options is not available

since all labels are no longer displayed regardless of the view tab that you are using.

Drawing borders - Toggles the display of the drawing border on or off.

Grids - Turns the grid display on or off.

Notes - Turns the display of notes on or off. Notes can contain links to a file, for example, a Microsoft Word file.

Inconsistency indicators - Turns the display of inconsistency indicators on or off.

Claims - Turns on the display of the options selected on the **Claims** tab. This toggle is only active when Projects have been enabled and created.

Select - Includes other options for the selection of items in your drawing.

Prevent selection of inserted objects - Makes inserted foreign data non-locatable in a drawing. For example, if you insert a file using **Edit > Insert Object** and do not want to accidentally select that file in the drawing, select this box. The inserted file appears gray in the drawing.

Grid Tab (View Properties Dialog Box)

Controls the behavior of the grid in the view.

View - Turns the grid display and grid snapping on or off. You do not have to display the grid in order to snap items to it.

Show grid - Turns the grid display on or off.

Snap grid - Turns the snap-to-grid feature on or off. When you select this option, items always align with the grid lines or nearest intersection of the grid lines. Grid lines are not printed.

Style - Specifies either **Static** or **Dynamic** grid style. A static grid displays solid grid lines that do not move as you zoom in or out. When you zoom in or out, the grid lines for a dynamic grid change with the level of magnification. A dynamic grid displays index lines that intersect with the darker, solid grid lines. The choice of grid style affects the option that is available in the **Spacing** or **Density** box and the availability of the **Index** options, too.

Spacing - Allows you to specify the space between the grid lines when you choose the **Static** grid style from the **Style** list.

Density - Allows you to specify fine, medium, or coarse levels when you choose the **Dynamic** grid style from the **Style** list. For a dynamic grid display the number

of index lines varies depending on the zoom level, and so you cannot change the **Index** list when using the **Dynamic** grid style.

Index - Specifies the grid index, which is the number of minor grid lines. This option is available only with the **Static** grid style.

Inconsistency Indicators Tab (View Properties Dialog Box)

The Inconsistency Indicator tab from the **View** > **Properties** dialog box controls the display of inconsistency indicators, which signify the suitability of design work that you perform while creating the drawing. The software verifies in real-time if the composition of a drawing and the underlying data model satisfy rules defined in Rule Manager. Choose the indicators and severities that you want to view in your drawing. Open the **View Properties** dialog box by clicking **View > Properties**.

Show - Lists options for objects you want to appear in your Drawing view.

Errors above severity - Turns the display of errors on or off. You can specify the severity level. For example, if you specify a severity level of five, all errors with severity of five or above appear. Errors are marked with a red **X**.

Warnings above severity - Turns the display of warnings on or off. Again, you can specify the severity level and control the warnings that you see. Warnings are marked with a blue **/** forward slash.

Approved warnings - Turns the display of approved warnings on or off. Approved warnings are marked with green check mark.

Claims Tab (View Properties Dialog Box)

The Claims tab sets options for the display of items in the Drawing view according to their claim states. This tab is part of the **View Properties** dialog box, which appears when you click **View > Properties**.

This tab is only available when Projects have been enabled and created.

Status - Displays the possible claim states and combination of claim states for drawing items.

Invalid claims - Specifies the display of invalidly claimed items in the Drawing view. Invalidly claimed items are items which differ from the Plant items and which you cannot modify and check into the Plant without first establishing a valid claim. Invalid claiming can happen only for Plants that support shared claiming of database items. If you specify a special display for invalidly claimed items in the Drawing view, then these display options override all other options designated in the **Claim** area of this dialog box.

Claim - Displays the possible claim states for all items except invalidly claimed items.

> **Not claimed** - Specifies the special display of drawing items that are not claimed at all.

> **Claimed only by others** - Specifies the special display of drawing items that are claimed solely by other projects or the Plant.
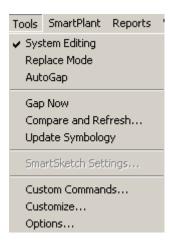
> **Claimed only by this project** - Specifies the special display of drawing items to which this project has the sole claim.

> **Shared claims** - Specifies the special display of drawing items when the active project shares a claim on with another project or the Plant. If your Plant is configured so that it honors only exclusive claiming, then these display options will have no effect in the **Drawing** view.

Colors - Allows you to select a color for items with the corresponding claim state.

Widths - Allows you to select a line width for items with the corresponding claim state.

Using the Tools Commands

Tools > System Editing

When System Editing is enabled for a plant, changed property values are automatically spread or propagated to related items according to the currently defined consistency criteria. In certain cases, you may want to set a property value on one item only and not allow it to be propagated to any related items. The System Editing command (on the Tools menu and the toolbar) allows you to turn off the propagation functionality temporarily during a design session. After the change has been made, you can use this same command to turn it back on again. System Editing is always turned on when you start a new design session.

System Editing can be enabled or disabled for the entire plant by means of the Enable System Editing setting that is exposed through Options Manager. For new plants, the value of this setting is True by default. For upgraded plants, the value of this setting is False. This allows you to enable System Editing for an upgraded plant at the time of your choosing.

To enable system editing Enabling System Editing for a design session select **Tools > System Editing**.

Tools > Replace Mode

The **Tools > Replace Mode** command controls whether catalog items are placed as substitutions for existing drawing items or placed normally. When this mode is active, the software places catalog items only when they replace an existing drawing item. When this mode is not active, the software places catalog items as usual. The **Replace Mode** command will copy properties of the current item onto the new item.

The **Replace Mode** command works for like objects only. The item that you want to replace in the drawing must be of the same class: equipment class, instrument class, and so forth. For instance, you can replace a vessel with another vessel, but you cannot replace a vessel with a pump. The software also takes into consideration the connect points that are in use on the currently placed item. This stipulation means you cannot replace an angled item for a straight item when the straight item is already connected at both ends. For example, you cannot replace a Flanged Nozzle with an Angle Parametric Nozzle if the nozzle is already connected to a pipe run, likewise for angled valves. **Replace Mode** also works on parametrics. Resizing is maintained during the replace if both symbols have an identical relative path.

The **Replace Mode** functions very much like **Edit > Replace**. However, the replace mode works for only one item at a time and is not activated from inside a dialog box.

You cannot use the **Replace Mode** for pipe runs or signal lines. You also cannot use this mode to replace items that are multiple representations. If you need to replace pipe runs or signal lines, use the the **Edit > Replace** command.

Replace an Item Using Replace Mode

1. Click **Tools > Replace Mode** or click **Replace Mode** on the main toolbar.

2. In **Catalog Explorer**, select the item that you want to place in the drawing.

📝 **Notes:**

- You cannot use this mode to replace drawing items with items from the Engineering Data Editor.

3. Move the selected catalog item over the drawing item that it is to replace.

📝 **Notes:**

- The Replace Mode command works for like objects only. The item that you want to replace in the drawing must be of the same class: equipment class, instrument class, and so forth. For instance, you can replace a vessel with another vessel, but you cannot replace a vessel with a pump.

- The software also takes into consideration the connect points that are in use on the currently placed item. This stipulation means you cannot replace an angled item for a straight item when the straight item is already connected at both ends. For example, you cannot replace a Flanged Nozzle with an Angle Parametric Nozzle if the nozzle is already connected to a pipe run, likewise for angled valves.

- If the drawing item can be replaced with the catalog item, the replace icon is displayed, and the drawing item is selected.

4. When the replace icon appears, click to replace the drawing item with the selected catalog item.

5. Continue to replace more items with your currently selected catalog item, or press **Esc** to quit placement mode.

6. Click **Tools > Replace Mode** again to turn **Replace Mode** off.

Gapping Lines: An Overview

A gap in a drawing is a condition that exists when two lines intersect graphically in the drawing but not physically in the plant. In order to portray this condition, you can specify that lines show a gap at these non-physical intersections.

You can either turn auto-gapping on, and the software gaps your lines automatically as you route them, or you can choose when the software performs gapping on your drawing by using the **Gap Now** command. Using the **Gap Now** command, instead of turning on auto-gapping, can increase the efficiency of your design session.

Symbology and priority for your line gaps is set in Options Manager.

Tools > AutoGap

The **Tools** > **AutoGap** command turns automatic gapping on or off in the drawing.

When **AutoGap** is on, the software automatically gaps drawings during the working session each time that you modify an item, when you print, or when you save a file. A progress indicator at the bottom of the window alerts you to avoid selecting another command while the software completes the gapping. The **Gap Now** command does not affect the setting for **AutoGap**.

When **AutoGap** is off, the software does not automatically gap drawings. By default, automatic gapping is off.

Tools > Gap Now

The **Tools** > **Gap Now** command will perform line gapping in your drawing at the time you select the command.

Adding a Manual Gap to a Line

1. Select the appropriate gap from the **Catalog Explorer**.

Symbols > Piping > Gaps

OR

Symbols > Instrumentation > Gaps

2. On the line, click the position for one side of the gap.

3. Then on the same line, click the position for the other side of the gap.

📝 **Notes:**

- Gaps are parametric symbols. You can resize the gap by dragging the parametric handles.

Comparing and Refreshing Versions: An Overview

When more than one version of a drawing exists, you can view two versions side-by-side and examine their differences by using the **Tools > Compare and Refresh** command. You can compare two versions from inside your own Plant or project database, or you can compare a version in your database to a version in the Plant or another project database. Keep in mind that you can compare a drawing only against a version of itself; that is, you cannot compare one drawing to another drawing.

Differences between drawing versions are assigned to logical change groups, which are listed on the **Change groups** area of the **Compare and Refresh** dialog box. Differences display in the following two categories.

- **Graphic** refers to an item that has changed only in its graphical representation in the design; that is, the item is moved or otherwise graphically manipulated in the drawing.

- **Data** refers to a mismatch in the properties assigned to an item that exists in both drawings; that is, a change, addition, or deletion of a property in the **Properties** window or **Engineering Data Editor** in SmartPlant P&ID or through automation.

Every change grouping and every changed item is assigned a category, and if more than one category applies (for instance, if you move an item and change one of its properties), then the highest priority category is displayed.

Change details displays information about selected groups in the **Change groups** area. Values for the Change details include the following.

- **Add** - Indicates that the selected group will be added to the version displaying on the right.

- **Delete** - Indicates the selected group will be removed from the version displaying on the right.

- **Modify** - Indicates the selected group will be modified on the version displaying on the right.

The two versions are displayed in two Drawing views, described as left and right views. The relationship between the two views depends on whether you are comparing two versions in your own database or comparing your version to a version in another database.

- Two versions are in the active database - The left-hand view is the older version, and the right-hand view is the newer version. That is, they are displayed in time-order from left to right.

- Two versions exist in different databases - The right-hand view is reserved for the version in your active Plant or project database, and the left-hand view belongs to the version in another database because you cannot be assured that time-order is the logical order to display the versions.

To refresh any differences between these two versions, you must set the **Action** column to **Refresh**. Change groups are marked if the changes affect items you have claimed. Be sure to refresh all of the changes that do not involve claimed items because these are the changes that have been checked in by other projects. You want to include these changes in your drawing so they will not be lost when you perform a check in.

Compare and Refresh Dialog Box

Compare Options - Displays the **Compare Options** dialog box and allows you to customize the display colors for items that differ between the two drawing being compared.

Print - Prints the entire **Compare and Refresh** dialog box contents. You can also specify to print either the right or left view by using the drop-down arrow to select either **Right View** or **Left View**.

Generate Report - Displays a report in Microsoft Excel. The report contains the details of the compared drawings.

Find in List - Highlights the line in the **Change groups** and **Change details** areas. You must first select an item in a drawing.

Find in Drawing - Zooms to the selected item. You must first select an item in either the **Change groups** or **Change details** area.

Zoom Area - Enlarges the selected area by allowing you to draw a fence around the area.

Zoom In - Enlarges the selected area where you click.

Zoom Out - Reduces the display of the selected area where you click.

Pan - Allows you to move the display in any direction by dragging the pointer across the view.

Select - Changes the pointer to an arrow allowing you to select an item.

Alphabetic or **Categorized** - Displays the properties in either an alphabetic list or by specific category.

Show Modified - Displays modified properties.

Change groups - Area that lists the changed items in groups. A listed item contains all the items this change effects.

Checkbox column - Displays colors in the drawings as defined using **Compare Options**. If not selected, the **Default items** color displays.

Number - Displays the number assigned to each change group.

Action - Defines which action is to be taken involving the selected change. Options include:

- **Validate** - Validates Invalid Claims for the selected group (no drawing changes are applied).

- **Refresh** - Applies any drawing changes to the open drawing version; and **No Action** - performs no action on the selected drawing group.

Category - Displays the category of the change. Options include:

- **Data** - Indicates that a property value has changed (for example, a property value for a vessel).

- **Graphic** - Indicates that a change has been made to an item in the drawing (for example, a vessel has been moved).

Claimed - Indicates that the items in the group have been claimed.

Valid Claim - Indicates that the claimed item is a valid claim.

Change details - Area that lists the details of the item in each change group.

Result - Defines the results of the compare. The column indicates any differences in the two versions and which version contains the change.

Change - Describes what action is required to make the drawing in the right frame match the drawing in the left frame. Actions include:

- **Add** - Adds the listed item to the version on the right.

- **Delete** - Removes the listed item from the version on the right.

- **Modify** - Changes the listed item in the version on the right.

Item Type - Describes the type of item.

Specific Item Type - Displays the specific type of item.

Category - Displays the category of the change.

- **Data** - Indicates that a property value has changed (for example, a property value for a vessel).

- **Graphic** - indicates that a change has been made to an item in the drawing (for example, a vessel has been moved).

Claimed - Indicates if an item is a valid or invalid claim.

Updating Options Manager Settings: An Overview

You can force the software to redraw the graphic representation of your data, the drawing, by using the **Update Symbology** command. This command refreshes the graphic symbology of symbols in your drawing based on the current settings in Options Manager. The **ApplySettingsCmd** macro also updates the line settings, Minimum Connector Segment and Routing Self-Avoidance, so that the current values in Options Manager are reflected in the drawing.

The symbology and other settings defined in **Options Manager** usually only take effect in those drawings that are created after those values are defined. You can force changes in Options Manager to appear in the current drawing by updating Options Manager settings, regardless of when the drawing was created.

 Notes:

- Any user can update drawings using these commands. However, check your permissions, which are assigned in SmartPlant Engineering Manager, to find out if you can make changes to the plant-wide symbology in Options Manager.

- Once you load the current plant-wide definitions into your drawing, you cannot revert to previous definitions. However, you can always override plant-wide symbology choices in your drawing by using drawing filters and choosing alternate symbology for items.

Tools > Update Symbology

1. Open a drawing.

2. Click **Tools > Update Symbology**.

 Notes:

- The line width and color symbology for your active drawing is changed to the most recent symbology defined in Options Manager. You do not have to update symbology for this drawing again, even if you end your design session, unless Options Manager symbology changes again.

- Redefining symbology in Options Manager usually only affects drawings created after the change. The **Update Symbology** command makes it possible to override this constraint.

- Once you load the current plant-wide symbology definitions into your drawing, you cannot revert to previous definitions. However, you can always override plant-wide symbology choices in your drawing by using drawing filters and choosing alternate symbology for items.

Tools > SmartSketch Settings

The **Tools** > **SmartSketch** Settings command sets up what relationships are recognized by the software. The relationships selected on the **SmartSketch** dialog box are recognized as you draw within **Auxiliary Graphics**.

Tools > Custom Commands

The **Tools > Custom Commands** opens the **Custom Commands** dialog box. You can choose a macro and run it. Some macros are delivered with the software, generally in the C:\Program Files\Smartplant\P&ID Workstation\Program directory. You can also record your own macros and store them wherever you want to.

Tools > Customize

The **Tools > Customize** opens the **Customize** dialog box, which allows you to customize toolbars or menus to fit your workflow. You can add commands to toolbars or menus, or you can create new toolbars and menus.

Tools > Options

The **Tools** > **Options command** changes settings that control recently used files, screen appearance, placement information, export formats, and so forth.

General Tab (Options Dialog Box)

The general tab sets options for updating links in the drawing and displaying the drawing, the status bar, and recently used files. The **Options** dialog box opens when you click **Tools > Options**.

Update links automatically at open - Updates links automatically when a drawing is opened.

Recently used files list - Sets the number of entries for the recently used files list on the **File** menu.

Display as printed - Displays the drawing as it appears when you print it.

Show status bar - Displays the status bar at the bottom of the main window. When you point at the command button on the toolbar, the description appears in the status bar at the bottom of the window. Also, prompts that explain what to do at each step of the command appear in the status bar, and messages about the operations the software is performing appear in the status bar.

Colors Tab (Options Dialog Box)

The color tab allows you to select options for the background, highlight, selection, and handle colors in the drawing. The **Options** dialog box opens when you click **Tools > Options**.

Background - Sets the default background color for all Drawing views in the active drawing.

Highlight - Sets the highlight color.

Selected items - Sets the color of selected items. The connect points in the drawing appear in this color, too.

Handles - Sets the color of handles when an item is selected.

Use Defaults - Sets all the colors listed above to the default display value.

📝 **Notes:**

- To set the symbology, (for example, the line weight and color) for item types throughout a plant structure, use Options Manager.

Placement Tab (Options Dialog Box)

Sets the default construction status for the active drawing, and tolerances for locating items and moving around in the drawing. The **Options** dialog box opens when you click **Tools > Options**.

Default construction status - Assigns the selected construction status to all placed items. This list contains the following options: **New**, **Existing**, and **Future**. The plant administrator sets the plant-wide default at setup, but this option allows you to change the default on a drawing-by-drawing basis. The construction status for an item appears in the **Properties** window when the item is selected. Also, the construction status can be set for individual drawing items by changing its value in the **Properties** window.

Locate - Specifies the range, called the locate zone, at which a candidate item is highlighted when another item approaches. A candidate item is any item in the drawing that meets the placement rules of the approaching item. For example, a pipe run can be a candidate item, and a valve can be an approaching item to that pipe run. The pipe is highlighted when the valve is within the locate tolerance. If you are not placing an item but only selecting drawing items, the locate tolerance defines how close that the pointer must approach in order to select the item.

Break-away - Sets the number of pixels at which an item disconnects from its parent item. For example, to move a valve, you drag the valve. If you keep the pointer within the break-away tolerance from the line, the valve does not disconnect from the line.

Place partner OPC in Stockpile - Indicates that when a new off-page connector, OPC, is placed, the partner OPC is automatically stored in the Stockpile. If you clear this option, the **Set Stockpile Location of Partner OPC** dialog box appears after you place a new OPC. Using this dialog box, you can indicate the drawing in which the partner OPC is stored until you place it in that drawing.

Files Tab (Options Dialog Box)

Sets options for inserting objects in the drawing. This tab is part of the **Options** dialog box, which opens when you click **Tools > Options** on the main menu bar. These settings take effect in your next drawing session.

Scale reference files - Sets a scale option for importing a drawing.

Coincident (1:1) - Imports a reference file, also known as an inserted object, at full scale (1:1). This setting causes the reference file to appear temporarily in the lower left of the drawing and determines a drawing scale that fits the reference file within the drawing. The reference file is at the end of the pointer, ready to be placed in the drawing, much like a catalog item during placement mode.

Select Scale - Sets the drawing scale to a standard ratio. The specified ratio defines the size of the drawing in relation to the size of the real-world object. For a 2:1 ratio, the *2* represents the size of the drawing and the *1* represents the size of the real-world object.

 Notes:

- The **Fit to Sheet** option in the **Select Scale** list determines a drawing sheet scale that fits the reference file within the sheet but allows you to specify where the file is inserted by clicking on the drawing sheet.

Custom Scale - Sets the scale for a drawing that you insert. For example, when you type 3 and 2 in each of the respective boxes, the scale of the foreign drawing is one and one half times its original size.

My Reports - Specifies the folder where your user-defined, or personal, report templates are stored.

Browse - Opens the **Browse** dialog box, which allows you to select a local or network folder to store your user-defined reports and report templates.

Using Catalog Explorer: An Overview

Catalog Explorer is an interface to view and manipulate an electronic catalog containing drawing symbols and their associated properties. Conceptually, the **Catalog Explorer** resembles a manufacturer catalog with pictures of parts to build a plant. The paper catalog contains many items arranged in sections; you can search for items or browse through the catalog and bookmark certain pages and sections. **Catalog Explorer** replaces the paper catalog with an electronic equivalent. You can browse the catalog tree view for drawing items that you need and create bookmarks for important categories.

Catalog Explorer consists of two main views: the **Tree** view and the **List** view. In the **Tree** view, you can view the nodes of the catalog file system and open or close the nodes to see the contents. In the **List** view, you see the contents of the opened node. You also use the **List** view to select catalog items before placing them in a drawing or in a stockpile. You can create a navigation bar in the region between the tree and list views for shortcuts to places throughout the catalog.

Toolbar buttons carry out commands and allow you to customize the tree and list views to suit your work session. These toolbar buttons are located at the top of the tree view.

Working With the Catalog Explorer Tree View: An Overview

The tree view displays a hierarchy of nodes that contain catalog items. You can navigate through the tree view by clicking the different nodes. To open a node, click the "+" to the left of the node name. To close a node, click the "-" to the left of the node name. If the selected node contains any drawing symbols, the software displays them in the list view.

Customizing Catalog Explorer: An Overview

The **Catalog Explorer** toolbar buttons access the **Catalog Explorer File**, **View**, and **Navigation** menus. The commands on these menus allow you to customize the **Catalog Explorer**. After customization you can control the content and display of components in the **Catalog Explorer** window. For example, you can control the display of icons in the list view and define shortcut buttons that allow you easy access to user-defined groups of catalog items.

The **Catalog Explorer File** menu provides access to commands that allow you to manipulate list view items and tree view nodes in the **Catalog Explorer**. Also, you can search for catalog items.

The **Catalog Explorer File** menu provides access to commands that allow you to manipulate list view items and tree view nodes in the **Catalog Explorer**. Also, you can search for catalog items.

Catalog Explorer File Commands

The **Catalog Explorer File** menu provides access to commands that allow you to manipulate list view items and tree view nodes in the **Catalog Explorer**. Also, you can search for catalog items.

Find Dialog Box

The catalog explorer find dialog box allows you to search the active catalog for a specific item based on user-defined search criteria. Open this dialog box by clicking **File > Find** in the **Catalog Explorer**.

Named - Specifies the name of the item for which you want to search. You can type an asterisk, "*", as a wildcard character to find multiple characters, or type a question mark, "?", as a wildcard character for a single character.

Class - Specifies the class of item for which you want to search. You can type a percent sign, "%", as a wildcard character to find multiple characters, or type a question mark, "?", as a wildcard character for a single character.

Look In - Specifies the drive or folder that contains the symbol library that you want to search.

Browse - Opens the **Explore Elsewhere** dialog box, allowing you to search any network drives or the local directory structure for a symbol library to display in the **Look In** box.

Results List - Lists the items that met the defined search criteria. You can drag catalog items from this area into a **My Catalog** set of items in the **Catalog Explorer** tree view.

Find Now - Starts a search of the symbol library defined in the **Look In:** box.

Stop - Ends the current search.

New Search - Clears all search criteria in preparation for a new search.

New Command

The **Catalog Explorer > Catalog > New** command creates a new node under the selected node in the **Catalog Explorer**.   This command is available for creating new nodes under the **My Catalog** > **Toolbars**.

Delete Command

The **Catalog Explorer** > **Catalog** > **Delete** command delete a node under the selected node in the **Catalog Explorer**.   This command is available for deleting nodes under the **My Catalog** > **Toolbars**.

Rename Command

The **Catalog Explorer** > **Catalog** > **Rename** command will rename a node under the selected node in the **Catalog Explorer**.   This command is available for renaming nodes under the **My Catalog** > **Toolbars**.

Refresh Symbol Toolbar Command

The **Catalog Explorer** > **File** > **Refresh Symbol Toolbar** activates plant- or user-level symbol toolbars or updates the symbol toolbars to incorporate any modifications to a toolbar.

Send to Stockpile Command

The **Catalog Explorer > Catalog > Send to Stockpile** sends the catalog item selected in the list view to the **Stockpile**. This command is particularly useful for instrument loops and packages and other items that reside in a stockpile but generally are not depicted graphically in a drawing.

Send to Drawing Stockpile Command

The **Catalog Explorer > Catalog > Send to Drawing Stockpile** sends the catalog item selected in the list view to the **Active Drawing Stockpile**. This command is particularly useful for instrument loops and packages and other items that reside in a stockpile but generally are not depicted graphically in a drawing.

Catalog Explorer View Commands

The **Catalog Explorer > View command** provides commands that allow you to control the display format of catalog items within the **Catalog Explorer** list view.

Large Icons - Displays the icon for an item in the list view in a larger format.

Small Icons - Displays the icon for an item in the list view in a smaller format.

List - Displays the icon and name for an item in the list view in a simple list.

Details - Displays the icon, name, and file location for an item.

Catalog Explorer Navigation Commands

The **Catalog Explorer** > **Navigation** commands in **Catalog Explorer** provides commands that allow you to control the display and assignment of shortcut buttons within the **Catalog Explorer**. Shortcut buttons provide shortcuts that bookmark a node in the active Tree view of **Catalog Explorer**. These buttons reside on the navigation bar between the tree and list view in the **Catalog Explorer** window.

For example, if your workflow requires you to route a lot of piping, you want to make a shortcut button to the **Process Lines** node. Then when you open **Catalog Explorer** and click this shortcut button, you open the node for the piping and all the piping types appear in the list view. Consequently you save the time and effort of browsing the catalog node structure in the tree view in order to find the node that you want to open in the list view.

The software does not limit the number of shortcut buttons that you can create, but you are limited by the size of the **Catalog Explorer** window. Also, shortcut buttons are shortcuts to catalog *nodes* only. If you want to make a shortcut to a particular symbol that you use a lot, you can use the symbol toolbar.

All of the commands for managing your shortcut buttons are found on the **Catalog Explorer Navigation** menu.

Add Button - This command adds a shortcut button to **Catalog Explorer** for the active tree view node. For example, if you frequently select nozzles, you may add a shortcut button to the nozzles node of the **Catalog Explorer** tree view by selecting **Nozzles** in the tree view and then choosing **Add Button** from the menu that appears when you click the down-arrow of the **Navigation** menu. The option **Show Buttons**

in the menu must also be selected in order for your shortcut button to appear between the tree view and list view windows of **Catalog Explore**r.

The storage of the Buttons in SP P&ID Catalog Explorer is in the Registry at

HKEY_CURRENT_USER\Software\VB and VBA Program Settings\SPCatalogExplorer\NavButtons

Remove Button - Removes the active shortcut button from the list of shortcut buttons in **Catalog Explorer**.

Show Buttons - In Catalog Explorer Navigation menu, a check mark next to the menu item indicates that shortcut buttons appear in **Catalog Explorer**. The buttons appear on the navigation bar between the tree and list views.

Designing Symbol Toolbars

Symbols toolbars allow you to create toolbars that contain the symbols you use most frequently, keeping them conveniently located for easy access. Symbol toolbars can be project specific or user specific. With the most often used equipment, instrumentation, or piping symbols, for example, your symbol toolbars can streamline your work considerably. The symbol toolbars can be docked on any side of the main window.

You can define up to two levels of toolbars. There is no limit on the number of symbols in a toolbar. For example, you can create a Vessels node under Toolbars, and under Vessels you can create Vertical and Horizontal. You cannot create another level under that. A divider appears in the toolbar between levels.

If you are an administrator or have Full rights to Catalog Manager, you may define project specific symbol toolbars in Catalog Manager. All users in a project will see project specific toolbars. Any user may define user specific symbol toolbars in SmartPlant P&ID. That user will only see user specific symbol toolbars.

The toolbar definition is stored in the ~:\Documents and Settings\<Your Login>\My Catalog\<Plant Name>\Toolbars\<Toolbar Name> folder.

Define a User Level Symbol Toolbar

1. In the **Catalog Explorer** tree view, open the **My Catalog** node.

2. Right-click on the **Toolbars** node under **My Catalog**.

3. Click **New** on the shortcut menu.

4. Enter a name for the new toolbar.

5. In the **Catalog Explorer** list view, find symbols you want to place on your new toolbar.

6. Drag symbols from the list view and drop them into the new toolbar in the tree view.

7. When you have added all the new symbols you want to the new toolbar, right-click on the name of the toolbar, and click **Refresh Symbol Toolbar**. The new toolbar will appear on the left side of the window.

   Notes:

   - You can dock the symbols toolbars on any side of the main window.

   - You can define up to two levels of toolbars. For example, you can create a **Vessels** node under **Toolbars**, and under **Vessels** you can create **Vertical** and **Horizontal**. A divider appears in the symbol toolbar between levels.

   - Plant-level symbol toolbars are created in **Catalog Explorer**.

   - You must have the proper permissions, granted in SmartPlant Engineering Manager, to create symbol toolbars.

Define a Plant Level Symbol Toolbar

1. Open Catalog Manager.

2. In the Catalog Explorer tree view, open the Symbols node.

3. Right-click on the Toolbars node. If no Toolbars node exists, create a Toolbars folder in your Symbols directory, and refresh the view.

4. Click New on the shortcut menu.

5. Enter a name for the new toolbar.

6. In the Catalog Explorer list view, find symbols you want to place on your new toolbar.

7. Drag symbols from the list view and drop them into the new toolbar node in the tree view.

Using Filters: An Overview

You can use filters in many ways throughout the software and its stand-alone applications and utilities.

You can apply filters to the **Engineering Data Editor** to displays a list of pumps and their characteristics in a tabular format, where you can edit the properties of the pumps.

You can use filters when you create report templates to display particular items on a report. You can use filters in Rule Manager to define how items interact within a view. You can associate filters with the source or target properties in rules. And you can use filters to find and replace items.

You can create filters to control the display of hierarchy items in Drawing Manager. These filters are not saved to any folder in Filter Manager, but the methods of creating these filters are the same as those for creating filters saved in Filter Manager.

Notes:

- If you are using a workshare environment and you are at a satellite site, do not create Project Filters at a satellite site. However, you can always create My Filters in the Filter Manager or **Select Filter** dialog box environment.

Compound Filter Properties Dialog Box

A compound filter consists of more than one simple filter. The simple filters are added to the compound filter by dragging the simple filter or by creating new simple filters under the compound filter in the filter hierarchy. Compound filters apply only to homogeneous item types. This dialog box specifies properties of a compound filter, including the name, description, and whether to match all or any of the simple filter criteria.

Description - Specifies a phrase or sentence about the filter. The description can be any combination of characters and has no length limit. The description appears as a ToolTip when you point to the filter name on the Filter Manager interface.

Filter Method - Allows you to choose either **Match all** or **Match any**.

Match all - Specifies that items matching ALL of the filtering criteria pass through the filter.

Match any - Specifies that items matching any one or more of the filtering criteria pass through the filter. **Match any** is the default matching method.

Name - Specifies the filter name. The name can be any combination of characters and has no length limit. Filter names within a plant must be unique. This name appears as the filter name on the Filter Manager interface.

Notes:

- All of the filters that make up the new compound filter need to be of the same item type, which displays in the Filter for list on the Filter Properties dialog box. For example, all of the simple filters making up a compound filter can be of type Equipment: Mechanical. You cannot mix Equipment: Mechanical with Equipment: Heat Transfer or any other item type.

- Once one simple filter has been assigned to a compound filter, if you create a new filter under that compound filter, the Filter for property will be specified and you cannot change it. This is; of course, because the entire simple filters under one compound filter must be of the same item type.

- When you point to the name of a compound filter in the tree view in Filter Manager, the filter description displays in a ToolTip. You set the filter description in the Description field in the Compound Filter Properties dialog box.

- If you have the Select Filter dialog box open, do not open Filter Manager and make changes to filters in that utility also.

Placing Equipment and Equipment Components: An Overview

You can place various types of equipment in your drawing, such as towers, boilers, heat exchangers, and reactors. You also place equipment components in drawings. Equipment components are items associated with pieces of equipment, such as nozzles and trays.

As you place equipment components, the software automatically creates a group relationship between the equipment and the component. As a group, the components move when you move the equipment. However, you can only associate an equipment component with a single piece of equipment. When you place an equipment component, the component is associated with the last selected or highlighted piece of equipment, creating an equipment group.

Depending on the rules defined for the equipment or equipment component, you can have limits to the possible placements for the equipment. For example, specific trays can be flagged for placement inside the piece of equipment, and nozzles can be flagged for placement outside the piece of equipment. Rule Manager defines rules that determine the way that model items interact during manipulation.

The software includes a standard group of equipment and equipment component symbols. However, you or your plant administrator can change and add to these symbols as needed for your plant.

Place Equipment

1. In the **Catalog Explorer** list view, click the equipment that you want to place.

2. Drag the item to the appropriate place in the drawing.

3. Position the item and release to place it.

   ✎ **Notes:**

   - In addition, you can click the item in the Catalog Explorer list view and then click the appropriate point in the drawing to place the item. Press Esc to quit placement mode. Or, you can escape placement mode by right-clicking.

   - Placing items by drag and drop are not cyclical. Placing items by clicking to select anc clicking to place are cyclical – that is, you will continue to be able to place the item until you right-click or click Esc.

4. In the **Properties** window, enter values for the equipment properties in the appropriate rows in the table.

Parametrics

Parametric items have yellow handles that allow the item's size to be changed. **Parametric handles** allow you to modify different parts of parametric items. Standard parametric handles represent up to four driving dimensions that have been applied to an item. Parametric handles can be located in four standard positions: top center, bottom center, left center, and right center of the range of an item. You can scale the item parametrically by dragging any parametric handle. The software shows you the result of the operation dynamically so that you can be sure to scale the item correctly.

✎ **Notes:**

- An equipment item can be defined as Parametric or Scalable but not both.

Scaling

Scaleable items have black handles that allow the item's size to be changed proportionally. To make a scaleable symbol, the **IsScalable** property must be set to True. No symbol is delivered with the **IsScalable** property set to **True**. You or your project administrator may change this property for your plant in **Catalog Manager**.

✎ **Notes:**

- An equipment item can be defined as Scalable or Parametric but not both.

Placing Multiple Representations

If the same piece of equipment needs to be represented on more than once, perhaps for a detailed or alternate view, in a P&ID plant, you may place multiple representations of the item using the Engineering Data Editor. You may think of a multiple representation, as a child item associated with the parent item that already exists in another drawing. You are permitted only a single representation of an item in any ONE drawing.
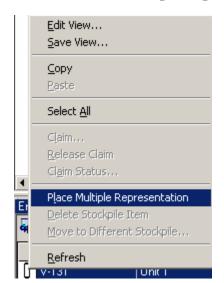
The Replace command will replace a multi representation symbol providing the definition attributes for source (parent) and target (child) are identical. The definition attributes are "Equipment Class", "Equipment SubClass", and "Equipment Type". Symbols can be replaced either using the single Replace command or the Find and Replace dialog.

Steps to Place Multiple Representations

1. Open a drawing and place an **Equipment Vessel**.

   a. **Symbols** > **Equipment** > **Vessels** > **Vertical Drums**

   b. Define a value for the Tag Prefix and note the value for Item Tag, this will make it easy to locate in the Engineering Data Editor.

2. **Exit** the **Drawing**.

3. Open another drawing that is a target for your **Multiple Representation** of an item. (Do not open the drawing with the parent item at all.)

4. Find the parent item in the **Engineering Data Editor** by clicking the **Other Drawings** button and choosing an item type from the list that best describes the type of item you want to represent. (e.g. Equipment)

Engineering Data Editor					
Item Tag	Plant Group Name	Tag Prefix	Tag Seq No	Tag Suffix	Equip Type
V-111	Unit 1	V	111		Narrow 1 diameter, 1 chamber, elliptica

5. In the **Engineering Data Editor**, right-click on the icon at the far left of the parent item.

6. Select **Place Multiple Representation** from the shortcut menu.

7. Drag your cursor into the Drawing View and Click the location in the drawing where you need to place the **Multiple Representation**.

Equipment Reference Data

In the delivered reference data, the following equipment categories exist under Equipment:

Black Box System: Contains a single black box symbol.

Heat Transfer Equipment: This category contains heat transfer equipment such as exchangers, cooling towers, heaters, or boilers.

Labels – Equipment: This category contains labels that may be placed on all equipment as well as a description category containing description labels for specific equipment types.

Mechanical: This category contains pumps, compressors, separators, or packaging equipment.

Other Equipment: This category contains civil (such as sumps and manholes), sound (such as silencers), and other electrical equipment.

Vessels: This category contains vessels such as reactors, towers, drums, tanks or silos.

Place Equipment Components

- Flanged Nozzle-Internal, Nozzle-Internal, and Pump Nozzle-Internal can all be placed internally (for example, in a tank). For internal placement of equipment, you drag the nozzle to the interior of a tank and place it. If you want the nozzle opening facing to the right, approach the tank from the right when placing the nozzle. In other words, the opening of the nozzle will be placed in the direction that you approach the tank during placement.

- If you need to place a nozzle so that it is associated with equipment but not attached to it, you can do the following.

  1. Attach the nozzle to the equipment that you want it related to.

  2. Select the nozzle.

  3. While holding down the **Alt** key, drag the nozzle an arbitrary distance away from the equipment.

- Components will have a placement style determined by Rule Manager. The default for trays is to place inside a piece of equipment while nozzles automatically place on the outside of the graphics.

Equipment Components Reference Data

In the delivered reference data, the following equipment component categories exist under Equipment Components:

General: This category contains demisters, weirs, and vortex breakers.

Heating Components: This category contains components such as air cooler fans, bundles and coils, burners, cooling tower cells, ducts, louvers, and some miscellaneous labels.

Labels – Equipment Components: This category contains labels that may be placed on equipment components such as nozzle id, item tag, and Material of Construction.

Mechanical Components: This category contains mechanical components such as screens, separators and buckets and scales.

Nozzles: This category contains a variety of nozzles, including flanged nozzles, manways, and instrument connectors.

Trays: This category contains a variety of trays including bubble cap trays, generic trays, pans, sieve and chimney trays, and valve trays.

Vessel Components: This category contains components specific to vessels such as beds, distributors, domes and boots, and some miscellaneous components.

Routing Lines: An Overview

Whether you are routing pipe or signal lines, you can manipulate all lines by using the same basic procedures. After you select a type of line route from the **Catalog Explorer** list view, use the commands on the **Line Routing** ribbon to help construct or modify the line.

Lines consist of a series of line segments. The software places handles, represented by black squares or dots, at each segment vertex along the line route. You can use these vertex handles to move, reroute, or connect lines. You can add vertices to add segments to the line using the commands on the **Line Routing** ribbon.

The software also provides a shortcut menu associated with lines. Right-click a line and choose from several commands. You use this menu to break runs or join runs. You can also use this menu to construct select sets of pipe runs.

Route a Line

1. In the **Catalog Explorer** list view select the line that you want to place.

   Notes:

   - This action displays the **Line Routing** ribbon and starts line placement mode.

   - Press **Alt** while routing a line or click **Diagonal Mode (Alt)** on the **Line Routing** ribbon to override the current line vector and allow routing in any direction and at any angle.

2. Click a connect point on an item in the drawing.

   Notes:

   - You can click an empty point in the drawing to start the line, too.

3. Click to place the vertex at the next point in the line.

4. Continue clicking to place the vertices of the line in the drawing.

5. When the line is complete, right-click to confirm the placement.

   Notes:

   - You can escape line placement mode by right-clicking two more times.

Move a Line Using Vertex Handles

1. Click an existing line in the drawing.

2. Click the vertex handle of the line route that you want to move.

3. Drag the vertex to a new position in the drawing and release.

   📝 **Notes:**

   - While moving the vertex handle to modify the line segment, click **Diagonal Mode (Alt)** on the **Line Routing** ribbon or press **Alt** to allow the line segments to move freely at any angle.

Line Routing Ribbon

The **Line Routing Ribbon** Appears when you select a line element from the **Catalog Explorer** list view or the **Engineering Data Editor** or when you select an existing line in a drawing. You can dock the **Line Routing** ribbon in the main toolbar; it remains there for the current design session unless you move it again.

Diagonal Mode (Alt)

The **Diagonal Mode (Alt)** command allows you to route lines in any direction, instead of the usual constraint of horizontal or vertical line routing. Selecting the Alt key while routing a line will also route a line at an angle. This procedure is usually popular with signal and utility lines, but you can use it for any line routing.

Insert Segment

The **Insert Segment** command causes the software to insert line segments instead of routing new lines or instead of moving existing lines.

Insert Line Segments into an Existing Line

1. In the drawing, select the existing line that you want to insert a segment into.

2. On the **Line Routing** ribbon, click **Insert Segment**.

   📝 **Notes:**

- Instead of clicking Insert Segment, you can hold Shift while moving a line to insert segments.

3. Drag the selected line to the new position.

Break Run

The **Break Run** command allows you to break a pipe run. After you break the run, select the pipe run and select Edit > Properties to enter or change any property values.

Break a Pipe Run

1. In the drawing, right-click the pipe run that you want to break.

2. On the **Line Routing** ribbon, click **Break Runs** or on the shortcut menu, click **Break Runs**.

   📝 **Notes:**

   - To activate the shortcut menu, select the pipe run and right mouse click.

3. In the drawing, click the point where you want to break the run.

   📝 **Notes:**

   - The software breaks the selected pipe run into two separate runs at the selected point.

Join Runs

The **Join Runs command** allows you to join two pipe runs. The properties of the pipe run after using **Join Runs** will be the values of the pipe run you selected first.

Join Pipe Runs

1. In the drawing, right-click the first pipe run that you want to join.

2. On the **Line Routing** ribbon, click **Join Runs** or on the shortcut menu, click **Join Runs**.

   📝 **Notes:**

- To activate the shortcut menu, select the pipe run and right mouse click.

3. In the drawing, click the second pipe run to join the two runs.

Pipe Gapping

See Gapping Lines: An Overview

Pipe Heat Tracing

Heat tracing is an intelligent property of piping. To graphically show the heat tracing for a piperun, set the HT Medium (heat tracing medium) property to a value such as E or ST. The heat tracing will automatically show up on that line and all components on the line.

The linestyle and orientation of the heat tracing is set in Options Manager.

Pipe Jacket

To jackets pipe, you must open Options Manager and make sure that a heat trace medium is selected in the **Jacketed Pipe Heat Trace Media** property. Also, for specifying pipe jacket nominal diameters, you must enter a file name and path for the **Pipe Jacket Nominal Diameter Configuration File** property, and then you can enter the relevant values on the **Pipe Jacket Nominal Diameter** dialog box, also found in Options Manager.

Line Connectivity

When a new pipe run is placed such that it connects to existing pipe runs at either end, those pipe runs are automatically joined together if they have the same properties. The symbol placement command creates a new line when an inline component is placed directly up against an existing nozzle or other inline component. The pipe run created in this way has a zero length connector that attaches the two symbols. Pipe runs that only have zero length connectors are aggressively joined up with the pipe runs they attach to; the properties for pipe runs with zero length connectors need not match. The zero length pipe run that gets created when one inline component is placed directly up against another inline component is automatically joined to the existing pipe run, too. When a new pipe run is placed that starts or ends at a component that belongs to an existing pipe run, the new pipe run is automatically joined with the existing pipe run. When a new pipe run is placed that starts or ends at the endpoint of an existing connector, the two pipe runs are joined and the new connector is actually merged with the existing connector. The automatic joining takes

place in the same way for both placement and modification of connectors. At the time that the connection is made, the two pipe runs are tested for compatibility. If they pass the property criteria, they are automatically joined together.

If two pipe runs have different properties, they are not automatically joined together. If you want them joined together, the **Join Runs** command is still available for this purpose. Of course when two pipe runs with differing properties are joined into one, the properties of the surviving pipe run are retained and the others are lost. Similarly, if you must define two different properties for two parts of a single pipe run, then the pipe run must be broken. The **Break Run** command is still available for this purpose.

Pipe runs always consist of a linear sequence of members and never include any branches. Placing a pipe run that branches off of an existing pipe run does not cause them to be joined together. Reducers are always located at the end of a pipe run and never in the middle. Placing a new pipe run that connects to a reducer does not cause the pipe runs to be joined.

Auto Join

Pipe runs are automatically joined together whenever possible. When a new pipe run is placed, and it connects to existing pipe runs at either end, those pipe runs are automatically joined together if they have the same attribute values. Pipe runs that only have zero length connectors are joined with the pipe runs they attach to. (The attribute values for pipe runs with zero length connectors do not need to match.) This means that both the bottom-up and the top-down approaches produce the same final result.

When a new pipe run is placed that starts or ends at a component that belongs to an existing pipe run, the new pipe run is automatically joined with the existing pipe run. When a new pipe run is placed that starts or ends at the endpoint of an existing connector, the two pipe runs are joined and the new connector is actually merged with the existing connector.

The automatic joining takes place in the same way for both placement and modification of connectors. At the time the connection is made the two pipe runs are tested for compatibility. If they pass the attribute criteria, they are automatically joined together. The zero length pipe run that is created when one inline component is placed directly up against another inline component is automatically joined to the existing pipe run.

Pipe runs always consist of a linear sequence of members and never include any branches. Placing a pipe run that branches off of an existing pipe run does not cause them to be joined together.

Reducers are always located at the end of a pipe run and never in the middle. Placing a new pipe run that connects to a reducer does not cause the pipe runs to be joined.

If two pipe runs have different attribute values, they are not automatically joined together. If you want them joined , the **Join Runs** command is to be used. When two pipe runs with differing attribute values are joined into one, the attributes of the surviving pipe run are retained and the others are discarded. Similarly, if you must define two different attribute values for two parts of a single pipe run, then the pipe run must be broken. The **Break Run** command is used.

Piping Reference Data

In the delivered reference data, the following piping categories exist under **Piping**:

Gaps: This category contains manual gap symbols.

Labels – Piping Segments: This category contains labels that may be placed upon piping segments indicating a property value such as flow direction, line number label, moc note, piping material class, and slope.

Routing: This category contains process and utility lines such as primary, secondary, and hose.

Using Piping Components: An Overview

Piping components are graphic elements that represent processes or functions within a particular piping segment. Piping components include valves, flanges, reducers, and strainers. In drawings, piping components are connected with lines.

Generally, piping components automatically orient themselves when you drag them near an existing pipe segment. For example, if you drag a horizontal valve to a vertical pipe, the valve automatically orients itself vertically. If several possible orientations exist for the piping component, you can use the Configuration tool, or PickQuick, to review the possible placements for the component and select a final orientation for the item. Rule Manager defines rules that determine the way that model items interact during manipulation.

As you place piping components, the software automatically creates a relationship between the pipe route and the component to create a pipe run. When you move the pipe run, the pipe route and the associated piping components both move. Your plant administrator can modify the relationships between piping components and piping in Rule Manager.

The software includes a standard group of piping component symbols. However, your plant administrator can change and add to these symbols in Catalog Manager as needed.

Place a Piping Component

1. In the **Catalog Explorer** list view, click the piping component that you want to place.

2. Click the item and place onto the appropriate pipe run in the drawing.

3. Position the item and release.

4. In the **Properties** window, enter values for the properties of the piping component.

   Notes:

   - As you drag piping components over a pipe run, or the pointer passes over runs while you are in placement mode, the pipe run is highlighted. If you place a piping component when a pipe segment is not highlighted, the piping component is placed in free space if the placement rules allow the action.

- If the item that you place has multiple possible orientations in relation to other items, you can use the Configuration tool, or PickQuick to select the appropriate placement for the item.

- If you want to move a piping component but preserve its connection to the pipe run, you can use the **Alt** key as you drag the component. Then even if you drag the component away from the line, the line routes itself so as to preserve its connection to the component.

Configuration (PickQuick) Tool

In addition to using the manipulation handles to orient an item after you place it, you can use PickQuick to select the orientation of some items, such as check valves or flow arrows or other inline items, when you are placing them. Pausing after you drag the item to the appropriate place in the drawing activates the Configuration tool, if it is available for the selected item, and allows you to select the proper orientation for the item.

🖉 **Notes:**
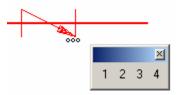
- PickQuick helps you to select items that overlap each other.

Place an Item With the Configuration (PickQuick) Tool
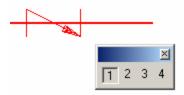
1. Drag the item to the appropriate position on the line.

2. Pause until ellipses appear beside the pointer.

3. Click to display the Configuration toolbar near the pointer.

4. Move the pointer over each toolbar button to preview the available orientations.

5. When the appropriate orientation is displayed, click the corresponding button on the Configuration toolbar to place the item in that orientation.
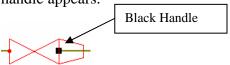
Select an Item Using the Configuration (PickQuick) Tool
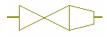
1. Pause the pointer over the item that you want to select.

2. When the pointer appears with ellipses, click.

3. On the **PickQuick** toolbar, which appears near the pointer, move the pointer over the numbered **PickQuick** buttons, without clicking, to highlight the corresponding items.

   - The first item is highlighted.

   - The second item is highlighted.

4. When the item that you want to select is highlighted, click the corresponding button on the **PickQuick** toolbar.

Placing two components (valve and reducer) end to end

1. Drag the end of the item (Reducer) over the item (Gate Valve) until the black handle appears.

Black Handle
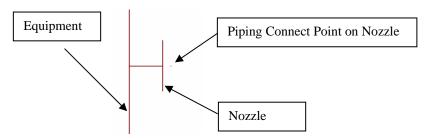
2. Click to place the item or continue to pause to receive the Configuration toolbar.
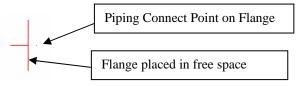
Placing two components (nozzle and piping flange) end to end
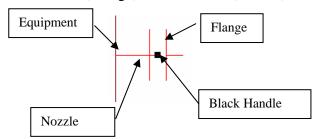
1. Notice the piping connect point that is off the face of the Nozzle. This piping connect point will be the point which the flange will connect too.

| Equipment | Piping Connect Point on Nozzle |

| Nozzle |
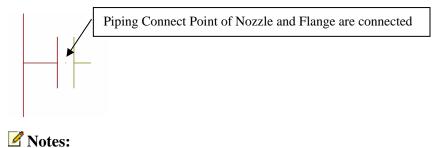
2. Notice the flange also has a piping connect point that is off the face of the Flange. This piping connect point is the point which connects to the point on the Nozzle.

| Piping Connect Point on Flange |

| Flange placed in free space |

3. Drag the item (Flange) to the appropriate position on the Item (Nozzle).

4. Pause the item (Flange) over the item (Nozzle) until the black handle appears.

| Equipment | Flange |

| Black Handle |

| Nozzle |

5. Click to place the item or continue to pause to receive the Configuration toolbar.

Piping Connect Point of Nozzle and Flange are connected

Notes:

- If you want to be able to place two inline components an arbitrary distance apart, regardless of minimum distance settings in Options Manager, do the following.

    1. Place the first component in the line.

    2. Place the second component into the line and connected to the connect point of the first component on the end that you want to attach to.

    3. While holding down the **Alt** key, drag one component along the line away from the other leaving the desired distance between the two.

- If the components are already close to each other but not connected at coincident connect points, you must drag one off of the segment to disconnect it and then move it back to get the connect points attached.

Break Components

Some components inherently limit the copying and suppress the consistency checking of certain properties. For example, a reducer is a break component for the Nominal Diameter property. When changing the Nominal Diameter in a pipeline, the spread of that change is limited by any reducers that exist in that pipeline. The consistency checking that would normally be done between the connected pipes is also suppressed at the reducer.

A break component can be created for any property or collection of properties of a pipe run. To create a break component, you must first create a special filter. This filter must apply to pipe runs and must include the properties to be broken as criteria in the filter. The value for each property in the filter is not important. The final step in creating a break component is to open the symbol in Catalog Manager and select the **Property Breaks** command to select the filter. After creating a break component, when it is placed into a drawing, it breaks the specified properties.

Using Off-Page and Utility Connectors: An Overview

A connector is not actually a label but more like a component that includes a label. Placement behavior for a connector is very similar to a standard piping component label. OPCs, utility connectors, and piping components are inserted into a line.

The drawing needs a connector when a line run continues on another drawing. The minimum information in the label portion is the drawing name to which the connector points and the connector number. Connectors do not have a property for flow direction, although most connectors use an arrowhead to indicate direction. When you place a connector, it does not set or respond to flow direction. The connector represents continuation, not connectivity or flow.

During document creation, when you place the first connector of the pair, the software places the partner in either the Stockpile or the stockpile of another drawing of your choice. The default placement for the partner is into the Stockpile. When you create a connector and place it in a drawing, the property for the drawing name is not defined. The partner in the stockpile reflects the drawing name in which its match was located. When you place the partner from the stockpile, the first connector is updated to reflect the drawing name where you placed the partner connector. As a result, you never need to type a drawing name, and the software guarantees a valid match.
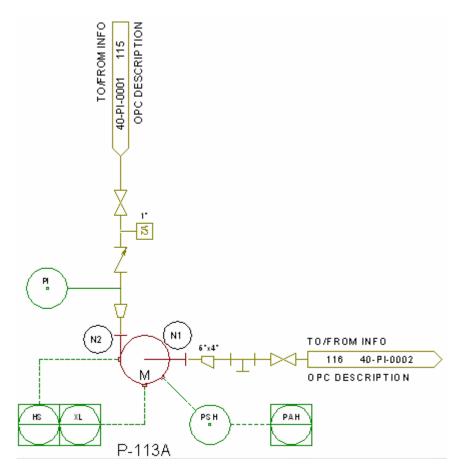
In order to switch between *to* and *from* orientations, you must disconnect the OPC from the line and then rotate or mirror it. Then you can reconnect the line to the other end of the OPC. At the time of placement, you can pause when placing the OPC and wait for the PickQuick toolbar in order to choose the correct orientation.

You cannot change the drawing name in the labels, the value is automatically assigned by the software. However, you can change other properties.

If you copy a connector, the software must generate new connector numbers and add a partner to the stockpile. The property for drawing name changes to **Null** or **Undefined** for the copy.

OPC pairs have a *to* and *from* orientation. The software does not use the graphical *to* and *from* distinctions, but reading the drawing is easier for you with this differentiation.

The below OPC (#115) is a "to" connector if its connected point appears at the "finish" end of the off-page connector. The below OPC (#116) is a "from" connector if its connected point appears at the "start" end of the off-page connector.

If the OPC pair is on drawings, you can **Delete** a connector from the drawing, and the software moves it to a stockpile and updates the properties of its partner; the **Delete from Model** command is dimmed. If the OPC you are deleting is on a drawing and the mate is in the Stockpile, then the **Delete** and **Delete from Model** commands are active; selecting **Delete** will delete the select OPC and place it in the Stockpile, select **Delete from Model** will delete the pair of OPCs (the one from the drawing and the one residing in the stockpile) from the SPPID environment.

If the OPC pair resides in the stockpile to remove them from the stockpile select the **Delete Stockpile Item** from the EDE commans.

✎ **Notes:**
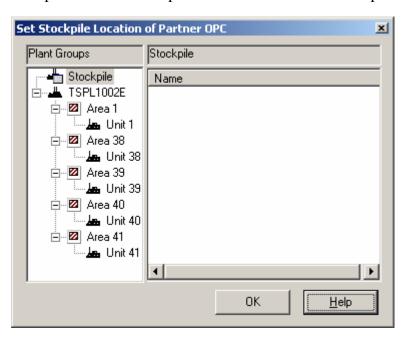
- In **Catalog Explorer**, Off-Drawing or Off-Unit OPCs are available for instruments. Two types of connectors are available for process lines: Off-Drawing or Off-Unit OPCs and Utility Connectors. The correct symbology differentiates OPCs. You define symbology with Options Manager.

Place an OPC or Utility Connector

1. Select the connector you want to place from the list view in **Catalog Explorer** or from the **Stockpile**.

2. Click the location in the drawing where you want to place the connector.

   📝 **Notes:**

   - You can place a connector in free space on the drawing and route a line to it, or you can place it at the end of a line that already is drawn.

   - If you are placing a connector in free space, rotate or mirror the OPC after placement in order to get the correct orientation. Then route the line to it.

   - If you are placing a connector at the end of an existing line, you can wait for the **PickQuick** toolbar so that you can choose the correct orientation.

   - If you are placing a connector at the end of an existing line, you can pause and wait for the **PickQuick** toolbar so that you can choose the correct orientation.

3. The **Set Stockpile Location of Partner OPC** dialog box opens after you place a connector on a drawing, unless you have selected the **Place partner OPC in Stockpile** option on the **Placement** tab of the **Options** dialog box. The **Set Stockpile Location of Partner OPC** dialog box allows you to select the stockpile or a drawing stockpile into which the partner of the new connector is placed.
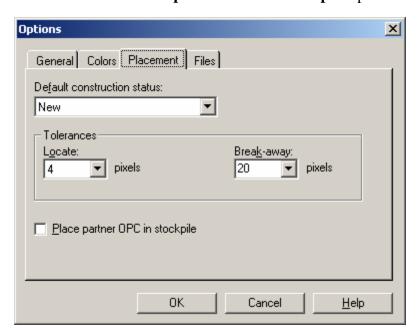
- **Plant Hierarchy** - Displays the hierarchical tree representation of the plant. When you select a node, the drawings associated with that node appear in the **Drawing list** area.

- **Drawing list** - Displays all the drawings associated with the selected node on the **Plant Hierarchy** tree. Choose a stockpile from this view to associate your partner connector with.

4. Select the location for the mating OPC.

- **Stockpile** or a specific **Drawing Stockpile**

5. Select **OK**

Change Default Storage Location for Partner Connector

Place partner OPC in stockpile indicates that when a new off-page connector, OPC, is placed, the partner OPC is automatically stored in the stockpile. If you clear this option, the Set Stockpile Location of Partner OPC dialog box appears after you place a new OPC. Using this dialog box, you can indicate the stockpile or drawing stockpile in which the partner OPC is stored until you place it in that drawing.

1. Click **Tools** > **Options**

2. Select the **Placement** tab.

3. Select or clear the **Place partner OPC in stockpile** option.
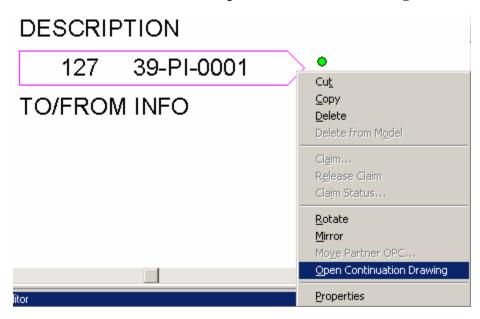
4. Select **OK**

Open Continuation Drawing Command

When the partner of the selected Off-Page or Utility Connector is in a drawing or a drawing stockpile, this command opens that drawing. This command is not available if the partner connector is located in the Stockpile.

1. Select the OPC or Utility Connector for which you want to view the drawing where the partner is located.

2. Right-click the connector.

3. From the shortcut menu, select **Open Continuation Drawing**.

4. Select **Window** > **Tile Horizontal** command to view both drawings.

Using Utility Connectors
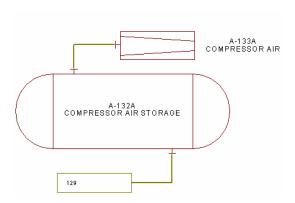
A Utility Off-Drawing Piping Connector and a Utility connector symbol are delivered in the Catalog Explorer. When you place one of these connector types, the mate will automatically have the same graphics as the one you placed.

✎ **Notes:**
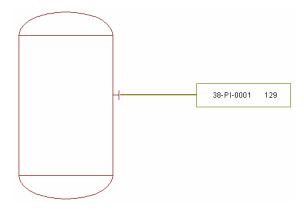
*   To graphically differentiate utility connectors on drawings versus the utility header drawing, you will need to use the Replace command to overwrite the graphics of one of the connector symbols after placement.

For example:

1. Define a value for Fluid Code on the Pipe Run in which you will be placing a **Utility Off-Drawing Piping Connector** upon

2. Place a **Utility Off-Drawing Piping Connector** on the Pipe Run you defined a value for Fluid Code.
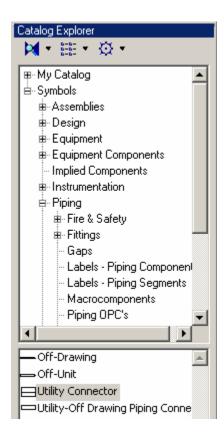
3. Place the mating **Utility Off-Drawing Piping Connector** from the **Stockpile** onto the drawing, which shows the connection to the utility header drawing.
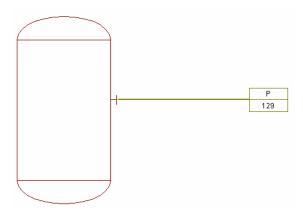
4. Turn on **Replace Mode** either from **Tools** > **Replace** or from the toolbar.

   Error! Objects cannot be created from editing field codes.

5. Click on the Utility connector symbol in **Catalog Explorer** that you are replacing with.

6. Select the Utility Off-Drawing Piping Connector that you just placed. This will change the graphics of the item while maintaining the relationship to the connector on the other drawing.

7. Deselect the Replace Command when finished.

✎ **Notes:**

- Placement of new items will not be permitted until you deselect the replace command.

Piping Component Reference Data

In the delivered reference data, the following piping component categories exist under **Piping**:

Fire and Safety: This category contains fire and safety components such as hydrants, monitors, shower and eyewash, and sprinklers.

Fittings: This category contains reducers, end components, flanges, spacers, and other pipe fittings.

Labels – Piping Components: This category contains labels that may be placed upon piping components indicating a property value such as commodity code, nominal diameter, opening action, and reducer nominal diameter.

Macro Components: This category contains macro components that represent one or more piping components such as vents, drains, and connections.

Piping OPCs: This category contains off-page connectors for off-drawing, off-unit and utility connectors.

Segment Breaks: This category contains piping segment break labels indicating a point where certain properties change value such as temperature, pressure, piping materials class, and supply responsibility.

Specialty Components: This category contains specialty components such as floats, strainers, vents, silencers, and arrestors.

Valves: This category contains a variety of valves including 2 Way Common (such as gate valve, globe valve), 2 Way Other (such as knife valve and instrument root valve), 3 Way, 4 Way, and Angle.

Placing Instrumentation and Loops: An Overview

You can place various types of instrumentation in your drawing, such as orifice plates, flow controllers, and pressure regulators. Instruments are devices used directly or indirectly to measure or control a variable, such as flow or temperature, in a plant process. Instruments can be items such as flow control elements, computing devices, or electrical switches. Two types of instruments exist: inline instruments and offline instruments. Offline instruments may or may not have implied items. Implied items are items that are placed in the database when another item is placed (for example, an offline instrument may imply the placement of an instrument root valve). Signal lines are intelligent line strings that connect offline instruments, inline instruments, and piping.

Inline instruments are components that you can insert into a piping segment. Inline instruments include instrument valves, such as pressure regulator valves, flow controllers, flow indicators, and other instrument components, such as orifice plates. As you drag inline instruments over a pipe segment, the pipe segment appears selected. If you place an inline instrument when a pipe segment is not selected, the instrument is placed in free space.

Offline instruments are components that you do not insert into a piping segment. Typically, these instruments monitor and control inline instruments. Offline instruments include flow controllers, level gauges, and system functions such as digital control stations (DCS) or computers.

Signal lines are used to connect off-line instruments to other instrument components and instrument connectors, or to connect off-line instruments and other signal lines to pipelines, piping components, and equipment nozzles.

You can create instrument loops containing any combination of inline and offline instruments. Instruments loops are a group of one or more instruments or control functions arranged so that signals can transfer from one function to the next to measure and control a process variable.
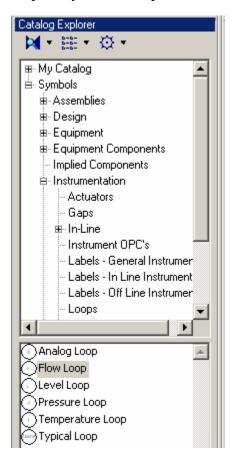
Placement of Inline Instruments

Inline instruments will place in a line similar to the placement of piping components. Offline instruments are not placed in a line. No loop association is required prior to placing an instrument.
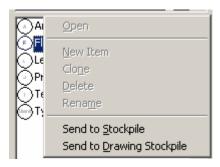
Placement of Instrument Loops

Instruments loops are a group of one or more instruments or control functions arranged so that signals can transfer from one function to the next to measure and control a process variable. To associate an instrument or group of instruments with a loop uses the Loop Tag property.

1. From the **Catalog Explorer** tree view **Loops** node, right-click the instrument loop that you want to place.
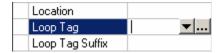
2. Select **Send to Stockpile** or **Send to Drawing Stockpile** from the shortcut menu.

3. Define properties for the loop by entering values in the **Properties** window. The software automatically generates an item tag when you enter a tag suffix.
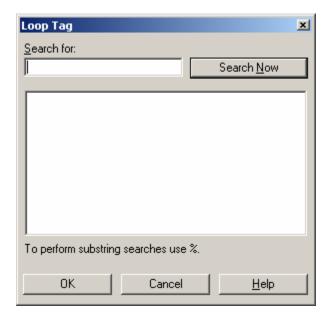
Add Instruments to a Loop

1. Select the item that you want to associate with the instrument loop.

2. Select the **Calculation** button ⊡ in the **Loop Tag** property for the instrument in the **Properties** window.

Location	
Loop Tag	
Loop Tag Suffix	

3. On the **Loop Tag** dialog box, enter characters in the **Search for** box in order to display the available loop tags.

   The **Loop Tag** dialog box (Properties Window) allows you to **Search for** and specify the loop tag for the item whose properties currently appear in the **Properties** window. This dialog box opens when you click the **Calculation** button ⊡ next to the **Loop Tag** property.

- **Search for** - Allows you to type in descriptors to display the available loop tags, limited by parameters that you enter here. The percent character, **%**, is a wildcard. For example, if you want to list only loop tags that contain the letter **F**, you could type **%F%** in the **Search for** box. If you want to display all available loop tags, simply type **%**.

- **Search Now** - Finds and displays the loop tags that match the criteria in the **Search for** box. You can choose a tag from the main window and click **OK** in

order to assign the tag to the item whose properties currently appear in the **Properties** window.

4. Select **Search Now**.

5. Choose the appropriate loop from the resulting list.

6. Select **OK**. (Or you can double-click on the loop to both select and accept the selection.)

Remove an Instrument from a Loop

1. In the drawing, select the item or items that you want to remove from the loop.

2. In the **Properties** window, remove the value for the **Loop Tag** property for the selection.

📝 **Notes:**

- When an instrument is assigned to a loop, validation automatically creates an instrument item tag. If you remove the loop tag property, the instrument item tag remains the same until the instrument is assigned to a new loop.

- You can remove an instrument from a loop by deleting the item from the model, too.

Instrumentation Reference Data

In the delivered reference data, the following instrument categories exist under **Instrumentation**:

Actuators: This category contains a variety of actuators, including diaphragm, manual, and solenoid, which may be placed upon in-line instruments.

Gaps: This category contains manual gap symbols.

In-Line: This category contains a variety of in-line instrument components including orifice plates, flow nozzles, flow instruments, silencers, control valves and regulators.

Instrument OPC's: This category contains off-drawing and off-unit connectors.

Labels – General Instrument: This category includes component labels such as tags, remarks, and accessories.

Labels – InLine Instruments: This category includes in-line instrument component labels including set points, responsibilities and requirements.

Labels – Offline Instruments: This category includes component labels, including requirement, responsibilities and IDs, for off-line components.

Loops: This category contains default loops that may be used including loops for temperature, pressure, flow and level as well as a typical loop.

Off-Line: This category includes functions, off-line without implied and off-line with implied components. Functions include average, bias, and root functions. Off-line without implied includes single- and multi-functions, solenoids, and pilot lights. Off-line with implied includes components involving flow, level, temperature and pressure.

Relief Devices: The category contains relief devices that are attribute-breaking for diameter, temperature and pressure.

Signal Line: This category contains specific types of instrument signal lines including electrical, mechanical, pneumatic, and hydraulic.

System Functions: This category contains system functions including DCS and PLC functions.

Labeling: An Overview

Your main tool to add or edit text content that appears in a label is the **Properties** window. By adding or editing text, you are entering data into the database. Whenever you select a label. The corresponding labeled properties for its item appear in the **Properties** window. The same properties appear in the long list of properties when you select the item itself. Since only the label properties appear when the label is selected, you have an easier method to find to the label properties.

A label always reflects the current property values of the item that it labels. Labels that populate a property at placement are called driving labels. In other words, the predefined property value in the label overwrites the current property value on the item that it labels. Labels that do not overwrite the property at placement are called driven labels. Labels are defined as *driven* or *driving* in Catalog Manager.

At label creation time, the plant administrator can define text content inside the label with the SmartText editor in Catalog Manager. When placing a label later, you cannot edit text such as **OLL=** on the OLL Elevation Equipment label. Nonetheless, you can place free text in an item note.

Different labels have different label properties. The following table identifies some of the label properties that are defined during label creation in Catalog Manager. After the creation of a label, these properties dictate label placement behavior. You do not see label properties while using the design software; these properties perform their function in the background and are invisible to you.

Label Property	Title Block	Flow Arrow	Component	Break
Labeled Item Type	Yes	Yes	Yes	Yes
Placement Type	1-Point	1-Point	1- or 2-Point	2- Point
Leader Line	Not applicable	Not applicable	Yes	Yes
Terminator Type	Not applicable	Not applicable	Yes	Yes
Offset Distance	Not applicable	Not applicable	Yes	Not applicable
Offset Source	Not applicable	Not applicable	Yes	Not applicable
Behavior	Not applicable	Not applicable	Yes	Not applicable
Area Break	Not applicable	Not applicable	Yes	Not applicable
Fit Graphics to Text	Yes	Not applicable	Yes	Yes

Table Notes:

- The four types of labels (title block, flow arrow, component, and break) appear in columns. Label properties appear in rows.

- **Yes** indicates that the property applies to that type of label.

- **Not applicable** indicates that the property does not apply to that type of label.

- Special notes, like **1- Point** for one-point placement, indicate that the property applies under certain conditions.

Placing Labels: An Overview

You can place a label in a few basic steps. First, you select the label that you want from **Catalog Explorer** and then identify the item in the drawing that you want to label.

Labels can have two-point placement or one-point placement. You use one-point placement when a single point in the drawing identifies both the item to label and the location of the label. You use two-point placement when you need one click to identify the item to label and a second click to indicate a different location for the label.

Some labels have no symbology associated with them; that is, they contain text only (for example, the **Short Description** label for Equipment). If you place such a label, but the properties that normally appear have not been entered yet, you have an empty label. Since there is no symbology associated with this type of label, it would be essentially invisible on the drawing. However, empty labels display a question mark so that you can find them more easily and not unnecessarily repeat work.

You can add a leader line to a label by right-clicking on the label and choosing **Leader line display**. If you place a leader line with the label, the first click also identifies the end of the leader.

Labels can be moved by selecting the label and dragging the black square at the center of the label. Drag the label to the location that you want it to occupy in the drawing.

You set the placement type of a label at its creation in Catalog Manager. Afterwards, you cannot change the placement type in the design software.

Using the Types of Labels: An Overview

The type of label that you place determines the workflow that you follow. You define the label type when you create a label in Catalog Manager. As defined in Rule Manager, rules apply when you are placing a label.

The following four types of labels are **Title Block**, **Flow Arrow**, **Component**, and **Break**.
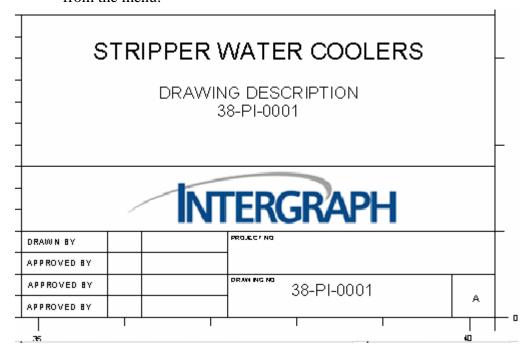
Title Block

The title block label is unique among label types because you do not have to identify the item to receive the label. The title block usually displays general information

about a drawing such as the Title, Drawing Number, Revision info, etc. After you select the label in the Catalog Explorer, you can click or drag it to the location in the drawing where you want to assign the label.

Utilize Catalog Manager, to modify the Title Block Label to exclude or include additional properties. The different template files, which are utilized during the creation of a new Drawing, contain different Title Blocks. To place a Title Block Label within the Title Block either embed the Title Block Label within the Template or place the Title Block Label once the drawing is opened. The Title Block Label is embedded within a few of the delivered Templates such as the Drawing Title*.sym.

There are two methods of editing values for the properties utilized in a Title Block Label.

1. Once the **Title Block Label** is placed, select the label and enter/modify values through the **Property Window**

2. From **Drawing Manager**, select the **Drawing** and select **Edit** > **Properties** from the menu.
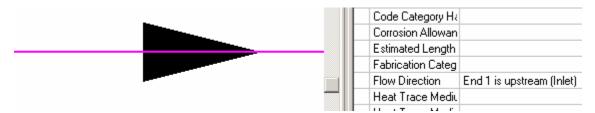
Flow Arrow

You place flow arrows in process piping lines. When you place the label, the arrow orients itself with the flow in the pipe run if you have already defined flow direction. If you have not specified a flow direction, then the direction you place the flow arrow defines it for you (that is, a flow arrow is a driving label).

As you drag the label from the list view of **Catalog Explorer**, pipe runs in the Drawing view are highlighted when you pause over them. Click a highlighted pipe run to place the label. Flow arrows require only one click to place them.

After placing the label, you can change the flow direction of the pipe run in the **Properties** window, and the flow arrow reorients itself accordingly. You cannot change *label* properties that appear in the **Properties** window for a flow arrow.

Code Category Ha	
Corrosion Allowan	
Estimated Length	
Fabrication Categ	
Flow Direction	End 1 is upstream (Inlet)
Heat Trace Mediu	
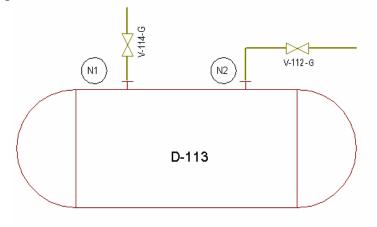
Component

This type of label identifies physical commodities in the model. A typical example is an Equipment ID label. Properties assigned to the label affect its behavior.

Component labels have a unique property: **Label Behavior**. The values are defined in a select list in Data Dictionary Manager and are chosen in Catalog Manager when you create a label. The default value is **Follow**. Values for this property can include the following:

- **Follow (no rotate)** - The label does move but does not rotate, corresponding to changes to the labeled item.

- **Follow** - The label does move and rotate, corresponding to changes to the labeled item.

- **Fixed** - The label does not move if you move the labeled item.

This illustration shows component labels that you use to label vessels, nozzles, and piping components. It also depicts both fixed and rotated orientations of labels.
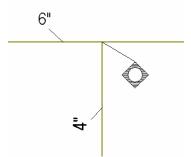
Break

The consistency criteria specified in Rule Manager defines the general behavior of the properties. Break labels provide a way to define exceptions to the general rules. A break label provides a visible and plottable symbol that signifies the end of one property value condition and the beginning of a new condition along a pipe. A changed property value is never propagated across a break label for that property. A break label also suppresses consistency checking for the property it breaks.

A break label can be created for any property or collection of properties of a pipe run. A break label is created in the Catalog Manager just like any other label, except that the Label Type property is set to **Attribute Break**. A SmartText field must be created for each property that is to be broken. In many cases the graphical shape of the label indicates which properties are broken. In these cases the Visible flag on the SmartText is set to False so that the text is not visible. It is also normal to turn the leader line on within the Catalog Manager so that it is automatically displayed at placement time. All of the symbols under Piping\Segment Breaks in the delivered catalog are break labels and are two point placement.

A break label can only be placed at the endpoint of a graphical pipe line where it connects to a component or a branch point. Furthermore, a break label can only be placed at a point where the specified properties are listed in the consistency criteria for the applicable rules. The break label stays attached to the point it was placed on and cannot be dragged away from it.

 Notes:

- You can turn on or off the property for display leader line after you place the label. To display a leader line for the label, right-click the label, and then click Leader line display on the shortcut menu.

- Display Sets can be utilized to turn off the display of Break labels.

Working With the Properties Window: An Overview

The **Properties** window display is a two-column table that provides the common properties of the current selection in the active **Design** window. The active items always determine the content of the **Properties** window. You can only have one **Properties** window open. If only one component is selected, or all components are of the same type, all the properties of that type of component appear. If your select set contains components of different types, then only the common properties appear. If the selected items have different values for their common properties, the value box is blank.

You can modify the values for properties in the **Properties** window. You can display and edit relationships that involve components, component types, or assemblies.

At the top of the **Properties** window is the **Selected Items** list, which itemizes all of the selected objects individually and as a select set. You can display item properties by locating the appropriate item in the **Selected Items** list, as well as by selecting an item in the **Design** window. If more than one item is selected in the **Design** window, then the **Selected Items** list contains the choice **Select Set** as well as individual items in the set.

You can use the **Copy Bulk Properties** and **Paste Bulk Properties** buttons to copy properties from one item to another. This action can be particularly useful when correcting inconsistencies, for instance, or when you must apply changes to more than one item.

✎ Notes:

- The software includes a default set of bulk properties, but you can change those properties specified as bulk in the **Usage** box on the **Layout** tab of the **Advanced Table Properties** dialog box.

- When you pause your pointer over the buttons on the **Properties** window toolbar, a ToolTip appears, giving the name of the button.

- An item and the properties that belong to it are associated when the item is created in Catalog Manager.

- If you copy or paste values in the grid area, use **Ctrl+C** and **Crtl+V**. Do not use the **Edit > Copy**, **Edit > Paste**, or **Edit > Cut** to modify property values in this grid.

Properties Window Toolbar

You can customize the display of the **Properties** window by using the following **Properties** window toolbar buttons:

Alphabetic - Lists properties in alphabetical order.

Categorized - Displays properties grouped by specific categories. Categories are defined and properties are assigned to those categories in Data Dictionary Manager.

Show Brief Properties - Displays those properties that have been previously specified as brief on the **Advanced Table Properties** dialog box.

Copy Bulk Properties - Copies the bulk properties of a selected item. The properties are then available to paste onto other items. For more information, see Copy Bulk Properties.

Paste Bulk Properties - Pastes copied bulk properties onto the newly selected item or items. You must have previously selected an item or items and copied bulk properties. For more information, see Paste Bulk Properties.

Display Null - Displays a null value in blank properties that currently contain no value. This option helps avoid confusion between empty cells and cells containing a zero.

Show Case Data - Displays a list of case conditions, for example, temperatures, pressures, and so forth. Case data includes both process cases and control cases, which are displayed for equipment, pipe runs or instruments, according to relevance. To see the case data that appears for corresponding items, look in Data Dictionary Manager.

Data Types of Properties

The data types of a property determine how you will enter data for and manipulate that property's value. The following is a list of data types that exist in the SmartPlant"

String – String data types allow you to simply type in the value that you want to assign to a property. For example, you may type in a Description or a Name for an item.

Select Listed – In SmartPlant P&ID, some properties are likewise restricted to preset values. These values for SmartPlant P&ID properties are called select-listed values; they reside in a select list. Select entries are the individual members of a select list. Select lists and their select entries are defined in SmartPlant Data Dictionary Manager

Some examples are Nominal Diameter, MOC Class (material of construction class) and Cleaning Reqr (Cleaning requirements).

Formatted (or unit of measure) – Formatted properties have a default unit of measure that will be used whenever you enter a value for them. For example, the default format for temperature in an English setting project is Fahrenheit. If you enter 100 for Insulation Temperature in the Properties Window, it will automatically give it 100 F. However, if you wish to enter data in a different unit of measure, you must first choose a format from the predefined list of units of measure and then enter the numeric value.

For formatted properties, such as estimated length or maximum operating temperature, you can enter a value without a format. Use a single quote, ', at the beginning of your entry. In this way, you can enter free text into a formatted property and no units of measurement are assigned.
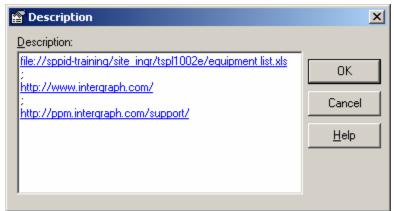
Hyperlink Documents or URLs

You may hyperlink documents or URLs to items by clicking on the **Calculation** button ⋯ in the **Description** field of an item and entering data.

The entry needs to be of the form of:

- file://servername/sharename/directory/document.extension
  OR
- http://www.intergraph.com

Data Import

SmartPlant P&ID allows you to import data from external sources. Currently, you can import data from the **Exchanger List**, and **Vessel List** from Zyqad, and from the **Equipment List**, **Pipe Run List**, and **Nozzle List** reports from SmartPlant P&ID. The imported spreadsheet must be in a specific format. To import data, click **File** > **Import** > **Data File**. SmartPlant P&ID looks for a match on **Item Tag** in the database. If a match is found, the item properties are updated in the database, and the changes are reflected in the **Properties window** for the item. If no match is found, the data and graphic symbol defined in the report are placed in the **Stockpile**. You can then place the defined item from the **Stockpile**.

System Editing: An Overview

System Editing works within the context of the active drawing. If you select **Tools > System Editing** and a check mark displays by the command, the function is active.

You can also turn on **System Editing** by clicking [icon]. When the command is active and you make additions or changes to a drawing, the changes are propagated according to relationships and rules.

Notes:

- **System Editing** is not supported from or to plant item groups such as loops and packages.

When a new item is placed from the Catalog or the Stockpile, creating a new relationship, property values flow across the new relationship as specified in the rules. The flow of property values is as follows.

- Properties - The properties that can flow across this relationship are those that are listed as consistency criteria in all of the applicable rules.

- Local Editing - A property value can be copied in either direction across the new relationship but is never spread any further.

When an existing item is modified and a new relationship is created, property values are not copied across the relationship. For example, if an existing instrument is dragged from its current position and dropped onto a different pipe run, propagation is not initiated. If the endpoint handle of an existing pipe run is dragged and dropped onto an existing piping component, propagation is initiated. Consistency checking is performed at the new relationship and any differences in property values are displayed as inconsistencies.

When a directional property, for example, Flow Direction and Slope Direction, is propagated, it is not enough to set the same value on all items the propagation scope. The direction of the underlying geometry must be compared to the direction of the geometry of the start item. One of the possible property values must be selected based on the direction of the geometry. In cases where the direction of the geometry cannot be compared (such as a midpoint branch condition) no propagation is possible.

The Scope of System Editing

Connectivity - All objects that are connected to the selected object are candidates for inclusion in the propagation (**System Editing**) process.

Rules - The consistency criteria specified in rules define which properties are propagated and in which direction. The propagations scope can be expanded across a relationship if there is an applicable rule for the property being modified.

Inconsistencies - The propagation scope is not expanded past and existing inconsistency for the property being modified. Using **System Editing** does not resolve inconsistencies. You must resolve the inconsistency.

Breaks - Propagation breaks can be defined for specified properties at specified relationships in a design. The propagation scope is not expanded pass a break for the property being modified.

Access Control - When working in a project, the propagation scope only includes objects that have been claimed.

System Editing and Consistency Checking

You can effectively propagate a changed property value across drawing boundaries. If a changed property value is propagated and it reaches an OPC, it simply stops at that point. An inconsistency indicator at the OPC shows that the value in the active drawing is inconsistent with the value on the connected pipe in the continuation drawing. To resolve this type of inconsistency, the user must open the continuation drawing. With the continuation drawing open, the user can double click on the inconsistency indicator to display the **Consistency Check** dialog. One of the solutions available for resolving inconsistencies is to copy the property value from the first drawing into the active drawing. When this solution is used, the property value from the pipe run in the first drawing is copied to the pipe run in the active drawing and spreads outward from there according to the standard propagation rules. Thus, property values can be propagated across document boundaries.

Inconsistencies - **System Editing** never expands across an inconsistency for the property being propagated. For a given set of connected items with a consistent property value, **System Editing** allows you to easily change that value to a new value. However, if the items are not consistent to begin with (inconsistencies exist), then **System Editing** will not automatically make them consistent. Other tools are available (such as **Solutions** on the **Consistency Check** dialog box) for resolving inconsistencies.

Break Components - Some components inherently limit the copying and suppress the consistency checking of certain properties. For example, a reducer is a break component for the Nominal Diameter property. When changing the Nominal Diameter in a pipeline, the spread of that change is limited by any reducers that exist in that pipeline. The consistency checking that would normally be done between the connected pipes is also suppressed at the reducer.

A break component can be created for any property or collection of properties of a pipe run. To create a break component, you must first create a special filter. This filter must apply to pipe runs and must include the properties to be broken as criteria in the filter. The value for each property in the filter is not important. The final step in creating a break component is to open the symbol in Catalog Manager and select the

Property Breaks command to select the filter. After creating a break component, when it is placed into a drawing, it breaks the specified properties.

Break Labels - The consistency criteria specified in the rules define the general behavior of the properties. Break labels provide a way to define exceptions to the general rules. A break label provides a visible and plottable symbol that signifies the end of one property value condition and the beginning of a new condition along a pipe. A changed property value is never propagated across a break label for that property. A break label also suppresses consistency checking for the property it breaks.

A break label can be created for any property or collection of properties of a pipe run. A break label is created in the Catalog Manager just like any other label, except that the Label Type property is set to **Attribute Break**. A SmartText field must be created for each property that is to be broken. In many cases the graphical shape of the label indicates which properties are broken. In these cases the Visible flag on the SmartText is set to False so that the text is not visible. It is also normal to turn the leader line on within the Catalog Manager so that it is automatically displayed at placement time. All of the symbols under Piping\Segment Breaks in the delivered catalog are break labels.

A break label can only be placed at the endpoint of a graphical pipe line where it connects to a component or a branch point. Furthermore, a break label can only be placed at a point where the specified properties are listed in the consistency criteria for the applicable rules. The break label stays attached to the point it was placed on and cannot be dragged away from it.

System Editing and Projects

When working in a project, the system enforces the rule that unclaimed objects cannot be modified. If you select an unclaimed object, the property grid does not allow you to enter new values for its properties. Similarly, System Editing does not allow property values to flow to unclaimed objects. If you have a claimed object that is connected to some unclaimed objects, and if you change a property on the claimed object, propagation will not change the property value on the unclaimed objects.

The items in the propagation scope are initially collected using the standard propagation rules. However, all unclaimed items are removed from the propagation scope before any property values are changed.

Items that are claimed for relationship are treated as unclaimed for the purpose of propagation.

Understanding System Editing and OPCs

An OPC can be placed as a freestanding item in a drawing or it can be placed onto an existing pipe run. If an OPC has been placed as a freestanding item, a pipe run can be attached to it. When the relationship between the OPC and the pipe run is created (either by placing the OPC or by placing the pipe run), the system finds the applicable rules, copies the specified properties and evaluates the consistency criteria.

When the first OPC (of an OPC pair) is connected to a pipe run, there is no pipe run connected to the mate OPC. Since there is no other pipe run, the system will not find any applicable rules.

When the second OPC (of an OPC pair) is connected to a pipe run, however, the system will find the rules that apply to this pair of pipe runs. The system copies the specified properties from the pipe run in the other drawing to the pipe run in the active drawing. This copy operation will not overwrite any existing values. (The **Copy** is forced into the **Copy If Null** mode.) The properties that are copied across the OPC are propagated into the active drawing. This can mean that property values on items other than the connected pipe run are affected. However, this is only done if the current value is **Null**. Existing values are not overwritten. After the copy process is complete, the consistency criteria is evaluated and inconsistencies are generated for all inconsistent properties.

Each time a drawing is opened, a routine is executed to update all of the OPCs in that drawing. Also, the relationship between the OPC and the connected pipe run is updated. During this update, the rule base is searched to find all of the applicable rules. If the mate OPC has been connected to a new pipe run since the last time this drawing was opened, it may find a different set of rules that apply. After finding the applicable rules, the consistency criteria is evaluated and inconsistencies are generated for all inconsistent properties. Property values are not automatically copied across OPCs during this processing. If you want to copy changed property values across OPCs, you can use the **Solutions** that are available in the **Consistency Check** dialog box.

The text in the label on the OPC is also updated. If the mate OPC has been placed into a different drawing since the late time this drawing was opened, this change will be reflected in the updated label.

System Editing and Changing Property Values Across OPCs

When a property value is changed on a pipe run that is connected to an OPC, it is expected that the connected pipe run in the continuation drawing should be changed

in a corresponding manner. You can achieve this result with a workflow that *pulls* the change into the other drawing.

This workflow allows you to *pull* a changed property value across an OPC into the continuation drawing. When a property value is changed on a pipe run that is connected to an OPC, an inconsistency is generated at the OPC. This inconsistency indicates that the two connected pipes have different values. To resolve this inconsistency, you must close the active drawing and open the continuation drawing. In that drawing, another inconsistency is shown at the mate OPC. You can double-click on that inconsistency to display the **Consistency Check** dialog box. A solution is displayed to **Copy** the new value from the first drawing onto the pipe run in the active drawing. If you apply this solution, the value is copied into the active drawing and propagated to the related items. This resolves the inconsistency in the second drawing. The next time the first drawing is opened, the inconsistency will be resolved there as well.

Tools > System Editing

When System Editing is enabled for a plant, changed property values are automatically spread or propagated to related items according to the currently defined consistency criteria. In certain cases, you may want to set a property value on one item only and not allow it to be propagated to any related items. The System Editing command (on the Tools menu and the toolbar) allows you to turn off the propagation functionality temporarily during a design session. After the change has been made, you can use this same command to turn it back on again. System Editing is always turned on when you start a new design session.

System Editing can be enabled or disabled for the entire plant by means of the Enable System Editing setting that is exposed through Options Manager. For new plants, the value of this setting is True by default. For upgraded plants, the value of this setting is False. This allows you to enable System Editing for an upgraded plant at the time of your choosing.

Adding Design Elements

The Design category encompasses a range of elements that may be used on a P&ID such as Annotations (or Notes), Area Breaks, Packages, and Revision Clouds. Title block labels are also contained in this category.

Using Annotations: An Overview

Annotations are model items used to add notes, remarks, and other information to your drawing. You can use annotations to include any type of information previously

inserted into a drawing using primitive commands. Annotations include everything from geometric shapes to extension and dimension lines.

Because annotations are frequently parametric, often you can resize them to fit your needs.

Placing annotations works like placing any other component in a drawing. When you place an annotation, which has an item type of item note, on a drawing, you can then place a label on the item note. These labels allow you to enter the remarks, notes, and other information that you need to effectively annotate your drawings.

Notes:

- You cannot annotate drawings by opening them in SmartSketch and adding dumb graphics to the drawing. Opening and saving a .pid file in SmartSketch can corrupt the file, making the drawing unusable.

- Annotations are a way to place free remarks in a drawing. There are several design labels of different formats available in **Catalog Explorer**. Most of these labels require placing an annotation point or other design graphic on the drawing first and then attaching a label to the graphic. The plain annotation point is located in the **Graphics** folder under the **Annotation** node; however, there is one item in the **Labels** folder with an annotation point built into it. It is called **Item Note & Label**, and it looks like this:

Annotation Point   Item note

- It is possible to associate the **Item Note & Label** with a P&ID object such as Equipment, Valve, etc. If you delete the P&ID object and the **Item Note & Label** is associated to this object the **Item Note & Label** will also be deleted.

Place an Annotation

1. In the **Catalog Explorer** list view, select the annotation that you want to place from the **Graphics** folder.

   a. Note the **Item Note and Label** from the **Design** > **Annotation** > **Labels** folder has a built-in **Annotation Point**.

2. Click to place the **Annotation** at the appropriate place in the drawing or drag the annotation over a drawing item, **Equipment** or **Piping Component**, to "link" the **Annotation** to the item.

3. Press **Esc** to quit placement mode.

4. To add text to the **Annotation**, place a label from the **Design** > **Annotation** > **Labels** folder on the **Annotation**.

    a. Note, if you did not originally place an **Item Note and Label** from the **Design** > **Annotation** > **Labels** folder.

5. Assign text to the **Annotation** in the **Note Text** property box in the **Properties** window.

Select Associated Item Command

The **Edit** > **Select Associated Item** command will highlight the associated item when an **Item Note and Label** is associated with a drawing item, such as Equipment or Piping Component. This command is also available from the shortcut menu when the annotation point of the Item Note and Label is selected.

Packages and Systems

Package, Contract Package, Hydro Test Package, Test Systems, Safety Class and Hydraulic Circuits are similar to Instrument Loops in that they are a group of one or more items with a common set of properties. For example, you may want to have a package to denote items that will be included in a palette. A system could be used to denote the portions of a P&ID used for testing prior to startup. To associate items with a Package, Contract Package, Hydro Test Package, Test Systems, Safety Class, or Hydraulic Cicuits you will use the Pkg Item Tag property, Contract Package No, Hydro Test Package No, Test Sys Item Tag property, Instr Safety Class Item Tag or Hydraulic Circuit Item Tag respectively.

Place a Package

1. Right-click the package symbol in **Catalog Explorer** in the **Design** node.

2. From the shortcut menu, select either **Send to Stockpile** or **Send to Drawing Stockpile**, whichever is appropriate.

3. In the **Properties** window, enter a value for the **Item Tag** property or enter a value for the **Item Tag** in the **EDE**.

4. In the Drawing view, select items to be part of the package.

5. In the **Properties** window, choose **Select Set** from the **Properties** box.

6. In the **Properties** window, click in the **Package Item Tag** box and select the appropriate value displayed list.

   📝 **Notes:**

   - Some catalog items do not possess the Package Item Tag property. Do not include those items in your select set because the package item tag does not appear in the Properties window if any member of the select set does not possess that property.

   - Plant item groups, such as instrument loops, packages, safety classes, and so forth, frequently reside in a stockpile with their members residing on drawings. Moving the plant item group itself to another stockpile has special limitations:

     i. **From a drawing stockpile to the Stockpile** - No constraints apply. This action can occur at anytime.

ii. **From the Stockpile to a drawing stockpile** - If the plant item group contains only items in the destination drawing or does not contain items in any drawing, this action is allowed.

iii. **From a drawing stockpile to another drawing stockpile** - Same constraint as above applies.

Area Breaks

An area break is a graphical fence around an area of a drawing where a package or system may be applied.

Place an Area Break

1. Select the area break in Catalog Explorer in the Design node.

2. Place the area break shape around items in the Drawing view for which this area break applies.

   Notes:

   - You can place a rectangular area break by dragging the pointer in the Drawing view, like fencing items with the **Select Tool**.

   - Or you can place a multi-sided area break by clicking at a starting point and then clicking for each vertex of the shape.

3. Press **Esc** to quit placement mode.

4. Create a select set of the items that you want to include in the area break. Including the area break shape itself or not, this all depends upon which property your assigning a value for to the item you have selected.

5. In the **Properties** window, specify the properties for the select set.

   Notes:

   - You can nest area breaks that have different values for the same property with one restriction: one area break must completely enclose the other area break. The value for the inside area break overrides the value for the outside area. For example, for a group of items, you can designate a portion as **Supply By Contractor** and a smaller portion as **Supply By Owner**.

Hydraulic Circuits
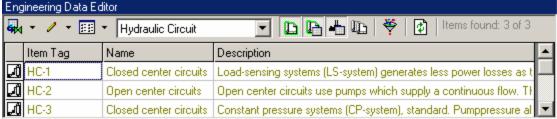
With Hydraulic Circuits you can define many-to-many relationships between the Hydraulic Circuit and SmartPlant P&ID drawing items; that is, multiple Hydraulic Circuits can be associated with Pipe Runs or Vessels.

Associate Hydraulic Circuits with an Item
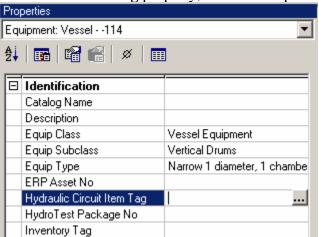
✎ **Notes:**

- Before selecting the pipe run for associating hydraulic circuits, open Options Manager and make sure that Hydraulic Circuit appears among the items included in the **StockpileItems** property.

1. In the **Catalog Explorer** tree view, select the **Symbols** > **Design** node.

2. In the lower pane, select **Hydraulic Circuit**, right-click, and on the shortcut menu, click **Send to Stockpile**.
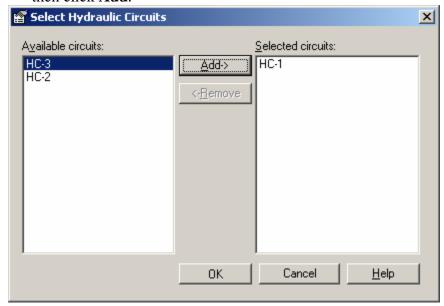
3. Assign a value for the hydraulic circuit's **Item Tag** property.

4. Select one or more items and in the **Properties** window, beside the **Hydraulic Circuit Item Tag** property, click the ellipsis.

5. On the **Select Hydraulic Circuits** dialog box, from the **Available circuits** list, select a hydraulic circuit that you want to associate with the item, and then click **Add**.

6. Repeat the previous step for each additional hydraulic circuit that you want to associate with the item.

7. Close the dialog box and beside the **Hydraulic Circuit Item Tag** property, view the hydraulic circuits that you added.

Select Hydraulic Circuits Dialog Box

Allows you to select one or more hydraulic circuits for an item of equipment, an equipment component, or a pipe run. This dialog box opens when you select the item and click the ellipsis beside the **Hydraulic Circuit Item Tag** property in the **Properties** window.

Available circuits – Displays the list of available hydraulic circuits that are not selected for the item. To select one or more hydraulic circuits, highlight them in this window one at a time and click **Add**.

Selected circuits – Displays the list of hydraulic circuits that have been selected. To deselect one or more hydraulic circuits, highlight them in this window one at a time and click **Remove**.

Revision Clouds

A revision cloud is a graphical cloud indicating a change in one or more items or attributes contained within the cloud. You must place a revision triangle to add the Revision number property to the cloud.

Place a Revision Cloud

1. Using the Catalog Explorer, select the **Symbols > Design > Revision Cloud.**

2. Click the starting point in the drawing for the revision cloud area.

3. In a clockwise direction, continue clicking to place vertices on the revision cloud.

4. Right-click to complete the revision cloud.

5. Set the **Area Break Style** property to either a **Cloud - Large** or **Cloud - Small**.

   📝 **Notes:**

   - You can change the symbology, the line width and line style, using the **Symbology** option in **Options Manager**.

Design Reference Data

In the delivered reference data, the following categories exist under Design:

- **Annotation:** This category contains the annotation point, annotation labels, and geometric shapes.

Auxiliary Graphics: An Overview

You can use Auxiliary Graphics to add simple graphics to your SmartPlant P&ID drawings. When you have completed adding graphics using Auxiliary Graphics, these graphics display in your SmartPlant P&ID drawing inside a SmartFrame. The SmartFrame and its contents are stored in the database.

To use Auxiliary Graphics, select Edit > Insert > Auxiliary Graphics. Selecting the command opens a drawing sheet that also displays your SmartPlant P&ID drawing. You can then create graphics using the displayed Draw toolbar. Existing objects in your current SmartPlant P&ID drawing cannot be selected or edited but do display for viewing purposes. To return to your SmartPlant P&ID drawing, select File > Close Auxiliary Graphics. When you return to SmartPlant P&ID, you can only move the graphics placed using Auxiliary Graphics. To edit the graphics again, you can either double-click on the SmartFrame, right-click and select Edit, or select Edit > Insert > Auxiliary Graphics.

The Auxiliary Graphics Draw toolbar contains commands for working with continuous lines and arcs, tangent arcs, curves, free form drawing, circles, rectangles, fillets, text and so forth. The Draw toolbar displays down the left side of the drawing sheet. Note that any SmartPlant P&ID command not available during the Auxiliary Graphics session is disabled.

You can customize the Auxiliary Graphics Draw toolbar using Tools > Customize. The available commands display with (Auxiliary) following the command name. Toolbar choices include:

Draw (Auxiliary)

Features (Auxiliary)

Trim (Auxiliary)

Extend (Auxiliary)

Group (Auxiliary)

Manipulation (Auxiliary)

Priority (Auxiliary)

Edit (Auxiliary)

Relations (Auxiliary)

Text (Auxiliary)

For information about how to customize a toolbar, refer to Customize Command (Tools Menu) in the online Help.

Notes

Any customizations made to the Draw toolbar are not saved when you close the Auxiliary Graphics session.

The Undo buffer is cleared each time you enter an Auxiliary Graphics session. The SmartPlant P&ID Undo buffer is also cleared at this time.

Auxiliary Graphics are turned on and off with Notes. By clicking **View>Properties>Notes**, the display of notes is turned on or off.

Drawing and Editing in Auxiliary Graphics Mode

The drawing, editing and other commands match those used for SmartSketch. However, only basic RAD commands are included. Commands exclusive to SmartSketch are not included.

The leader and balloon commands were excluded for Auxiliary Graphics. However, double-clicking on a graphic element in Auxiliary Graphics will bring up a cursor so the user can create a SmartLabel. This will be accomplished by using a SmartSketch Components AddIn, or by creating our own AddIn, using the SmartSketch Add in as an example. The double-click event of all drawing types will be overridden to start this command.

Editing Auxiliary Graphics

Click **Edit > Insert > Auxiliary Graphics**. A drawing sheet displaying your SmartPlant P&ID drawing displays. You can then create graphics using the displayed Draw toolbar. Exiting objects in your current SmartPlant P&ID drawing cannot be selected or edited but do display for viewing purposes.

Exiting Auxiliary Graphics

Click File > Close Auxiliary Graphics to return to your SmartPlant P&ID drawing.

When you return to SmartPlant P&ID, you can only move the graphics placed using Auxiliary Graphics. To edit the graphics again, you can either double-click on the SmartFrame, right-click and select Edit, or select Edit > Insert > Auxiliary Graphics.

Toolbars

When you enter Auxiliary Graphics mode, the Draw toolbar is displayed as the default toolbar. A toolbar called Change and one called Relationship are also created, but are initially hidden. They can be activated from the Toolbars dialog.

Draw Toolbar

The Draw Toolbar contains Select Tool, Line/Arc Continuous, Tangent Arc, Curve, Circle By Center Point, Rectangle, Point, and Text Box.

Change Toolbar

The Change Toolbar contains Move/Copy, Offset, Rotate, Mirror, Scale, Bring To Front, Send To Back, Pull Up, and Push Down, Group, Ungroup.

Relationship Toolbar

The Relationship Toolbar contains Connect, Horizontal/Vertical, Parallel, Perpendicular, Tangent, Colinear, Concentric, Symmetric, Equal, Lock, Relationship Handles, Alignment Indicator.

Customizing Toolbars

You can customize the Auxiliary Graphics Draw toolbar using Tools > Customize. The available commands display with (Auxiliary) following the command name. Toolbar choices include:

Draw: includes the drawing commands: Line/Arc Continuous, Tangent Arc, Arc By 3 Points, Arc By Center Point, Circle By Center Point, Circle by 3 Points, Tangent Circle, Ellipse By Center Point, Ellipse by 3 Points, Rectangle, Curve, Point, FreeSketch, and FreeForm.

Features: contains the commands Fillet and Chamfer.

Trim: contains the commands Trim and Trim Corner.

Extend: contains the Extend to Next command.

Group: contains the commands Group, Ungroup, Rectangular Pattern, and Circular Pattern.

Manipulation: contains the commands Offset and Scale.

Priority: contains the commands Bring To Front, Send To Back, Pull Up, and Push Down.

Edit: contains the commands Paste, Redo, Delete, Undo List, Redo List, and Edit Properties.

Text: contains commands Text Box, Fill, and Character Map.

SmartSketch Settings
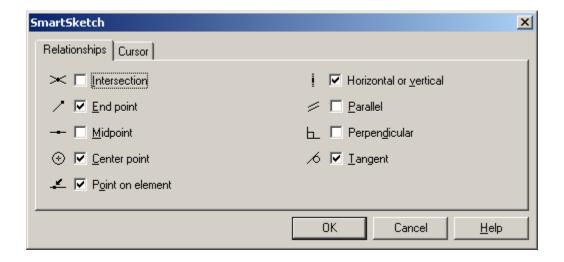
The SmartSketch dialog will also be available for Auxiliary Graphics. This dialog controls which glyphs are shown while the user is drawing. It also controls how close the cursor must be to an element for the glyphs to appear. The dialog is activated by selecting Tools>SmartSketch Settings in the SmartPlant P&ID menu bar while in Auxiliary Graphics mode. During normal SmartPlant P&ID mode, the option will be disabled, or not selectable.

Relationships Tab

This tab allows you select the relationships that are recognized by the software as you draw. Set the relationships you want to recognize, and clear the relationships you do not want to recognize. Clear all relationships so that the software does not place relationship handles.

relationship handle

Graphic used to represent a geometric relationship between elements, key points, and dimensions, or between key points and elements. The relationship handle shows that the designated relationship is being maintained.

Cursor Tab
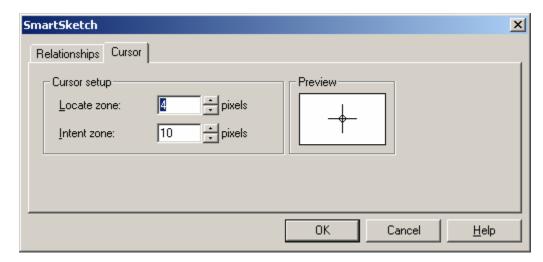
Cursor Setup - Sets the sizes of the Locate Zone and Intent Zone around the pointer.

Locate Zone - Sets the size of the locate zone radius. The locate zone is a region around the pointer.

The software recognizes relationships based on elements within the locate zone so that you do not have to move the pointer to an exact position. For example, if part of an element is within the locate zone, the software recognizes a Point On relationship. The size of the locate zone is indicated by a circle around the center of the pointer crosshair. Values from 3 to 12 pixels are valid.

Intent Zone - Sets the size of the intent zone radius. Intent zones allow drawing commands to interpret your intentions as you draw. Values from 3 to 12 pixels are valid.

Preview - Shows the size of the locate zone and the symbol for the selected relationship.

Modifying and Moving Placed Items: An Overview

After you place items in a drawing, you can modify the items in several ways. Obviously, you can move items or groups of items from one location to another in the drawing. If you want to change the orientation of items, you can mirror and rotate most items. You can also move groups of items to another open drawing in the same plant hierarchy.

Depending on the rules defined for particular items in the drawing, limits exist for possible placements. For example, specific trays can be flagged for placement inside a piece of equipment, and you can only move these inside the equipment. Rule Manager defines rules that determine the way that model items interact during manipulation. The software processes rules defined for items as you move them, as well as you place them.

You can resize or scale parametric items by using their parametric handles. Resizing parametric items maintains certain characteristics of the item, such as angles between elements of the symbol, while it allows modification of other characteristics of the item, such as the length of its sides. Many items are now scalable, too, and can be resized by using their scale handles.

You can modify the properties of each item by selecting the item and changing individual properties in the **Properties** window. Also you can edit some item properties in the **Engineering Data Editor** in the **Design** window by selecting a table cell and filling in the appropriate information. You can select an entire row by clicking on the corresponding row number to the left of the table and then viewing the properties in the **Properties** window and editing them there.

If you want to remove an item from a drawing, you can delete the item. If the plant model requires the deleted item, the item appears in the stockpile for later placement. If the plant model does not require the item, it disappears from the drawing completely. You can permanently remove required items from the plant model using the **Delete from Model** command.

📝 Notes:

- When modifying the fluid code properties of a line, you select a fluid code from the select list. The fluid code list displays only codes applicable to the type of Fluid System that is defined. To redisplay a complete list of available fluid code values, delete the property for the fluid system, and then the complete list of fluid code values appears.

Select an Item

1. On the main toolbar, click the **Select Tool**.

2. Do one of the following:

    *   To select one item, click it.

    *   To select more than one item, hold **Shift** or **Ctrl** and click each item.

    *   To select more than one item at once, drag to fence the objects.

    *   To select one of several overlapping items, use **PickQuick**.

3. To clear the selection of an item or items, do one of the following:

    *   Click in an empty portion of the drawing.

    *   Right-click in an empty portion of the drawing.

    *   Select another item without holding **Shift** or **Ctrl**.

    *   To clear the selection of one item and leave other items selected, click the item while holding **Shift** or **Ctrl**.

        📝 **Notes:**

        *   When the **Select Tool** is active, selectable items are highlighted as you pass the pointer over them. When the item you want to select is highlighted, click to select it.

        *   Click **Tools > Options** to change the item highlight and selection colors and locate and break-away tolerances.

        *   To select all items in a drawing, right-click a blank area in the drawing and click **Select All** on the shortcut menu, or, to accomplish the same thing, press **Ctrl + A** or click **Edit > Select All**.

        *   Also, you can select items in the **Engineering Data Editor**. If they reside in the active drawing, they are selected in the **Engineering Data Editor**. You can build up a select set using this method, too.

Move an Item Using the Move Command

1. Select one or more items.

2. Select the **Edit** > **Move** command or on the main toolbar, click **Move**.

3. Click to define the *from point*

4. Click to define the *to point* or define the Step Distance or X and Y location in the **Move Ribbon.**

    ✎ **Notes:**

    *   If you want to copy the items, press **Ctrl** when you place the item.

    *   If you move an item that owns other items, the owned items move also. For example, if you move a vessel that owns trays and nozzles, the trays and nozzles move with the vessel.

    *   You can drag an item without using **Move**. Select the item and then drag it to its new location. If you want to copy the item, press **Ctrl** while you drag.

    *   If you want to move an item to another drawing, move it to the Stockpile, open the second drawing, and place the item from the stockpile or use the **Move To Drawing** command.

    *   You can use other view manipulation commands, such as **Zoom**, **Fit**, and **Pan**, while you are using the **Move** command. When you finish manipulating the view, the software returns you to the **Move** command at the point where you were when you started manipulating the view.

Move Ribbon

Specifies the *from point* and *to point* when you move items.

Copy - Copies the items in the select set when you move them.

Step Distance - Increases or decreases the value in the ribbon boxes. For example, typing a step value of 0.25 and pointing away from the *from point* increases the distance in discrete steps, from 0.25 to 0.5, 0.75, and so forth.

X - Allows you to enter an explicit value for the distance to move in the x-coordinate, or you can simply observe the value as you drag the object.

Y - Allows you to enter an explicit value for the distance to move in the y-coordinate, or you can simply observe the value as you drag the object.

Select Tool Command

The **Select Tool** command from the Main toolbar changes the pointer to the arrow-shaped selection pointer so that you can select, modify, and manipulate items. The circle at the end of the pointer arrow is the locate zone. While the Select Tool is active, the **Select Tool** ribbon is displayed.

Move an Item With the Select Tool

1. On the main toolbar, click the **Select Tool**   .

2. Position the pointer over the item but not over any of the handles of the item.

   📝 **Notes:**

   - Dragging a handle modifies the item instead of moving it.

   - When the **Select Tool** is active, selectable items are highlighted as you pass the pointer over them. When the item you want to select is highlighted, click to select it.

3. Drag the item to its new position.

Copy an Item With the Select Tool

1. Click **Select Tool** on the main toolbar.

2. Select an item.

3. Position the pointer over the item, but not over any of the handles of the item.

4. Hold **Ctrl** and drag the copy to its new location. Placement rules are in effect, and so you must place the copy in an allowed position as if it were a new catalog item.

Select Tool Ribbon

Appears when nothing is selected. After you select an item, the **Select Tool** ribbon is replaced with a ribbon for editing the selected item.

Inside - Specifies that items inside the fence are selected.

Overlapping - Specifies that items both overlapped by the fence and inside the fence are selected.

Top Down - Specifies that symbols are recognized in the locate zone as opposed to individual elements in a symbol.

Bottom Up - Specifies that individual elements, which make up symbols, are recognized in the locate zone as opposed to the whole symbol.

Move To Drawing Command

The **Edit > Move To Drawing** allows you to move a select set of drawing items into another open drawing. These conditions must be met in order for this command to be available:

- The source and destination drawing must be open, and neither of these drawings can be read-only.

- The drawing objects to be moved must be selected.

- The selected items must not have a connection to any items outside the select set.

- You must have the appropriate permissions to modify the items in the select set.

If more than one possible destination drawing is open, the **Move To Drawing** dialog box appears, and you can select the appropriate destination drawing.

The **Move To Drawing** command manipulates the select set in order to make sure the selected items are appropriate items to move. The command removes these objects from the select set:

- Inserted OLE objects are automatically removed from the select set.

- Title blocks are automatically removed from the select set.

- If there is a label in the select set and the item that it labels is not in the select set, that label is removed.

- If there is a dependent object (such as a nozzle, equipment component, actuator, or item note) in the select set, and if the parent object that it is dependent upon is not in the select set, it is removed.

The command expands the select set to include the following objects:

- If there are labels that are not in the select set but are attached to objects in the select set, these labels are automatically added to the select set.

- If there are dependent objects (such as a nozzle, equipment component, actuator, or item note) <u>not</u> in the select set but they are dependent on objects in the select set, these objects are automatically added to the select set.

OPCs cannot be moved from one drawing to another using this command.

Move To Drawing Dialog Box

Allows you to choose the destination drawing that you want to place a item(s) into. This dialog box displays only when you have more than one possible destination drawing open. You must also have the item(s) selected that you want to move. Then, click **Edit > Move To Drawing** and click in the drawing to define the **Select from point**. The **Move to Drawing** dialog box displays. It contains a list of possible drawings that your item(s) can be moved to. Select the drawing and click **OK**.

Available drawings - Lists the open drawings that are available to be a destination for the **Move To Drawing** command. Those drawings that are not included in the list are the source drawing, open read-only drawings, open drawing templates, and open assemblies.

Move a Select Set to Another Open Drawing

1. Open the destination drawing if it is not already open.

2. Select the items that you want to move into the destination drawing.

3. Click **Edit > Move To Drawing**.

4. In the Drawing view, click the *from* point.

   📝 **Notes:**

   - For the purposes of placement in the destination drawing, the *from* point becomes the origin of the select set.

   - While you are using a *from* point (or *to* in the destination drawing), the pointer is a crosshair.

5. If more than one possible destination drawing is open, then select the desired destination drawing from the **Available Drawings** list on the **Move To Drawing** dialog box.

In the Drawing view of the destination drawing, click the *to* point.

Copying Items: An Overview

Copying an item makes it possible to use a drawing item that is already in place as a starting point for placing a new item, without removing the current item from the drawing. Several methods exist for doing this, such as using the **Copy** button on the main toolbar or using **Ctrl** while you move, mirror, or rotate a drawing item.

Remember that placement rules are in effect when you are copying an item to a new location, just as they are when you originally place an item from a stockpile or from **Catalog Explorer**. So a copied nozzle must be placed on a piece of equipment, for instance.

Manipulating Items With Component Handles: An Overview

Many components allow you to change their shape or orientation using component handles. Component handles allow you to rotate or mirror a component around axes, and you can use component handles to scale or resize parametric items in a drawing. Items can have several different types of component handles.

- ✚ The **mirror handle** allows you to mirror about the horizontal and vertical axes, defined in relation to the origin of the item. You can mirror items by dragging the mirror handle in the appropriate direction. The software shows you the result of the operation dynamically so that you can be sure to mirror the item correctly.

- ● The **rotate handle** provides a mechanism for rotating components around their origin. You can rotate items by dragging the rotate handle in the appropriate direction. The software shows you the result of the operation dynamically so that you can be sure to rotate the item correctly.

- □ **Parametric handles** allow you to modify different parts of parametric items. Standard parametric handles represent up to four driving dimensions that have been applied to an item. Parametric handles can be located in four standard positions: top center, bottom center, left center, and right center of the range of an item. You can scale the item parametrically by dragging any parametric handle. The software shows you the result of the operation dynamically so that you can be sure to scale the item correctly.

- ■ **Scale handles** allow you to change the scale of a drawing item. A scalable symbol displays four solid square scale handles. A handle is displayed at each corner of the range of the symbol. You can drag any one of the handles, and the symbol scales either larger or smaller depending on the direction that you drag the handle.

Not all drawing items in can be mirrored, rotated, or scaled. Catalog Manager defines these properties for items when the symbols for the items are created. Note also that select sets cannot be mirrored or rotated with component handles. The **Mirror** and **Rotate** commands on the toolbar can be used for select sets.

Mirror Command

Mirror an Item Using Its Mirror Handle

1. Select an item in the drawing.

2. Click the mirror handle ✚.

3. Drag the mirror handle in the direction that you want to mirror the item.

Edit > Mirror

The **Edit** > **Mirror** command reflects one or more selected items about a line or axis that you define. You can mirror without copying, or mirror and copy.

To mirror an item using a defined mirror axis, click **Mirror** on the main toolbar. The software reflects one or more selected items about that axis. You can mirror the item itself, or you can make a copy of the item in the mirrored position.

Mirror an Item About a Mirror Axis That You Define

1. Select one or more items.

2. Select **Edit** > **Mirror** or from the main toolbar, click Mirror.

3. Define one end of the mirror axis by clicking in an appropriate empty portion of the drawing or by clicking a keypoint **Error! Objects cannot be created from editing field codes.**. The software displays the mirror axis and the mirrored items dynamically (that is, as you move your pointer).

4. Position the pointer so that the mirrored items appear where you want them, and then click.

   📝 **Notes:**

   - If you want to copy the mirrored items, hold **Ctrl** when you click to place your items.

- Instead of positioning the mirror axis dynamically, you can use the **Position Angle** box on the **Mirror** ribbon.

Mirror Ribbon

The **Mirror Ribbon** appears on the main toolbar when the **Mirror** command is active, and allows you to either specify mirroring options or to observe mirroring options.

Copy - Copies the item or items when you mirror them.

Position Angle - Sets the angle of the mirror axis. The origin of the angle measurement is the point that you clicked for the beginning of the mirror axis. Setting the position angle to zero extends the axis horizontally to the right of the screen, 90 extends the axis vertically to the top, 180 extends the axis horizontally to the left, and 270 extends the axis vertically to the bottom.

Rotate Command

Rotate an Item Using Its Rotate Handle

1. Select an item in the drawing.

2. Click the rotate handle ●.

3. Drag the rotate handle in the direction that you want to rotate the item.

   Notes:

   - You can click the rotate handle closer to or further from the center of the item to change the step angle for the rotation.

Edit > Rotate

The **Edit** > **Rotate** command rotates one or more items a precise distance or angle about a specified point.

To rotate and item, select the item and then click **Rotate**. To define the rotation axis, you must specify two points by clicking in the drawing.

First Click - Defines the rotation point.

Second Click - Defines the rotation handle.

The software then dynamically displays a reference axis for the rotation. If you want to define a precise location for the rotation, you can enter values in the **Rotate** ribbon. You can enter increments in the **Step Angle** box if you want to control the increments of the rotation. For example, if the box is set to 30.0, the rotation is displayed in 30 degree increments. The default setting is 0 degrees so that the rotation is fully dynamic, that is, it can take on any value. The **Rotation Angle** box on the ribbon displays the angle of change between the old and new position. Angles are always displayed as a positive value.

Third Click - Completes the rotation of the item.

Rotate an Item About an Axis You Define

1. Select one or more items.

2. Select **Edit** > **Rotate** or on the main toolbar, click **Rotate**.

3. Click at the center of rotation.

4. Then, click to define the other end of the rotation axis.

   📝 **Notes:**

   - The software dynamically displays the rotation axis and items being rotated. The location and position of the rotation axis defines the rotation *from point*.

5. Rotate the items into position by dragging the rotation axis.

6. Click to define the rotation to point.

   📝 **Notes:**

   - To rotate by arbitrary increments, type a value in the **Step Angle** box on the **Rotate** ribbon.

   - In order to copy the rotated items into the new position, you can hold **Ctrl** while you click to define the to point.

Rotate Ribbon

The **Rotate Ribbon** appears when the **Rotate** command is active, and allows you to either observe rotation options as you rotate an item or specify rotation options.

| 🗗 | Step angle: | 0.00 deg | ▼ | Rotation angle: | 0.00 deg | ▼ | Position angle: | 0.00 deg | ▼ |

Copy - Creates a copy of the item that you are rotating.

Position Angle - Displays the angle between the horizontal axis through the center of rotation (A) and the point that you rotate from (B). Then, if you rotate the item, the position angle (C) changes to become the angle between the horizontal axis and the point to which you rotate. The position angle is always a positive value measured counter-clockwise from the axis.

Rotation Angle - Defines the rotation angle. The rotation angle is the angle between the point from which you rotate and the point to which you rotate. In the following picture, the center of rotation (A) shows the point (B) from which you rotate the item.

The rotation angle (C) is the angle of change between the old and new positions of the rotated item. The value is always positive.

Step Angle - Specifies the rotation step angle. The step angle specifies the increments, in degrees, that an item rotates on a rotation axis. You can enter increments in the **Step Angle** box. For example, if the box is set to 30.0, the rotation is displayed in 30 degree increments. The default setting is 0 degrees so that the rotation is fully dynamic.

Scale a Parametric Item

1. Select the parametric drawing item that you want to resize.

2. Drag one of the parametric handles in the middle of each side of the item to change the size of the item.

3. Continue dragging the parametric handles until the item is the size that you want.

   📝 **Notes:**

   - Certain properties of the parametric item, such as angle values, remain the same when you resize it.

Scale a Drawing Item

1. Select your scalable item in the Drawing view.

   📝 **Notes:**

   - A scalable symbol includes four solid square scale handles. A handle is displayed at each corner of the range of the symbol.

   - Scalable symbols are created by setting the **IsScalable** property of the symbol to **True** in Catalog Manager.

2. Drag any one of the handles to make the symbol either larger or smaller, depending on the direction that you drag the handle.

Label alignment

After two or more labels are selected, a user may use the label alignment toolbar buttons to arrange the labels with respect to the others in the group or with respect to the labeled item. The alignment buttons use the range of the labels excluding the leader lines. So, if the label has only a text box then the ranges of the label and text box are the same; however, if the label has a text box surrounded by a rectangle, the ranges of the label will be different than only the text box (in this case the entire range of the label is used).

Rotation angle: 90.00 deg

Alignment choices:

Align Left – aligns the labels so that the left-most line of the range is aligned

Align Right – aligns the labels so that the right-most line of the range is aligned

Align Center about Vertical – calculates the center points of each label's range in the vertical direction and places them in a line

Align Center about Horizontal – calculates the center points of each label's range in the horizontal direction and places them in a line

Align Top - aligns labels so that the top of the ranges are in line

Align Bottom – aligns labels so that the bottom of the ranges are in line

Copy labels

After a label has been placed, a user may copy that label and place it on another item in the drawing. The user would select the label, right-click and choose **Place Ne**w. This will start the label placement process, using one or two-point placement and other placement characteristics for the label.

1. In the Drawing view, select the label that you want to copy.

2. Right-click the label and select **Place New** from the shortcut menu or click from the Run Macro toolbar.

3. Associate the new label icon that appears at the end of the pointer with the new item by highlighting the item and clicking.

Undo Command

The **Edit > Undo** allows you to reject the last action that you completed.

Up to ten operations are saved in the undo list, and you can undo them by repeated use of the **Undo** command. The undo list is cleared if you perform any of the following actions:

- Change the properties of the Drawing view on the **View Properties** dialog box

- Turn the display of inconsistency indicators off or on by clicking **View > Show Inconsistencies**

- Click **File > Save**

- Open a different drawing or toggle to another open drawing

- Open a new Drawing view or new **Engineering Data Editor** (**EDE**)

- Using commands in the **EDE** or on the Connector shortcut menu, move a connector to another drawing stockpile

- Changing or viewing the claim status of drawing objects

An undoable action is defined in several ways. Running a command is usually an action. Modifying item properties is an action, and the action ends when another item is selected. This definition of an action holds true for select sets, too, and for multiple items selected in the **EDE**. That is, modifying the properties of a select set is one action in and of itself, not one action for each member of the set.

The **Undo** command does not reverse view manipulations; however, the **View > Previous** command is still available for that purpose.

Undo Your Last Action

1. Click **Undo**.

   📝 **Notes:**

   - Pressing **Ctrl** + **Z** also undoes your last action.

   - The **Undo** command stores up to 10 actions making this command available repeatedly.

   - If the **Undo** button is not available, then the undo list has been cleared, and you can no longer undo your last action.

Finding and Replacing Drawing Items: An Overview

The software has the ability to find items and inconsistency indicators in your drawing. You can define search criteria based on a catalog item or an item type already in place in your drawing. Or, you can define criteria based on a delivered filter or on a user-defined filter. Having found all drawing items of a certain type, you can modify their properties or reposition them or even replace them with an item of a compatible type.

The software also searches your drawing for inconsistency indicators so that you can review and correct inconsistencies systematically.

Replacing drawing items is straightforward, too. You have the capability to replace a single drawing item with a comparable item, replace many items one-by-one, or even replace all items of one type at the same time.

When you find or replace drawing items, the selected items are added to a select set, and so you can view the common properties of the select set items in the **Properties** window and edit them if appropriate.

The **Find** and **Replace** commands are not available in the **Engineering Data Editor**.

Find and Replace Dialog Box

Sets options for searching for and replacing drawing items. This dialog box opens when you click either **Edit > Find** or **Edit > Replace**. Depending on your command choice, you access one of the following tabs.

 Notes:

- The **Replace** command does not allow a non-breaking component to be replaced with a break component. Piping components and instruments can be defined in the catalog as break components. Reducers are good examples of break components. When a break component is placed into a pipe run, the pipe run is automatically broken at that point. A break component suppresses the consistency checking and limits the propagation of the properties that are broken. The Property Breaks command in the **Catalog Manager** is used to define a catalog item as a break component.

- You can use the **Replace** command or the **Find and Replace** dialog box to replace a multiple representation symbol providing the definition attributes for the source (parent) and target (child) are identical. The definition attributes are **Equipment Class**, **Equipment SubClass**, and **Equipment Type**.

Find Tab (Find and Replace Dialog Box)

The **Find Tag** allows you to search the drawing for specified items.

Find what - Lists search criteria, and allows you to enter new search criteria or select from a list of the five most recently used criteria. The items listed above the dashed line in the list are used to set new search criteria by searching using the **Catalog Item**, **Filter**, or **Inconsistency** options. Selecting the **Catalog Item** option opens the **Select Catalog Item** dialog box. Selecting the **Filter** option opens the **Select Filter** dialog box.

Search in - Defines the scope of the search. You can search the active drawing or just the contents of the active window. If the **Active Window** option is selected, the **Scroll mode** feature is not available. If the **Active Drawing** option is selected, the **Scroll mode** feature is automatically set to **Zoom**, and you can select a preferred scroll mode.

Scroll mode - Defines how the active view should be changed when a drawing item is found. If you select **Zoom**, the active view is zoomed to a multiple of the range of the found drawing item. If you select **Pan**, the active view is shifted without changing the view scale so that the center of the drawing item is in the center of the view. If **Scroll Mode** feature is turned off, then the active view is left unchanged.

Direction - Specifies the direction of the search. Because drawing objects are stored linearly in the database, using either **Next** or **Previous** moves forward or backward through the document.

Find - Initiates a search for a single item that is part of the specified search scope. Each time you click **Find**, the software searches for and highlights the next drawing item that matches the defined search criteria in the defined direction. Each time an item is found that matches the **Find what** criteria, the active view is changed based on the **Scroll mode** setting.

Find All - Initiates a search for all items matching the search criteria and adds them to the select set. The active view is updated based on the **Scroll mode** setting.

Replace Tab (Find and Replace Dialog Box)

Sets options for searching for and replacing drawing items. This tab is on the **Find and Replace** dialog box.

Find what - Lists search criteria, and allows you to enter new search criteria or select from a list of the five most recently used criteria. The items listed above the dashed line in the list are used to set new search criteria by searching using the **Catalog Item**, **Filter**, or **Inconsistency** options. Selecting the **Catalog Item** option opens the **Select Catalog Item** dialog box. Selecting the **Filter** option opens the **Select Filter** dialog box. Since you cannot replace an inconsistency, if you select the

Inconsistency option from the **Find what** list, the **Replace with** option is not available.

Replace with - Allows you to enter a new replacement item or select a replacement item from a list of the five most recently used items, which are listed below the dashed line. You can define a new replacement item by selecting **Browse** and picking a catalog item from the **Select Catalog Item** dialog box.

Search in - Defines the scope of the search. You can search the active drawing or just the contents of the active window. If the **Active Window** option is selected, the **Scroll mode** feature is not available. If the **Active Drawing** option is selected, the **Scroll mode** feature is automatically set to **Zoom**, and you can select a preferred scroll mode.

Scroll mode - Defines how the active view should be changed when a drawing item is found. If you select **Zoom**, the active view is zoomed to a multiple of the range of the found drawing item. If you select **Pan**, the active view is shifted without changing the view scale so that the center of the drawing item is in the center of the view. If **Scroll Mode** feature is turned off, then the active view is left unchanged.

Direction - Specifies the direction of the search. Because drawing objects are stored linearly in the database, using either **Next** or **Previous** moves forward or backward through the document.

Find - Initiates a search for a single item that is part of the specified search scope. Each time you click **Find**, the software searches for and highlights the next drawing item that matches the defined search criteria in the defined direction. Each time a item is found that matches the **Find what** criteria, the active view is changed based on the **Scroll mode** setting.

Replace - Replaces the item selected when you clicked **Find**. The **Replace** button is available only when the item found can be replaced by the item defined in the **Replace with** box. Each time you click **Replace**, the located item is replaced, and the **Find** command continues based on the **Direction** setting. As each drawing item is found, the active view is updated based on the **Scroll mode** setting.

Replace All - Replaces all items that match the **Find what** criteria with the item defined in the **Replace with** box without prompting you. After all of the replacements are completed, the software reports the number of matches found and the number of replacements that were made. The active view is also updated depending on the **Scroll mode** setting.

Selecting Connected Items: An Overview

You can use the **Select Connected Items** command to navigate the line connectivity from a selected line segment to add components and other runs to a select set based on matching properties. In cases where you want the software to continue

constructing a select set beyond a break in a line, for example, a nozzle, this command makes it possible to accomplish that goal.

This functionality is useful for finding branches from runs and finding all the runs and components on an entire drawing that have a common property value.

Once the items are selected in the drawing, it is then a simple matter to choose the select set in the **Properties** window and continuing editing properties for not only runs, but also lines and networks and all runs included in the drawing.

Notes:

- The properties that define a run are specified in Options Manager.

Select Connected Items Dialog Box

Allows you to construct a select set from lines and inline components, even if there are breaks in a network. This dialog box remains open until you dismiss it, and you can highlight items in the drawing and manipulate members of the select set while the dialog is active. The **Select Connected Items** command, which appears both on the **Edit** menu and on the **Line Shortcut** menu, is available only when either a single line segment or a single component is selected.

Scope - Provides options for defining the extent of your select set. The **Scope List** allows you to choose extent, and the **Include Runs** and **Include Components** buttons allow you to incorporate different items into the select set. At no time can neither the **Include Runs** nor **Include Components** buttons be selected; that is, one or the other or both is always active.

Scope List - Allows you to choose from **Run**, **Line**, **Network**, **Drawing**. The default value for this list is **Line**.

Notes:

- The **Run** option means that lines in a single run are selected. The **Line** option selects runs that are connected at end points, including components. The **Network** option adds branches, and the **Drawing** option selects all the runs and inline components in the open drawing.

Include Runs - Incorporates lines into the select set.

Include Components - Adds inline components into the select set.

Matching Properties - Displays options for defining the select set according to common properties. The properties for the originally selected item are displayed by default.

Alphabetic - Lists properties in alphabetical order.

Categorized - Displays properties grouped by specific categories. By default the properties are displayed categorically. Categories are defined and properties are assigned to those categories in Data Dictionary Manager.

Preview - Highlights in the Drawing view the items that are selected according to common properties in the **Properties List** of this dialog box.

Properties List - Allows you to choose common properties for your select set. You can select more than one property in this list by using the **Ctrl** or **Shift** keys.

Create a Select Set From a Line

6. Select a single line segment or inline component in the drawing.

7. Click **Edit > Select Connected Items**.

8. On the **Select Connected Items** dialog box, you can change the scope of the select set by changing the display in the **Scope** list.

9. You can change the scope to include inline components or exclude lines, too, by using the **Include Runs** and **Include Components** buttons.

10. At any time, you can select a different line segment or component in the drawing in order to change the properties displayed in the dialog box.

   📝 **Notes:**

   - The properties for the new selection are displayed in the **Select Connected Items** dialog box, but the display of the **Properties** window itself is not updated until you click **OK**.

11. If you want to construct a select set based on common properties, select those properties from the **Properties** list in the **Matching properties** area of the dialog box.

   📝 **Notes:**

   - You can use the **Ctrl** or **Shift** buttons to select more than one property value.

   - You can choose an empty property value for matching criterion, too.

   - If no properties are displayed in the dialog box, select a line segment or component in the drawing that already has a property value you know you are interested in.

12. Click the **Preview** button to see the items that qualify according to the matching criteria chosen in the previous step. The matching items are highlighted in the Drawing view.

13. Once you have selected the items you want for your select set, click **OK**. The selected items are centered in the **Drawing** view.

14. Use the **Selected Items** list in the **Properties** window to display the common properties for the select set.

   📝 **Notes:**

   - If you click **Cancel**, the **Select Connected Items** dialog box closes, and the original item is selected in the drawing.

Selecting Drawing Items: An Overview

To select an item, you can use the Select Tool on the main toolbar. You can select pipe routes, equipment, valves, labels, and so forth, and you can select item groups, assemblies, or embedded or linked objects.

Selecting Single Items

To select an item, click the **Select Tool** on the main toolbar. When you click the **Select Tool**, the pointer changes to an arrow with a locate zone indicator at the end. As you pause on items in a drawing, the items appear in the highlight color. When an item is highlighted, click to select it.

When you select an individual item, the following things happen:

- The item changes to the selection color. You can change the selection color with the **Options** command on the **Tools** menu.

- The handles of the item appear if the item has handles. Handles are solid squares at significant positions on a selected item, such as end points and center points. Handles allow you to directly modify the item, such as dragging a handle to change the shape of the item. Although you can select more than one item at a time, only one item can have handles at a time.

- If the item is linked or embedded into the current drawing, selecting it allows you to double-click it for editing.

When you select multiple items or grouped items, the items change to the selection color.

You can also select drawing items by first selecting them in the **Engineering Data Editor**. The corresponding items are selected in the Drawing view if they reside in the active drawing.

Selecting Multiple Items

You can select more than one item at a time by clicking **Select Tool** and then holding **Shift** or **Ctrl** as you click the items that you want to select. Additionally, you can click the **Select Tool** and then drag to fence items.

You can use the **Select Tool** ribbon to choose if you want to select only items completely enclosed by the fence or any item that is partly enclosed by the

fence. Many manipulation commands, like **Delete**, **Move**, **Copy**, and **Rotate**, can act upon all items in the select set.

If you select multiple items in the **Engineering Data Editor**, these items are also selected in the active drawing, if they reside there. You construct a select set using the **Engineering Data Editor** in this way.

Whenever you have constructed a select set, you can choose single items from that set by using the **Selected Items** list at the top of the **Properties** window. A single item chosen in a select set this way is highlighted in a different color in the Drawing view.

Select Tool Ribbon

The **Select Tool Ribbon** appears when nothing is selected. After you select an item, the **Select Tool** ribbon is replaced with a ribbon for editing the selected item.

Inside - Specifies that items inside the fence are selected.

Overlapping - Specifies that items both overlapped by the fence and inside the fence are selected.

Top Down (not applicable to SPPID)- Specifies that symbols are recognized in the locate zone as opposed to individual elements in a symbol.

Bottom Up (not applicable to SPPID) - Specifies that individual elements, which make up symbols, are recognized in the locate zone as opposed to the whole symbol.

Canceling the Selection of Items

To cancel the selection of an item or group of items, click any empty point on the drawing.

Navigating in the Engineering Data Editor: An Overview

The **Engineering Data Editor** (**EDE**) uses a grid or tabular format to display the query results for the database. The **EDE** is automatically displayed in the lower left of the main window, and you can also open the **EDE** to appear in the **Design** window.

> ✎ **Notes:**

> - The maximum number of items that can be displayed in the **EDE** is 65,537. If your database query results in more than this number, the **Query Results** dialog box opens. You can then choose to refine your query in order to generate fewer items or to enter report-only mode and run a report on your query results.

> - The **EDE** is where you view items in the **Stockpile**, the **Active Drawing**, the **Active Drawing Stockpile**, or **Other Drawings**. For more information, click **Related Topics**.

Query Results Dialog Box

Displays options for continuing your work in the **Engineering Data Editor** when you have requested more information than can appear in the interface. You can either refine your query or run a report with your query results.

Add - Places a new entry at the end of the existing definition list and enables the options in the **Edit** group so that you can edit the new entry.

Additional Filter Criteria - Displays all new criteria to add to the current filter. To add to or modify the definition list, you must select a line in the list and then define or edit the property in the **Edit** group.

Continue with this query so that I can generate a plant or personal report - Causes the **Engineering Data Editor** to enable only these commands: **Edit view**, **Save view**, **Delete view**, **My Reports**, and **Plant Reports**. This option is useful when creating a report for large data sets such as an entire plant database.

Delete - Removes the selected criterion from the definition list. This button is available only when you select a criterion in the definition list.

Edit - Allows you to define or edit a single line of filter definition criteria.

Match all - Specifies that items matching ALL of the filtering criteria pass through the filter.

Match any - Specifies that items matching any one or more of the filtering criteria pass through the filter. **Match any** is the default matching method.

Maximum Allowed - Displays the currently allowed number of items in the **Engineering Data Editor**.

Operator - Specifies the relationship between the property and its value. Relationships include, for example, greater than, >; equal to, =; not equal to, <>; and so forth.

Options - Allows you to choose between refining your query or entering report-only mode.

Property - Displays a list of all properties for a certain item type. Examples of properties include Equipment Type, Instrument Loop Item Tag, and Estimated Length. You define or modify filtering criteria by selecting a property, an operator, and a value.

Refine Query Definition - Allows you to use the **Additional Filter Criteria** list in this dialog box to refine your query in order to display an allowed number of items in the view.

Rows Returned - Displays the number of items that your query returned.

Value - Lists appropriate values for the property specified in the **Property** column. If a list of attributes is not already associated with the **Value** box, you must type a value, which can be free text, or choose null. You can type a percent sign, **%**, as a wildcard character to find multiple characters, or type a question mark, **?**, as a wildcard character for a single character. Do not use an asterisk, *****, in the **Value** box.

The EDE toolbar

At the top of the **EDE** is the **EDE** toolbar. The menu buttons are found here:

- Stockpile

- Edit

- View

Click the down-arrow next to the menu button to expose the available commands.

The list on the toolbar contains filters that query the database and display the corresponding items in the **EDE**. If you want to list all items select **Plant Item** from the list.

You can also filter the display by using the buttons that correspond to different stockpiles and drawings:

- Active Drawing

- Active Drawing Stockpile

- Stockpile

- Other Drawings

By default, initially only the **Active Drawing** and **Active Drawing Stockpile** buttons are selected.

The **Asking Filter** button displays the **Filter** tab of the **Advanced Table Properties** dialog box, from which you can quickly filter the **EDE** display by selecting a base filter and properties appropriate for the selected item type.

You can re-query and thereby refresh the **EDE** display by clicking the **Refresh** button.
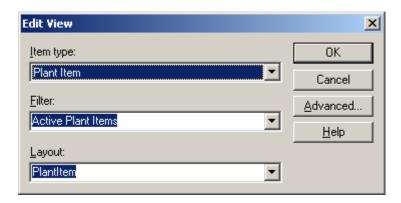
Finally a summary box Items found: 31 of 31 displays a comparison between the number of items in the view and the total number of items, of the type chosen in the list, that exist in the database.

Customize the EDE

Commands on the **Edit** menu allow you to customize the display of the **EDE** further. You can use the **Display Settings** command, for instance, to color code items according to the stockpile that they are currently in. Use **Edit View** to display the **Table Properties** dialog box, which allows you to define the items that you want to see in the **EDE**. You customize a filter and layout for the display on the **Advanced Table Properties** dialog box.

Table Properties Dialog Box

The Table Properties Dialog Box enables you to select the item types, filters, and layouts for the **Tabular** or **Engineering Data Editor**. This dialog box appears when you right-click an existing table and select **Edit View**.

Item type - Lists all the item types that you can select (for example, equipment or motors).

Filter - Lists all of the filters that have been saved for the selected item type.

Layout - Lists all of the layouts that have been saved for the selected item type. For a list of default layouts, see Default Table Layouts.

Advanced - Displays the **Advanced Table Properties** dialog box, where you can define and save filters or layouts for the **Tabular** or **Engineering Data Editor**. You can specify the default filter and layout for a certain item type also. Specify brief and bulk properties in this dialog box, too. Brief and bulk properties appear in the **Properties** window when you select the **Show Brief Properties**, **Copy Bulk Properties**, or **Paste Bulk Properties** commands from the **Properties** window toolbar.

Default Table Layouts

The software includes several default layouts. You can use them to base your own layouts on by changing the layout name, adding or deleting properties, and saving under a new name. Different layouts are included with SmartPlant Electrical and SmartPlant P&ID.

Area Break	Nozzle
Drawing	OPC
Equipment	Package
Equipment Component	Pipe Run
Exchanger	Piping Component
Mechanical	Plant Item
Equipment Other	Plant Item Group Other

Vessel	Representation
Instrument	Safety Class
Instrument Loop	Signal Run
Item Note	System

Advanced Table Properties Dialog Box

The **Advanced Table Properties** dialog box allows you to define, select, and save filters and layouts for the **Tabular** or **Engineering Data Editor**. This dialog box opens when you click **Advanced** on the **Table Properties** dialog box.

Filter Tab (Advanced Table Properties Dialog Box)

The **Filter Tab** allows you to define, select, and save a filter for the **Tabular** or **Engineering Data Editor**.

Base filter - Displays the name of a saved filter.

Name - Displays the name of the filter, as it is defined in Filter Manager.

Default - Indicates if the named filter is the default filter for the item type selected on the **Table Properties** dialog box. You can change a filter to the default by selecting this box.

Browse - Opens the **Select Filter** dialog box, which allows you to select a base filter.

Definition - Includes areas to add, remove, or edit filter criteria.

Match all - Specifies that items matching ALL of the filtering criteria pass through the filter.

Match any - Specifies that items matching any one or more of the filtering criteria pass through the filter. **Match any** is the default matching method.

Add - Places a new entry at the end of the existing definition list and enables the options in the **Edit** group so that you can edit the new entry.

Delete - Removes the selected criterions from the definition list. This button is available only when you select a criterion in the definition list.

Edit - Displays options that allow you to define or edit a single line of filter definition criteria.

Property - Displays a list of all properties for a certain item type. Examples of properties include revision number and name. You define or modify filtering criteria by selecting a property, an operator, and a value.

Operator - Specifies the relationship between the property and its value. Relationships include, for example, greater than, >; equal to, =; not equal to, <>; and so forth.

Value - Lists appropriate values for the property specified in the **Property** column. If a list of attributes is not already associated with the **Value** box, you must type a value, which can be free text, or choose null. You can type a percent sign, **%**, as a wildcard character to find multiple characters, or type a question mark, **?**, as a wildcard character for a single character. Do not use an asterisk, *****, in the **Value** box.

Layout Tab (Advanced Table Properties Dialog Box)

The **Layout Tab** defines, selects, and saves a layout for the **Tabular** or **Engineering Data Editor**.

Name - Displays the name of a saved layout. You can enter a new layout name and then click **Save** to save the layout for re-use. Or, you can select any of the layouts in the list.

Default - Indicates whether the named layout is the default layout for the item type on the **Table Properties** dialog box. You can designate a layout as the default by checking this box and then clicking **Save**.

Usage - Lists the property display types associated with the **Properties** window. Options include a blank value, **Brief**, **Bulk**, and **Brief/Bulk**. The selected item type and display properties define what appears in the **Properties** window when you select the **Show Brief Properties**, **Copy Bulk Properties**, or **Paste Bulk Properties** commands from the **Properties** window toolbar. In order for your choice in the **Usage** box to take effect, you must exit and reenter the design software.

Delete (Name) - Removes the named layout.

Save - Saves the data in this dialog box as a named layout. Layout names must be unique.

📝 Notes:

- Saving a layout is not the same as saving a view in the **Engineering Data Editor**. For more information about saving views, see **Save an Engineering Data Editor View**

Definition - Displays all defined criteria associated with the selected layout. This group contains two list views. In order to add to or modify either list, you must select a line in the list and then define or edit the property in the **Edit** group.

Display Property - Lists the columns that appear in the **Engineering Data Editor**. To edit an entry, select it and modify the property in the **Edit** group.

Caption (Definition) - Displays the caption for each column in the **Engineering Data Editor**.

Sort Property - Lists the properties that apply to the item type specified in the **Table Properties** dialog box.

Order - Lists the sort orders **Ascending** and **Descending**. You can select one.

Type - Lists the sort types **Numeric** and **Alphanumeric**. You select one.

Add - Adds a new entry at the end of the existing entries and enables the options in the **Edit** group so that you can edit the new entry.

Insert - Places a new entry above the currently selected entry and enables the options in the **Edit** group so that you can edit the new entry.

Delete (Definition) - Removes the currently selected entry.

Up - Moves the currently selected entry up one line. Moving a row up either moves a display column to the left or moves a sort order up, depending on the row you have selected.

Down - Moves the currently selected entry down one line. Moving a row down either moves a display column to the right or moves a sort order down, depending on the row you have selected.

Edit - Allows you to edit a single entry selected in either the **Display Property** or **Sort Property** list views. If you select an entry in the **Display Property** view, the **Edit** group allows you to edit the property and the caption. If you select an entry in the **Sort Property** view, the **Edit** group allows you to edit the property, order and type.

Property - Displays a list of all the properties that apply to the item type defined in the **Table Properties** dialog box. If you select an entry in the **Display Property** view, you can specify the property and its caption. If you select an entry in the **Sort Property** view, you can specify the property, its order, and its type.

Caption (Edit) - Displays the caption for each column in the **Engineering Data Editor**.

Save View Command
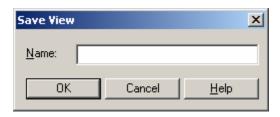
The **Engineering Data Editor > View > Save View** opens the **Save View** dialog box, which allows you to name and save the current **Engineering Data Editor**. These

custom views appear at the bottom of the filter list on the **Engineering Data Editor** toolbar.

Save View Dialog Box

The **Save View** dialog box allows you to specify a name for your custom **Engineering Data Editor**. This name, which applies to the currently active **EDE**, is displayed at the bottom of the filter list on the **Engineering Data Editor** toolbar with other saved views. Open this dialog box by clicking **View > Save View** in the **EDE**.

Name - Provides space for you to type in the name of your custom view.

AutoFilter Command

The **Engineering Data Editor > View > AutoFilter** allows you to filter the items displayed in the **Engineering Data Editor**. When you click **AutoFilter**, arrows appear at the top of each column. Click the arrow at the top of the column that contains the data that you want to filter on and choose the value from the list that appears. For instance, if the **Engineering Data Editor** displays **Equipment: Vessels**, but you want to see only vertical drums, use the **AutoFilter** command (click the arrow at the top of the **Eq Subclass** column and choose vertical drums).

To create more elaborate filters and displays, you need to use the **Edit View** command and enter options on the **Table Properties** dialog box and the **Advanced Table Properties** dialog box.

Select this command again to deactivate it.

Freeze Panes Command

The **Engineering Data Editor > View > Freeze Panes** allows you to scroll through the **Engineering Data Editor** while keeping the display of specified rows and columns constant. For instance, the first column can contain the item tag, but you want to always see the item tag as you scroll to the right end of the rows to view the supplier. You can accomplish this layout by using the **Freeze Panes** command.

Display Settings Command

The **Engineering Data Editor > View > Display Settings** opens the **Display Settings** dialog box, which allows you to specify various display options for the **Engineering Data Editor**. For example, you can choose the size of icon that is displayed in the view, and you can specify the font that items in the Stockpile are listed in. You can choose that read-only cells are filled with a special background color.

Editing Properties in the EDE

You can edit some of the properties viewed in the table by selecting a table cell and filling in the appropriate information. If the property has a select list associated with it, then you choose the value from the select list. If the cell is populated by a read-only property or a property that is only automatically generated by validation, you cannot edit that cell. You can select a entire row by clicking the corresponding icon in the left-most column of the table. Then you can view the properties in the **Properties** window and edit item properties there, too.

The contents of cells can be copied to other cells, if the cell contents are compatible.

EDE Working with the Stockpile

SmartPlant P&ID uses a stockpile concept to store data that does not currently reside in the drawing, such as certain items deleted from the drawing, off-page connector mates, loops, and imported data. The stockpile may be accessed using the **Engineering Data Editor**. The **Stockpile** contains items that are visible to all users who are working in the plant. The **Drawing Stockpile** contains items accessible only to that drawing. You may click on the **Stockpile** and/or **Drawing Stockpile** toolbar buttons to view items either in one or both of the two views of the Stockpile.

You may move items from one stockpile view to another by using the **Actions** toolbar button in the **Engineering Data Editor**.

Deleting Symbols

When certain symbols are deleted from the drawing view, they will reside in the **Stockpile**. This means they have been deleted from the drawing, but are still available to be placed elsewhere in that drawing or in another drawing. There are two factors that govern whether or not an item will go to the **Stockpile** once it's deleted from a drawing:

1. The **Tag Reqd Flag** property is set to **True** (properties are defined using Catalog Manager).

2. The Item Type must be specified in the **Options Manager**. Select the Reference Data **Settings** and the **Stockpile Items** field.

You may delete items residing in the **Stockpile** by right clicking on the item and selecting **Delete Stockpile Item**.

Notes:

- When deleting an item from the drawing, a right-click displays the option to **Delete** or **Delete from model**. Selecting Delete from model bypasses the stockpile, and deletes the item from the database and the drawing.

Off Page Connector Mates

When an off page connector (OPC) is placed in the drawing, its mate may automatically go to the stockpile or may be placed in the drawing stockpile of the mating drawing. When the mating drawing is defined, the matching off page connector can be placed on the pipeline. This will associate the lines in the two drawings.

Delete a Connector From the Model

1. Right-click the connector that you want to remove from the model.

2. From the shortcut menu, click **Delete From Model**.

   Notes:

   - This command is available only when the partner of the selected connector is stored in the Stockpile.

   - If you want to move the connector to the Stockpile instead of deleting it completely from the database, click **Delete** on the shortcut menu.

Instrument Loops

A **loop** is a logical grouping that defines a collection of **instrumentation**. To create a loop in SmartPlant P&ID, right-click the loop in the **Catalog Explorer** and select **Send to Stockpile** or **Send to Drawing Stockpile**, and assign to it the desired properties. When an instrument is placed, a **Loop Tag** property is available with a drop-down list of the loops created in the stockpile.

Plant Item Groups

Package, **Contract Package**, **Hydro Test Package**, **Test Systems**, and **Safety Class** are similar to **Instrument Loops** in that they are a group of one or more items with a common set of properties. For example, you may want to have a package to denote items that will be included in a palette. A system could be used to denote the portions of a P&ID used for hydro testing prior to startup.

To create a **Package**, **Contract Package**, **Hydro Test Package**, **Test Systems**, or **Safety Class** in SmartPlant P&ID, right-click on the item in the **Catalog Explorer** and select **Send to Stockpile** or **Send to Drawing Stockpile**, and assign to it the desired properties.

To associate items with a **Package**, **Contract Package**, **Hydro Test Package**, **Test Systems**, or **Safety Class**, you will use the **Pkg Item Tag property**, **Contract Package No**, **Hydro Test Package No**, **Test Sys Item Tag property**, or **Instr Safety Class Item Tag** respectively.

Moving and Placing Stockpile Items: An Overview

You can move the items that reside in stockpiles (the Stockpile, which belongs to the Plant or the active project, or drawing stockpiles) from one stockpile to another, or you can place them in a drawing or delete them from the model altogether.

You view the items in a stockpile in the **Engineering Data Editor** (**EDE**). In particular click **Stockpile** to view items in the Stockpile, or click **Active Drawing Stockpile** to view items in the stockpile of the active drawing. Be sure that the other item buttons on the **Engineering Data Editor** toolbar (that is, the **Active Drawing** or **Other Drawings** buttons) are not active when you want to view stockpile items alone.

In the **EDE**, the **Action** menu contains the commands for manipulating stockpile items. These commands make it possible to place or move or delete stockpile items. You can also access some of these commands from the **Engineering Data Editor** shortcut menu, which opens when you right-click in the **EDE**.

You have the capability to place an item from a stockpile directly onto an open drawing. You enter placement mode when you click the icon associated with a stockpile item in the left-most column of the **EDE**.

You can import items into the Stockpile, too, by importing reports with the File > Import > Data File command on the main toolbar.

Moving Between Stockpiles

The **Move to Different Stockpile** command in the Engineering Data Editor allows you to move an item from one stockpile to a different stockpile. The selected model item must be claimed before you can use this command.

 Notes:

- Plant item groups, such as instrument loops, packages, safety classes, and so forth, frequently reside in a stockpile with their members residing on drawings. Moving the plant item group itself to another stockpile has special limitations:

  o **From a drawing stockpile to the Stockpile** - No constraints apply. This action can occur at anytime.

  o **From the Stockpile to a drawing stockpile** - If the plant item group contains only items in the destination drawing or does not contain items in any drawing, this action is allowed.

  o **From a drawing stockpile to another drawing stockpile** - Same constraint as above applies.

Plant item groups are listed in Data Dictionary Manager.

Move to Different Stockpile Dialog Box

Opens when you right-click a stockpile item icon in the **Engineering Data Editor** and select **Move to Different Stockpile**. You can move your selected stockpile item into the drawing stockpile of another drawing.

Tree view - Allows you to navigate in the plant hierarchy in order to display the drawing into which stockpile you want to move the selected item.

List view - Displays a list of available drawings that can accept the selected stockpile item.

Linking and Embedding Objects: An Overview

This software is compatible with Object Linking and Embedding (OLE) software. You can transfer text, numbers, or images between drawings or documents that were created with this software and other OLE applications. You can move whole documents around or just parts of a document that you select. The information that you move is an object.

For example, you can use Microsoft Word to make a comment and then display the comment in your drawing, or you can insert a Microsoft Excel worksheet into a drawing. You can even create a drawing that contains another drawing, notes from Microsoft Word, and an Excel worksheet.

You can insert information created in another application with one of many methods:

- You can cut and paste an item to delete it from one location and move it to another location.

- You can copy and paste an item to duplicate it in another location.

- Linking stores data in one location and places a copy with a link in another location. When you change the original data, the copy can be updated either automatically or manually.

Embedding copies information and stores the information in another document that was created in a different application. If you change the embedded object, the original information does not change and vice versa.

Notes:

- Linked or embedded files are not transferred by workshare. You must transfer those files manually, and they should be located in the same directory as the drawing files.
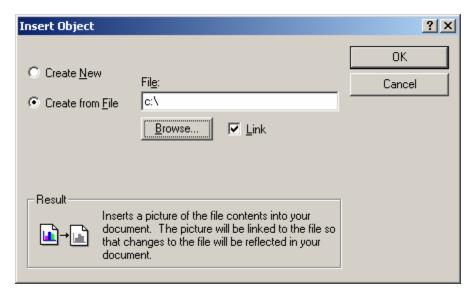
Linking or Embedding

An object is data from one application that you insert in a drawing. You can link or embed information between drawings or documents that were created with this software and OLE software.

Linking stores data in one location and places a copy and a link in another location. The linked data is stored in the source document itself. When you paste the information in the destination document with the **Paste Link** option, a reference point or box is inserted into the destination document. This box displays the information that you pasted. When you change the original data, the pasted data is updated either automatically or manually. You can edit the linked information by opening the source

document. You can insert linked information with the **Paste Special** command on the **Edit** menu. You can link an entire document to another document by inserting the information with the **Insert > Object** command on the **Edit** menu.

Embedding copies data in one document and stores the information in another document. If you change the copied data, the original information does not change, and vice versa. You can embed information with the **Paste Special** command on the **Edit** menu. You can embed an entire document inside another document by inserting the information with the **Insert > Object** command on the **Edit** menu.

If you want to keep the document size small, then link the data to another document. Linked information increases the document size only by the size of the displayed image, not by the size of the data itself. Linking is also useful when you want to share information in many different places and set that information to update automatically. For example, if you want to display a drawing in several different reports that you created in Microsoft Word, you could link the drawing inside the documents that contained the reports. Then, later, you could update the drawing by editing it in this software. Each report would automatically display the results of the drawing updates.

If document size is not an important factor, then use embedding. Embedded objects make the file size larger.

Embedding is useful when the person viewing the document with the embedded information does not have access to the software that created the object. The person can still view the information if it is embedded in the document. For example, if you wanted to send out several drawings for review, you could embed each drawing into a Microsoft Word document and send the Microsoft Word document to each reviewer.

Embedded information is also a good choice if the information does not need updating.

Embedding by Cutting or Copying, and Pasting

The easiest way to embed an object is to use the **Cut**, **Copy**, and **Paste** commands on the **Edit** menu in any OLE software. If you copy or paste information into an application in which you cannot edit the information, the software will automatically embeds the information in the document. You can then edit the embedded information with the software that created it. If you cannot edit the information, the information appears as a static picture.

Notes:

- If you want to copy the information, and not remove it from its original location, press **Ctrl** while dragging.

- On the **Paste Special** dialog box be sure that you select **Paste** and not **Paste Link**.

- Or if you want to link the object, select **Paste Link** on the dialog box.

Linking or Embedding an Object

If you want to insert an existing document, you can use **Edit > Insert > Object**, too. On the dialog box, select the **Create From File** option and then enter the name, or browse to the document. The entire document is embedded into the drawing.

Notes:

- If you want to embed the existing object, make sure you have not selected **Link** on the dialog box.

- If you want to link the existing object, select **Link** on the dialog box.

Editing a Linked or Embedded Object

To edit an object, you can double-click the object to open the software that created the object. If you do not have the source software installed, you cannot edit the object. The menus and toolbars of the current software are temporarily replaced by the menus and toolbars of the software that just opened.

Notes:

- You can click commands on a shortcut menu to activate the software that created the object. To get the shortcut menu, right-click the embedded object.

Calling Item Tag Validation

The **ItemTag** project performs calculations and validations for unique tag checking, automatic tag generation, and tag reformatting. This project generates unique **Item Tag** values, maintains consistency between the **Item Tag** value and the properties used in its calculation, and also removes any trailing or embedded spaces.

The **ItemTag** project deals specifically with the following item types: instrument loops, instruments, pipe runs, equipment components, signal runs with a plant item type pipe run (hydraulic, connect to process, and so forth), and equipment (other equipment, exchangers, mechanical equipment, and vessels). This project disregards all other item types.

Property validation is triggered when any property that comprises the **Item Tag** value is added or modified. Calculation can be triggered by the **Item Tag** property for any of the items shown in the tables below.

The following tables list item types that are validated or calculated by the **ItemTag** project, and the database tables and column names where modifications trigger calculation and validation.

<u>Legend:</u>

Italics: Required for checking item tag uniqueness.

Bold Italics: Required for the item tag, but can be generated automatically.

Instrument

Item Tag Structure	*Instrument Type Modifier Measured Variable Code Tag Sequence Number* Loop Tag Suffix Tag Suffix
Format	(Instrument Type Modifier)(Measured Variable Code)-(Tag Sequence Number) (Loop Tag Suffix)(Tag Suffix)

Instrument Loop

Item Tag Structure	*Instrument Loop Function **Tag Sequence Number** Tag Suffix*
Format	(Instrument Loop Function)-(Tag Sequence Number)(Tag Suffix)

Pipe Run

Item Tag Structure	Unit Code ***Tag Sequence Number*** Tag Suffix *Fluid Code*
Format	(Unit Code)(Tag Sequence Number)(Tag Suffix)-(Operating Fluid Code)

Equipment

Item Tag Structure	*Tag Prefix **Tag Sequence Number** Tag Suffix*
Format	(Tag Prefix)-(Tag Sequence Number)(Tag Suffix)

Equipment Component

Item Tag Structure	*Tag Prefix **Tag Sequence Number** Tag Suffix*
Format	(Tag Prefix)-(Tag Sequence Number)(Tag Suffix)

Signal Runs (Plant Item Type Pipe Run)	
Item Tag Structure	Unit Code *Operating Fluid Code* ***Tag Sequence Number***Tag Suffix
Format	Unit Code(Operating Fluid Code)-(Tag Sequence Number)(Tag Suffix)

With the automatic item tag generation feature, the next available number will be selected from the **Options Manage**r, and the values in **Options Manager** will be incremented to reflect the change.

If you choose to input the **Tag Seq No** information in the field, SmartPlant P&ID checks for duplicate values when the item tag is generated. If an item tag already exists with that value, you are prompted either to create a unique value or to allow the duplication.

Equipment

When you define the **Tag Prefix** property for equipment in a P&ID, the calculation program automatically creates the next **Tag Seq No** and formats the values together to generate the **Item Tag** property for the item.

Nozzles

Nozzle numbers are generated per equipment item and tag prefix. When you define the **Tag Prefix** property for a nozzle in a P&ID, the calculation program automatically starts with a **Tag Seq No** of 1 and formats the values together with **Tag Suffix** to generate the **Item Tag** property for the item. The **Item Tag** is unique with respect to the **Tag Prefix** and **Tag Seq No** properties. Each unique prefix will start the numbering at 1. For example, if you If you assign nozzles with a **Tag Prefix** of N and then assign a manway (which is also a nozzle) a **Tag Prefix** of M, those that have a prefix of M will start numbering on that piece of equipment with a 1.

Piping

When you define the **Fluid Code** property for a line, the calculation program generates the **Tag Sequence Number** and formats the value together with the **Fluid ode, Piping Materials Class, Nominal Diamete**r, and **Insulation Purpose** to generate the **Item Tag** property item.

Off-Page Connectors

When an off-page connector is placed in a drawing, the **OPC Tag** is automatically generated based on the next available sequence number from the **Options Manage**r. The off-page connector's match is automatically placed in the stockpile with the same **OPC Tag**.

Instrument Loops

When you define the Tag Suffix property for the loop, the calculation program will automatically generate the **Tag Sequence Number** and format the value together with the **Measured Variable** and **Type Modifier** to generate the **Item Tag** property.

The following tables specify items that are validated or calculated by the ItemTag project, including database tables and column names where modifications trigger calculation and validation routines. The following legend provides further definition of included **Item Tag** requirements:

      i. **Piping OPC's** > **Off -Drawing**

      ii. Send the mating **OPC** to the **Stockpile**

      iii. Notice the number generated at placement of the Off-Page Connector.

- Send an Instrument Loop to the Stockpile.

      i. **Symbols** > **Instrumentation** > **Loops**

      ii. Toggle the **Engineering Data Editor** to **Instrument Loop**.

2. Define the following property values in the Property Window for the below items by selecting the item in the drawing.

- Horizontal Drum - Tag Prefix

- Nozzle - TagPrefix

- Piping - Fluid Code

- Instrument Loop – Tag Suffix. (Utilize the Engineering Data Editor for the defining the Tag Suffix for the Instrument Loop.

3. Place another Horizontal Drum.

Working With Assemblies: An Overview

An assembly is a group of items that you can place in a drawing at one time. Assemblies can save time because you do not have to keep creating commonly-occurring groups of items. Instead of copying and pasting each individual item into a drawing at several locations, you can place the group of items: the assembly. An example of an assembly is a control valve with vents on both sides, an instrument loop that controls the valve, and a piping bypass around the valve.

You can create an assembly by selecting several items in a drawing and saving as an assembly. The file extension for an assembly is .pid, which is the same file extension as a drawing. If you want to retrieve the assembly from the **Catalog Explorer** tree view, you must save the assembly where the other symbols are located. The **Save As Assembly** command prompts you to save the assembly in the correct directory, which is specified in Options Manager.

Placing assemblies is like placing any other item from **Catalog Explorer** into a drawing. When you want to place an assembly into a drawing, you select the assembly from **Catalog Explorer** and click a location in the drawing. Or, you can drag the assembly into the drawing. The information in each label remains intact when you place the assembly. After you have placed an assembly, the items in the assembly are treated as separate items. You can delete just one item without concern that the software deletes the entire assembly.

Assemblies cannot be placed inline. You must place the assembly first and then route piping to it, if required.

 Notes:

- Assemblies can be created in one plant and used in another. However, the file structure and symbols must be identical. If you try to place an assembly into a plant that uses a different file structure, you get an error message that the software cannot find your reference files. The assembly placement quits. Move the symbols that cannot be found to the location indicated in the error message.

- When creating or placing assemblies, a log file is created for each action. The log files are placed in the Temp folder and are named <AssemblyName>_CreAsm.log and <AssemblyName>_PlaAsm.log.

Create an Assembly

1. Select several components in the drawing.

    Notes:

- You can select more than one item at once by dragging to fence the objects.

- To remove an item from the selection, click it while pressing the **Ctrl** key.

2. Click **File > Save As Assembly** on the main menu bar.

3. Specify the origin of the assembly.

   - A red target ⊕ appears at the end of your pointer. Use this to specify the origin of the assembly which is utilized during placement of the assembly.

     📝 **Notes:**

     - You can only use a standalone symbol as the origin of an assembly. For example, you cannot use labels, pipe runs, signal runs, or child items; however, you can use unattached ends of pipe runs.

4. Define a **File Name** and **Folder** location for the **Assembly** on the **Save As Assembly** dialog box.

5. Select **Save**

6. You will be notified when the assembly has been created.

7. Reference the log file in your ~\temp folder when assembly creation is complete.

   - <AssemblyName>_CreAsm.log

📝 **Notes:**

- Any graphics that have been band-aided should be deleted and replaced prior to using this command.

- The file extension for assemblies is .pid.

- You can save an assembly in any directory that you want, but the default assembly path is specified in Options Manager. If you use the

default folder, then you can retrieve the assembly from **Catalog Explorer**. In order to save assemblies, you must have write permissions to the folder defined in Options Manager.

- Assemblies can be created in one plant and used in another. However, the file structure and symbols must be identical. If you try to place an assembly into a plant that uses a different file structure, you get an error message that the software cannot find your reference files. The assembly placement quits. Move the symbols that cannot be found to the location indicated in the error message.

Place an Assembly

1. In the **Catalog Explorer** list view, click the assembly that you want to place.

2. Use the red target that appears at the end of your pointer to position the assembly appropriately.

   📝 **Notes:**

   - The red target is the origin of the assembly. Some assembly members also appear during placement to aid you.

   - Press **Esc** to quit placement mode, or you can escape placement mode by right-clicking.

3. Connect the pipe runs and instruments on either side of the assembly.

   📝 **Notes:**

   - You cannot place an assembly into a line that is already routed.

   - Modify or delete individual components in the assembly as necessary.

4. You will be notified when the assembly has been placed.

5. Reference the log file in your ~\temp folder when assembly placement is complete.

   - <AssemblyName>_PlaAsm.log

Answer = No, Piping Components do not meet the criteria required for an item to be placed in the stockpile upon deleting. The criteria is **Tag Required** property must equal **True**, as defined in **Catalog Manager** for the Globe Valve and Piping Components are not listed as **StockpileItems** in **Option Manager** > **Settings**.

1. **Delete** the **PipeRun** that was routed as a **branch**.

2. Does the item reside in the **Stockpile**?

Answer = Yes, **PipeRuns** do meet the criteria required for an item to be placed in the stockpile upon deleting and there is NOT another PipeRun with the same ItemTag still in the drawing. The criteria is **Tag Required** property must equal **True**, as defined in **Catalog Manager** for the **PipeRun** and **PipeRuns** are listed as **StockpileItems** in **Option Manager** > **Settings**.

3. Select the **PipeRun** from the **Engineering Data Editor** and place in free space, connect the **PipeRun** to the **main run**.

4. Review and Resolve the **Inconsistency** between the **main run** and the **branch**.

5. Route a **branch** from the **main run** to the right side of the drawing.

Working with Off-Page Connectors in the Stockpile

1. Place a Piping Off-Page Connector on the end of the branch which was routed from the previous step.

Consistency Checking: An Overview

Consistency checking verifies the suitability of work that a designer performs while creating the drawing. The software verifies, in real-time, if the composition of a drawing and the underlying data model satisfies rules that your company has defined. The software includes pre-defined standard industry design propagation: for example, pipe runs inherit properties from nozzles. Additional consistency checking and design propagation are defined in Rule Manager.

Consistency checking continuously monitors your work when you change or add items to a drawing. The software displays all the inconsistencies, describes specific problems, and offers hints. Using these solutions, you can decide the best method to resolve an inconsistency.

Consistency Checking and Break Labels

A property break is a point in a connected network of objects where a property value changes. A user can define a property break by the placement of a break label. A property break is an exception to the consistency criteria defined in a rule. A property break defines a limitation to the spread of the propagation scope. Whereas rules provide the general definition of the scope, property breaks provide specific limitations of the scope.

The Consistency Check dialog shows the consistency criteria that apply at the selected relationship indicator. All of the properties that are copied and compared across this connection are shown. When the user has placed a break label, the copy and compare columns show icons that indicate that no copy and compare operations are performed for the properties in the label.

Break labels can be created in the Catalog Manager environment. A break label is created just like an ordinary label, except that the Label Type property must be set to "Attribute Break". A SmartText field must be created for each property that the break label is intended to break. A break label can break one or more properties. The Catalog Manager currently supports all this functionality. No new functionality is required.

Break labels can be placed directly from the catalog explorer. A group of break labels is currently delivered under Piping\Segment Breaks. Break label placement is enhanced so that a break label can only be placed at a connection point where the specified properties are being propagated. When a break label is placed interactively, the corresponding property break data is added to the Relationship object.

Break labels can be deleted in the standard way using the Delete command. When a break label is deleted the break is removed from the Relationship object. However, if

there were another break label at this same point that also breaks this property, then the break would not be removed.
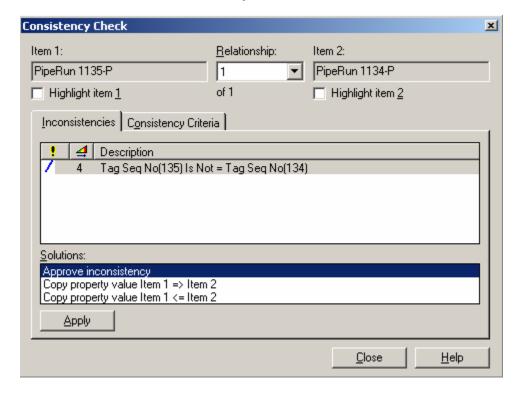
Consistency Check Command

Right-click an inconsistency indicator and choose **Consistency Check** to display the **Consistency Check** dialog box. As you create a drawing, consistency checking monitors your drawing for design discrepancies. If a location in the drawing has only one or multiple inconsistencies, you can correct the problems by reviewing them on the **Consistency Check** dialog box.

Consistency Check Dialog Box

The Consistency Check Dialog Box opens when you display the properties of an inconsistency indicator, and lists the inconsistencies, their specific problem area, and solutions for resolving the inconsistency. The dialog box also displays the consistency criteria and propagation breaks that apply at the selected relationship indicator.

You open this dialog box by right-clicking an inconsistency indicator and choosing **Consistency Check** from the shortcut menu. You can also select **Edit > Consistency Check**. When this dialog box first appears, the software automatically selects the first item in the list. Other sections of the dialog box provide details that are specifically related to the selected inconsistency.

All inconsistencies that occur at a junction, regardless of severity, are listed on the **Consistency Check** dialog box. When you select one of the inconsistencies in the list, the software displays information specific to that inconsistency in the **Solutions** box. A specific example of this type can involve data inconsistencies for nominal pipe diameter, normal operating temperature, and alternate design pressure between a piping component and pipe run. This represents three different problems but at the same junction.

The **Consistency Check** dialog box contains the Inconsistencies Tab and the Consistency Criteria Tab.

Inconsistencies Tab

All of the properties that are compared by consistency checking and copied across this connection by propagation are shown on these tabs. The information shown here is very similar to the information shown on the **Consistency** tab of the **Rule Properties** dialog in the **Rule Manager**. Open the **Consistency Check** dialog box by selecting **Edit > Consistency Check**.

Item 1 - Displays the inconsistency, item type, and item tag for the first item that is connected by the active relationship. When the **Consistency Check** dialog box first appears, all items involved in the inconsistency are selected.

Relationship - Displays the number for the active relationship and allows you to select the relationship to be reviewed. The drop-down list contains a number for each relationship that exists at the selected point. The graphic relationship indicator that corresponds to the number shown in this control is highlighted.

Item 2 - Displays the inconsistency, item type, and item tag for the second item that is connected by the active relationship. When the **Consistency Check** dialog box first appears, all items involved in the inconsistency are selected.

Highlight item 1 - Place a check mark in the check box to highlight the item in the drawing. This helps you to quickly locate the selected object. By default, this option is not selected.

Highlight item 2 - Place a check mark in the check box to highlight the item in the drawing. This helps you to quickly locate the selected object. By default, this option is not selected.

Inconsistencies - Displays the inconsistencies at the selected relationship. For each inconsistency, the status, severity and description are displayed. The icons used for the status column are the same as the icons used for inconsistencies in the Drawing View. The first inconsistency in the list is automatically selected. You can select any of the inconsistencies with a mouse click or the arrow keys.
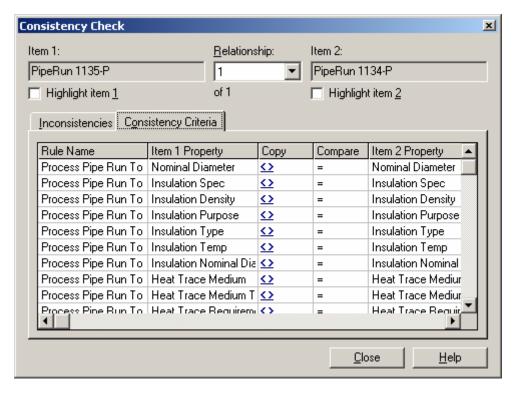
Description - Displays the actual error or warning. For a lengthy error or warning, a ToolTip appears to reveal the entire description. You cannot change the inconsistency description, which is provided for information only.

Solutions - Displays the list of available solutions for the selected inconsistency. The same solution that was previously selected is automatically selected again. If none was previously selected, the first solution in the list is automatically selected. Some of the solutions copy a property value across the relationship and initiate propagation. When one of these solutions is selected, the propagation scope is highlighted. This shows the items that will be changed before they are modified. After selecting an inconsistency from the **Inconsistencies** list you can select the correct solution from this list and after clicking **Apply**, the selected solution is performed.

Apply - Performs the solution you choose from the **Solutions** box. The inconsistencies are then re-evaluated and the dialog box is updated.

Consistency Criteria Tab

All of the properties that are compared by consistency checking and copied across this connection by propagation are shown on these tabs. The information shown here is very similar to the information shown on the **Consistency** tab of the **Rule Properties** dialog in the **Rule Manager**. Open the **Consistency Check** dialog box by selecting **Edit > Consistency Check**.

Consistency Criteria – This list view shows all of the properties to be compared and copied at this relationship. These consistency criteria come from the rules that apply to this relationship. The consistency criteria for each rule are originally specified within the Rule Manager. The name of the source rule is shown in the first column. The remaining columns are the same as shown in the **Consistency** tab of the **Rule Properties** dialog box in **Rule Manager**. Single selection mode is supported.

Item 1 - Displays the inconsistency, item type, and item tag for the first item that is connected by the active relationship. When the **Consistency Check** dialog box first appears, all items involved in the inconsistency are selected.

Relationship - If there are multiple relationship indicators at the same location, all of them are loaded into the **Consistency Check** dialog box. Each relationship indicator is assigned a number. The numbers are loaded into the drop-down list on the **Relationship** control. You can review each one individually by selecting from the entries in the drop-down list.

Item 2 - Displays the inconsistency, item type, and item tag for the second item that is connected by the active relationship. When the **Consistency Check** dialog box first appears, all items involved in the inconsistency are selected.

Highlight item 1 - Place a check mark in the check box to highlight the item in the drawing. This helps you to quickly locate the selected object. By default, this option is not selected.

Highlight item 2 - Place a check mark in the check box to highlight the item in the drawing. This helps you to quickly locate the selected object. By default, this option is not selected.

Rule Name - Displays the name of the rule.

Item 1 Property - Displays the name of an Item 1 property to be copied and/or compared is displayed in this column. Only those properties that are listed are copied across the relationship (propagated) and compared to the value in the related item (consistency checked).

Copy - The copy action to be performed is displayed in this column. This column controls how the property value is propagated when a property value is changed. The following table describes the possible values and their meanings.

Copy Action Symbol	Copy Action Name	At Property Modification
None	None	The property is not propagated across this relationship.
->	Copy 1 to 2 if Null	The value from Item 1 is copied to Item 2 during propagation but only if the current value on Item 2 is Null.
<-	Copy 2 to 1 if Null	The value from Item 2 is copied to Item 1 during propagation but only if the current value on Item 1 is Null.
<-->	Copy Bi-directional if Null	The value can be copied in either direction during propagation but only if the current value on the target item is Null.
->	Copy 1 to 2 Always	The value from Item 1 is copied to Item 2 during propagation.

<-	Copy 2 to 1 Always	The value from Item 2 is copied to Item 1 during propagation.
<-->	Copy Bi-directional Always	The value can be copied in either direction during propagation.

Compare - Displays how the property values from Items 1 and 2 are compared. The comparison occurs whenever any property on either item is changed. The following table describes the possible values and their meanings

Comparison Operator	Meaning
None	No comparison
=	Equal
<	Less than
<=	Less than or equal to
>	Greater than
>=	Greater than or equal to
<>	Not equal

Item 2 Property - Displays the name of an Item 2 property to be copied and/or compared. Typically, this property name is the same as the Item 1 Property. However, there are some cases where the property names are different. This property must have the same data type as the property listed for Item 1.
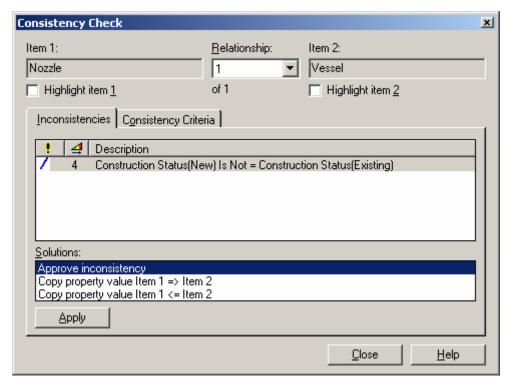
    a.   click, select **Properties**

    OR

    b.   Double click on the **Inconsistency**

    OR

    c.   Utilize the **Edit** > **Find** command

2. If Inconsistencies are not appearing on Drawing, right mouse click in the free space of your drawing and select **Show** -> **Inconsistencies** or select **View** > **Show Inconsistencies.**

3. Route a Horizontal Pipe in Free Space

    a. **Symbols** > **Piping** > **Routing** > **Process Lines** > **Primary Piping**

4. Define the Property of **Piping Materials Class** to be **1C0031**.

5. Route a branch in the vertical direction from the Horizontal Pipe.

    a. **Symbols > Piping** > **Routing** > **Process Lines > Primary Piping**

Using Insulation: An Overview

The complex process plants of today require the use of extreme temperatures, both hot and cold, to manufacture the products that you need. To make the plants energy efficient and economical and to provide safety to workers, the use of insulation is required. Over the years, company standards have been developed that dictate the type and thickness of insulation based on temperature and other operating factors. These standards make it feasible for the design software to select the proper insulation required and automatically enter it into the database. This practice insures proper design and speeds up the design process by freeing users from the laborious and error-prone task of entering property data.

Detailed insulation thickness calculations are necessary for each pipeline and piece of equipment in a plant. However, since insulation typically comes in a standard thickness, companies commonly generate tables of data that match the thickness required versus temperature and diameter. Different tables are generated for different conditions, for example, personnel protection or energy conservation. SmartPlant® P&ID Insulation Specification Manager provides users with a way to enter their table data such that it is electronically stored so that the software can access the data as required. In other words, the software does the table look-ups for you, so you do not spend valuable time looking through stacks of insulation tables.

Insulation Specification Access

In SmartPlant P&ID, you can use **Insulation Specification Access** to select the Insulation Spec for Piping and/or Equipment. For Piping, the software uses the Insulation Temperature, Insulation Specification, and Pipe Nominal Diameter combined with the selected specification to retrieve an Insulation Thickness, Type and Density from the lookup table. For Equipment, the software uses the Insulation Temperature and Insulation Purpose combined with the Specification to retrieve an Insulation Thickness, Type, and Density from the lookup table.

If the user specifies an Insulation Nominal Diameter, then the calculation will ignore the Pipe Run Nominal Diameter and use Insulation Nominal Diameter property for the lookup. (Where this might be used is with Jacketed Pipe. Although SmartPlant P&ID does not directly support Jacketed Pipe, with the Insulation Nominal Diameter, the user can now set diameter for the outer jacket using the IND and leave the pipe run nominal diameter set to the core pipe diameter. With the correct insulation spec for the jacket the system will determine the correct thickness.)

If you have already specified other insulation properties, such as Insulation Type or Insulation Purpose, then when you click the calculation button for the Insulation Spec property, it will display a list that contains only those insulation specifications that agree with the properties already chosen; nonetheless, you can choose a specification

from the **All** list. Selecting a spec from the **All** list will change the Insulation Purpose to match that defined by the Spec.

Normally, when using the insulation lookup function, the software will determine an insulation thickness. A property tracks whether the software or an individual user set an insulation thickness with an attribute displayed in the property grid called **Insulation Thk Source**. If the user decides to override the thickness provided by the software, SmartPlant P&ID will change the source property to User.

The insulation specifications are stored in a file with an .isl extension. The delivered file is called InsulationSpec.isl. A pointer to this file is located in the Options Manager > Settings.

Assign an Insulation Specification

1. Select the item, **Equipment** or **Pipe**; you want to add an **Insulation Specification** to.

2. Select **Insulation Spec** in the **Properties Window**.

3. Select the **Calculation** button: **Error! Objects cannot be created from editing field codes.**.

4. On the **Insulation Spec** dialog box, choose a specification from the either the **Matching** or the **All** lists.

   Notes:

   - If you have already specified other insulation properties, such as Insulation Type or Insulation Purpose, then the **Matching** list contains only those insulation specifications that agree with the properties already chosen; nonetheless, you can choose a specification from the **All** list.

   - If you assign a heat tracing to an item before you assign insulation properties, your choices of insulation properties and specifications are restricted.

5. Select **OK**.

Using the PipeSpec Utility: An Overview

The Piping Specification utility works with PDS 3D or SmartPlant 3D to validate the piping materials class with the temperatures, pressures, and diameters assigned to the pipe run and to search commodity codes and fabrication categories for piping components. The database tables and library files in the 3D product provide source information for the validation and search. The service limits validation and automatic commodity code lookup can be disabled simultaneously using a switch in Options Manager. For more information about modifying the PipeSpec settings, see *Options Manager Help*.

In Data Dictionary Manager, the **ValidateNomDiam.ForeignCalc** program ID, which is assigned to the **Nominal Diameter** property, starts the Piping Specification utility and triggers the commodity code and fabrication category lookups when a nominal diameter is changed. For more information about assigning program IDs, see *Data Dictionary Manager Help*.

Note:

- Error messages are placed in the **PipeSpecError.log** file in the directory assigned to the TEMP environment variable. Error messages help you identify the cause of failure when the utility does not complete the tasks as expected. For example, if minimum requirements are not met for the lookup, the missing properties are listed in the log file.

- The **ServiceLimits.log** file contains any errors encountered during the Service Limit Validation process, which runs as part of the PipeSpec Utility.

PDS 3D Files Used For PipeSpec

- **pd schema** - pdtable_102 table

- **ra schema** - pdtable_201 and pdtable_202 tables

- **library files** - us_pjstb.l, us_pjstb.l.r, and us_pjstb.l.t

- **.dll files** - PipeSpec.dll, pdpjs.dll, pdpjsx.dll, and ValidateServiceLimits.dll

Performing Service Limits Validation

The Piping Specification utility verifies that the temperatures and pressures assigned to a pipe run comply with the service limits associated with the selected Piping Materials Class. In continuous validation mode, which is activated by assigned settings in Options Manager, this verification occurs each time that you modify either the Piping Materials Class or a temperature–pressure pair in the process case data of the pipe run. The Service Limits validation requires at least one complete temperature-pressure pair from among design, alternate design, operating, and alternate operating cases. If any temperature-pressure pair violates the service limits of the selected Piping Materials Class, a warning displays the appropriate pairs. This warning appears in the design software by appending an error string to the name of the PMC.

Performing Commodity Code and Fabrication Category Look Up

The Piping Specification utility looks up the **Commodity Code** and **Fabrication Category** properties of inline piping components. In the continuous validation mode, this lookup occurs each time the **Piping Materials Class** or any of the four case **Max** temperatures (**Design**, **Alternate Design**, **Operating**, and **Alternate Operating**) are modified on the pipe run. Validation also occurs each time the **Option Code** or **Nominal Diameter** of the component is modified. If the modification occurs on a property of a piping component, then the lookup is restricted to that particular component, but if the modification occurs on a property of a pipe run, then the lookup encompasses every piping component on that run.

The minimum requirements to cause a lookup are that the piping component must be in a pipe run, that the PMC of the pipe run must be populated and comply with service limits, and that the nominal diameter of the piping component must be specified. If the PMC is assigned but does not comply with the service limits, then the **Commodity Code** property displays an error message.

The PipeSpec utility uses process case temperatures of the run during the commodity code lookup only if the code for that component has a maximum temperature limit value in the 3D database. For example, in PDS 3D, a value of -9999 for maximum temperature in pdtable_202 indicates a null value, and the process case temperatures on the pipe run are ignored for the lookup. If a maximum temperature exists for that component, then the lookup insures this value is larger than all of the process case temperatures assigned to the pipe run in which the piping component resides.

⚐ **Note:**

- The units for the PDS 3D maximum temperature are those specified in Options Manager.
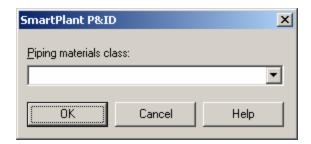
If any temperature values for the pipe run are unspecified, then a value of zero Deg-K is assumed for each of the unspecified temperatures. If multiple records are obtained in the lookup, then the utility returns a commodity code only if all of the records have the same code value. If not, an error is recorded in the error-log file with the appropriate message.

The **Fabrication Category** property of inline piping components is a select-listed property in SmartPlant P&ID. A relationship between the fabrication category and the commodity name can be defined in the 3D databases. The **Commodity Name** is a unique name for every symbol. In PDS 3D, this unique name is the **AABBCC Code** property. SmartPlant symbols are assigned the same **AABBCC Code** properties in Catalog Manager.

Similarly, the **Option Code** property is a select list of text values in SmartPlant P&ID, while it is a set of code numbers or indices in PDS 3D. **Short Value** for the **Option Code** select list contains the PDS 3D indices corresponding to the appropriate **Option Code** text in SmartPlant P&ID. The PipeSpec utility uses the entries in the **Short Value** box of the **Option Code** list to obtain the **Option Code** used in the PDS 3D database tables.

Assign a Piping Materials Class to a Pipe Run Using PipeSpec

⚐ **Notes:**

- The PipeSpec utility is not available unless the appropriate settings and program IDs are assigned in Options Manager and Data Dictionary Manager, respectively. For more information, see those Help files.

1. Select the pipe run.

2. Select the **Piping Materials Class** property for the pipe run in the **Properties Window**.

3. Click the **Calc** button 🔲.

4. On the **Piping Specifications** dialog box, choose the appropriate PMC from the list and click **OK**.

Notes:

- Based on the chosen PMC and other assigned properties, such as **Nominal Diameter**, **Maximum Design Pressure**, **Maximum Design Temperature**, and so forth, the PipeSpec utility copies properties and checks limits in the entire run, including inline components.

- If you subsequently assign nominal diameters to inline valves or other components, SmartPlant P&ID assigns commodity codes and fabrication categories to them that correspond to the PMC of the pipe run and the specifications in your 3D databases. If you choose a diameter, for instance, that does not exist for the assigned PMC, then the commodity code property becomes **Not In Spec**.

- If you choose temperature-pressure limits for process cases that do not agree with the assigned PMC, then the error **Service Limits Error** is added to the PMC for the run.

- Given the appropriate settings in Options Manager and Data Dictionary Manager, you can manually assign commodity codes to inline components, and the PipeSpec utility looks up accepted diameters and validates the assignment.

Assign a Commodity Code to a Piping Component Using PipeSpec

Notes:

- The PipeSpec utility is not available unless the appropriate settings and program IDs are assigned in Options Manager and Data Dictionary Manager, respectively. For more information, see those Help files.

1. Select an inline component that belongs to a pipe run whose piping material class has been defined.

2. In the **Properties** window, define the **Nominal Piping Diameter** property for the component.

Notes:

- You can verify that a commodity code and fabrication category has been assigned in the **Properties** window, or you can open the error log, PipeSpecError.log, if properties have not been assigned as expected.

- If continuous validation is turned on for the PipeSpec utility, then a pipe run with temperature-pressure limits that do not agree with its PMC produces the error **Error in PMC** in the commodity code for an inline component

- With continuous validation, not only is the commodity code defined automatically, but so it the **Fabrication Category** property for the inline components, given that the relationship exists in the 3D databases.

- Adding or changing the option code can change the commodity code, if the relationship is defined in your 3D database.

- If continuous validation is not activated but other appropriate settings and program IDs are assigned, you can still click the **Calc** button for the **Commodity Code** property and choose a code. However this does not verify that the component agrees with a piping specification.

Required ProgIDs for PSPEC Access

Database Table	Table Property	Calculation ID	Validation ID
*Piping Component	Option Code	PipeSpec.OptionCode Finder	PipeSpec.Commodity CodeValidator
*Pipe Run	Nominal Diameter	PipeSpec.NPD Finder	
Pipe Run	Piping Materials Class	PipeSpec.PMC Finder	PipeSpec.Com modityCodeVa lidator
Inline Component	CommodityCode	PipeSpec.Commodity CodeValidator	
Inline Component	Nominal Diameter		ValidateNomDiam .ForeignCalc
Case Process	Temperature		PipeSpec.Commodity CodeValidator
Case Process	Pressure		PipeSpec.Commodity CodeValidator

Important: The Calculation IDs for the **Nominal Diameter** property in the Pipe Run table and the **Option Code** property in the Piping Component table (items marked with an (*) asterisk) are available only when the pipe specification source is SmartPlant 3D. These validations are not available when the piping specification source is PDS 3D.

Printing Drawings: An Overview

You can print your drawings on many different devices ranging from dot matrix printers to laser printers and plotters. Except for the color settings and a few special effects, your drawing prints exactly as it appears in the Drawing view. This stipulation applies to special settings like the display of claimed items, inconsistency indicators, or filtered items.

Notes:

- Zero length pipe runs will print. For example, when a reducer is connected to a nozzle, the zero length pipe run between these symbols will print. Zero length pipe runs will not print if you print your drawing using Drawing Manager.

Printing the Active Drawing: An Overview
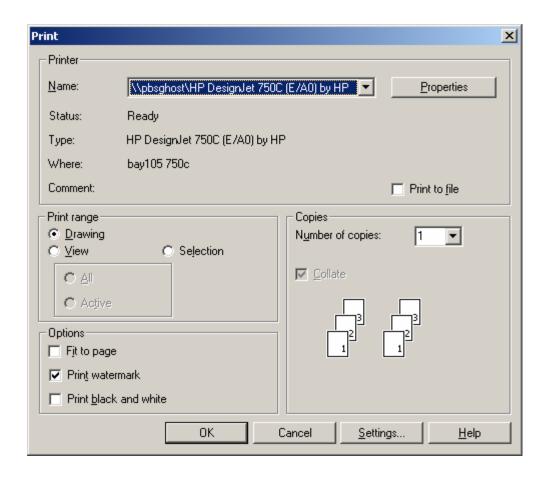
Print Command

The **File** > **Print** sends a copy of the active drawing to a specified plotter, printer, or file. Options are available for defining the printing area, range, number of copies, and other printing characteristics. Selecting File > Print also opens the Print dialog box.

Before using this command, you must install and select a printer. For help on installing a printer, see the printer documentation.

Print Dialog Box

Controls how a drawing is printed. This dialog box opens when you click **File > Print** on the menu bar.

Printer:

- **Name** - Specifies the printer you want to use. You can select from a list of all the available configured printers. The information below the **Name** box applies to the selected printer. The printer you select in the **Name** box is the default printer for the rest of the current design session until you specify a different printer.

- **Properties** - Opens the **Printer Document Properties** dialog box, which allows you to specify page setup and other printer settings.

- **Status** - Describes the state of the selected printer, such as, busy or idle. This area is read-only.

- **Type** - Displays the type of printer currently selected. This area is read-only.

- **Where** - Identifies the printer path, printer port, queue name, or physical location of the currently selected printer. This area is read-only.

- **Comment** - Displays any comments you entered during printer configuration. This area is read-only.

- **Print to file** - Stores your drawing in a file with extension .prn instead of sending it to a printer. The **Print to File** dialog box appears when you select the **Print to file** option on the **Print** dialog box and then click **OK**. You select a file name and location for the print file in the **Print to File** dialog box. Then you can print from a computer that does not have the application installed or print to a printer other than the one you currently have configured.

Print Range:

- **Drawing** - Prints your entire drawing.

- **View** - Activates the **All** and **Active** check boxes so you can then define the view or views to print.

- **Selection** - Prints the user-defined area. When you select this option, the **Settings** button at the bottom of the **Print** dialog box is unavailable.

- **All** - Prints each defined view associated with the drawing.

- **Active** - Prints only the active view.

Options:

- **Fit to page** - Prints your entire drawing on one page.

- **Print watermark** - Prints a faint graphic in the drawing background.

- **Print black and white** - Prints the drawing in black and white.

Copies:
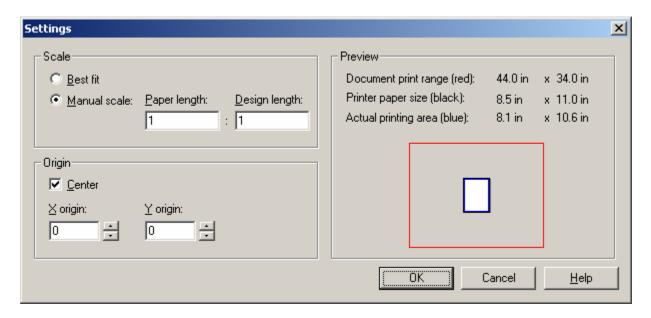
Number of copies - Displays the number of copies you want to print. Type the number or use the scroll buttons to specify a number.

Collate - Prints the copies in proper binding order.

Settings - Opens the **Settings** dialog box, which allows you to view and edit the scale and origin of your print area. This button is disabled when the **Selection** option in the **Print range** group is selected.

Settings Dialog Box

Specifies the area you want to print. This dialog box opens when you click **Settings** on the **Print** dialog box.

Scale:

- **Best fit** - Scales the selected drawing sheets or print area to fit the printer paper for the configured device.

- **Manual scale** - Specifies the scale value to apply to the print range during printing. For example, if the print range is a rectangle at 12 cm by 12 cm and you set a manual scale of 1:12, then the printed range appears to be 1 cm by 1 cm on the printer paper. If you want a 1:1 drawing of the current sheet scale, you can set the **Paper length** option to 1 and the **Design length** option to 1.

  o **Paper length** - Specifies the paper length for the document you want to print with respect to the **Design length** option.

  o **Design length** - Specifies a design length (size of the printed graphic) with respect to the **Paper length** option.

Origin:

- **Center** - Positions the print area center to the center of the printer paper. If you do not set this option, then the paper positions at bottom left to bottom left.

- **X origin** - Sets a shift in the x-direction from the origin.

- **Y origin** - Sets a shift in the y-direction from the origin.

Preview - Displays dynamically how the graphic prints on the sheet as you change other options on the dialog box.

Printing Multiple Drawings: An Overview

Rather than printing drawings one-by-one from within SmartPlant P&ID, Drawing Manager allows you to print more than one drawing at a time. You can select drawings for printing from any number of units or areas in a plant structure to be placed into the print queue and can specify printing options, such as watermark, whether to print inconsistencies, and drawing orientation, for the entire group of selected drawings. You also have the capability in Drawing Manager to schedule batch printing for a later time or at regular intervals by using the **Schedule** option on the **Print** dialog box.
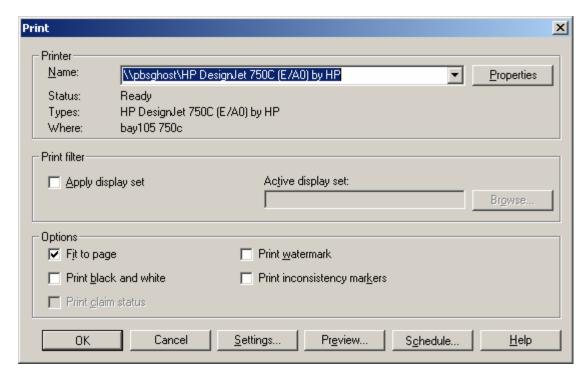
While you can print just one drawing at a time from within Drawing Manager, the **Print** command in Drawing Manager does not have all the single-print capabilities of the **Print** command in SmartPlant P&ID. For instance, Drawing Manager only prints an entire drawing; whereas, in SmartPlant P&ID, you can print only a selection or a view inside one drawing.

Print Command

The **File > Print** from Drawing Manager opens the **Print** dialog box where you can specify options for printing your drawings.

Print Dialog Box

The print dialog box controls how a drawing is printed. This dialog box opens when you click **File > Print** from Drawing Manager.

Printer:

- **Name** - Specifies the printer that you want to use. You can select from a list of all the available configured printers. The information below the **Name** box applies to the selected printer. The printer that you select in the **Name** box is the default printer for the rest of the current design session until you specify a different printer.

- **Properties** - Opens the **Printer Document Properties** dialog box, which allows you to specify page setup and other printer settings specific to the selected printer.

- **Status** - Describes the state of the selected printer, for example, busy or idle. This area is read-only.

- **Types** - Displays the type of printer currently selected. This area is read-only.

- **Where** - Identifies the printer path, printer port, queue name, or physical location of the currently selected printer. This area is read-only.

Print Filter:

- **Apply display set** - Associates symbols to a filter that defines which symbols to print. If not selected, then no filtering is applied to the drawing and the entire drawing prints.

- **Active display set** - Displays the selected display set. You can click Browse to select a defined display set.

Options:

- **Fit to page** - Prints your entire drawing on one page.

- **Print black and white** - Prints the drawing in black and white.

- **Print claim status** - Allows for the printed copy to include the status of claims. This check box is only enabled if you are in a Project.

- **Print watermark** - Prints a faint graphic in the drawing background.

- **Print inconsistency markers** - Causes inconsistency markers in the graphic to appear on the printed copy.

- **Settings** - Opens the **Settings** dialog box, which allows you to view and edit the scale and origin of your print area.

- **Preview** - Allows you to see how the image will look when printed. The image displays on your monitor.

- **Schedule** - Opens the **Schedule Task Wizard**, which allows you to specify options for printing the selected drawings at a later time or on a regular interval.

Generating Reports: An Overview

Reporting is the process of retrieving information from the database and displaying the information as formatted output. At any time during the design creation process, you can create a report. Each report consists of a Microsoft Excel workbook and a report definition, which describes the data to collect and how to organize the data in the workbook.

Each report that you create is based on an item type. This item type serves as the starting point for collecting data for your report. Examples of item types include equipment, nozzle, instrument, and pipe run. Several default report templates already exist; however, reports are fully customizable. You can create your own reports that contain the information that you want to see in a format you choose.

The relationships that exist between the various item types constitute additional available information for a report. For example, a nozzle is related to the equipment with which it is grouped. When creating your report, only items that have a relationship with your selected item type can be used as input. In order to discover how items and their properties are related, and thereby how you can map the properties that you want into your report templates, see the *Properties Glossary*, which is included in the glossaries attached to online Help.

Your report definition contains one or more report items organized in a tree hierarchy. Each report item is based on an item type. Each report contains at least one report item to define the item type of the report. For example, a report based on the equipment item type contains a report item named **Equipment**. This report item makes the properties associated with each piece of equipment available for inclusion in your report; however, you do not have to include every available property in your report if it is not appropriate. You can define additional report items to access more properties for more item types. For example, a nozzle report item can be added to access data about nozzles because nozzles and equipment are related.

The location of a report item in the tree hierarchy affects the properties that are collected for the associated item type. For example, if a nozzle is added as the top-level item in the tree, all nozzles in the database are collected for your report. If the nozzle is added as a child of **Equipment: Mechanical**, only the information about nozzles that are associated with pumps is collected.

You begin your reporting process by selecting a report template from the **Reports** menu. Then the software performs the following tasks:

1. Microsoft Excel starts. The report template is copied to the report output folder, and then the Excel workbook opens.

2. Your report definition is retrieved from the Excel workbook.

3. Your data is retrieved based on the report item definitions of the report template.

4. Data prints to the Microsoft Excel workbook using the cell mapping data in your report definition.

   Notes:

   - You must have installed Microsoft Excel on your computer to display reports.

   - The software stores the reports that you generate in your Windows directory under **~\Profiles*username*\My Reports\Output**.

   - In order to generate From and To data for all Microsoft Excel worksheets, from the Excel menu bar chose:

     i. **Tools** > **Macro** > **Macros** > **PrintFromToDataForAllSheet**

     OR

     ii. **Tools** > **Macro** > **Macros** > **PrintFromToDataForActiveSheet**
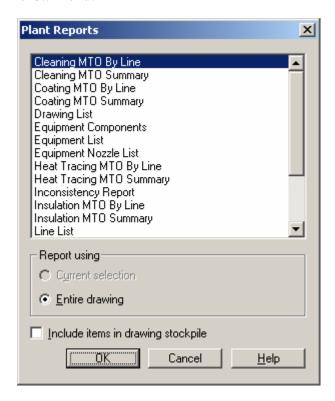
Plant Reports Command

The Reports > Plant Reports opens the **Plant Reports** dialog box, which displays a list of all plant-level reports associated with the current plant. This list is alphabetical. The location of these report templates is defined in Options Manager. Selecting a plant report from this list and specifying the items that you want to report on generates the associated report in Microsoft Excel.

This command is available on the **Reports** menu on the main toolbar and also on the **View** menu in the **Engineering Data Editor**. If you have a drawing in the **Design** window and you want to report on items in that drawing, use this command from the **Reports** menu. If you want to run a report based on items in your **Engineering Data Editor**, use this command from the **Engineering Data Editor View** menu.

Plant Reports Dialog Box

The **Plant Reports Dialog Box** allows you to choose a report and specify what items to report on. This dialog box opens either when you click **Reports > Plant Reports** on the main toolbar or when you click **View > Plant Reports** on the **Engineering Data Editor** toolbar. If you want to run a report on drawing items, use the **Reports** menu; if you want to run a report on table items, use the **Engineering Data Editor View** menu.

Report using: Displays options for specifying the scope of your report. The options that appear in this area depend on the view, **Drawing** view or **Engineering Data Editor**, which is active when you access this dialog box.

- **Current Selection** - Produces a report containing the items currently selected in your drawing or table. This option is not available if no drawing or table items are selected. You cannot choose the **Include items in drawing stockpile** option when you choose **Current Selection**.

- **Entire drawing** - Produces a report containing the contents of the entire drawing. This option is available only when you use the **Reports** menu command.

Include items in drawing stockpile - Allows you to specify whether or not you want items that reside in the drawing stockpile in your report. This option is available only when you use the **Reports** menu command.

My Reports Dialog Box

The **My Reports Dialog Box** allows you to choose one of your custom reports and to specify what items to report on. This dialog box opens either when you click **Reports > My Reports** on the main toolbar or when you click **View > My Reports** on the **Engineering Data Editor** toolbar. If you want to run a report on drawing items, use the **Reports** menu. If you want to run a report on table items, use the **Engineering Data Editor** menu.
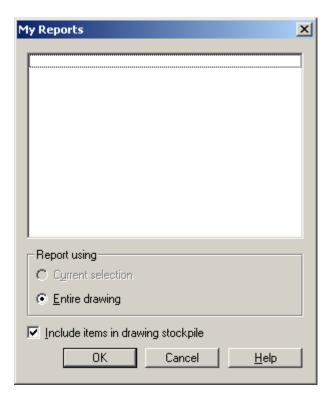
Report using - Displays options for specifying the scope of your report. The options that appear in this area depend on the view, **Drawing** view or **Engineering Data Editor**, which is active when you access this dialog box.

- **Current Selection** - Produces a report containing the items currently selected in your drawing or table. This option is not available if no drawing or table items are selected. You cannot choose the **Include items in drawing stockpile** option when you choose **Current Selection**.

- **Entire drawing** - Produces a report containing the contents of the entire drawing. This option is available only when you use the **Reports** menu command.

Include items in drawing stockpile - Allows you to specify whether or not you want items that reside in the drawing stockpile in your report. This option is available only when you use the **Reports** menu command, not the **Engineering Data Editor** command.

Importing Drawing Data: An Overview

There are several ways you can import data into SmartPlant P&ID. You can import data and items into the database using reports and other files. SmartSketch files can be imported into SmartPlant P&ID using the SmartPlant Migrator wizard. Also, you can also populate or redefine pipe run properties by importing an Aspen Zyqad stream data file. During the import process, you can assign the stream and its associated engineering data to the pipe run.

The log file for import activities is called SPImport.log and is saved to your local **Temp** directory.

Relationships between items cannot be imported, only items and their properties.

Import Data File Command

The File > Import > Data File allows you to import data and items into the database from reports and other files.

Using Reports to Import Items into the Stockpile

You can use the Equipment List and the Pipe Run List to import new items into the stockpile. This capability allows you to define values for an item that does not yet exist in the database. Both of these reports have hidden columns that store some key information that is necessary for importing this information.

Also, you can use a report to import information for items that already exist in the database. You must first run the report, add or edit property values for the items returned in the report, and then import the items back into the drawing. The property values for the items are updated. Valid reports are Equipment List, Equipment Nozzle List, and Pipe Run List.

Modify a Pump by Importing a SmartPlant P&ID Report into the Stockpile

1. Place a pump in a drawing and assign an item tag, for example, **P-100A**.

2. Run the **Equipment List** report.

3. In the resulting report, change the value in the **Item Tag** cell (for example, change **P-100A** to **P-101D**) by highlighting the last column in the report and

the column right next to it, then clicking **Format > Column > Unhide**. The path name of the associated symbol is available in column **T**. Columns **Y**, **Z**, and **AA** display the **Tag Prefix**, **Tag Sequence Number**, and **Tag Suffix** properties, respectively. In this example, these should be updated to **P**, **101**, and **D**, respectively.
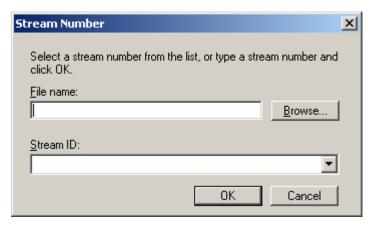
4. Select **File** > **Save**.

5. Select **File** > **Close**.

6. In **SmartPlant P&ID**, click **File** > **Import** > **Data file**.

7. Browse to the **Equipment List** report that you just created and modified.

8. Select **Open**.

Import Aspen Zyqad Stream Data

1. Select a pipe run from the drawing to display the Properties window for the pipe run.

Notes:

*   If the **Properties** window is not open, you can click **Edit > Properties** after you select the pipe run.

2. Select the **Calc** button next to the **Stream No**. property to display the **Stream Number** dialog box.

3. Select the **Browse** button and select the Aspen Zyqad stream data .txt, .xls, or .xml file from the appropriate folder.

4. Select the **Stream ID** list and select a **Stream ID**.
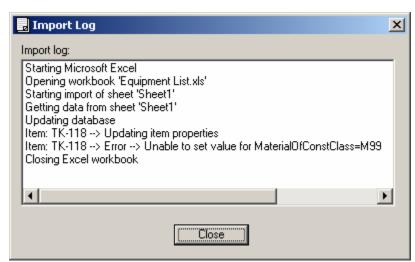
5. Select **OK** to populate the **Property Window** with the selected **Stream ID**.

Populate the Stockpile from a Spreadsheet

1. Select **File** > **Import** > **Data File**.

2. Browse to the location of your spreadsheet, select it, and select **Open**.

   - The status bar displays the progress of the import activity.

   - You can populate the stockpile with items from Aspen Zyqad exported spreadsheets and some SmartPlant reports.

1. If Errors are encountered during the import process the **Import Log** dialog box will be displayed. If no errors are encountered the **Import Log** dialog box will not be displayed but may be found the ~\temp folder.

 Notes:

- If a format change is made to an exported Aspen Zyqad report, you must make modifications the import code to accommodate the changed report layout.

- Aspen Zyqad spreadsheets in either Microsoft Excel or XML format are supported. These spreadsheets include the Vessel Equipment List, Pump Equipment List, and Heat Exchanger Equipment.

- Stockpile items that come from importing Aspen Zyqad data can be easily updated by re-importing a data file. Previously imported items, whether they remain in the stockpile or have already been placed in the drawing, are updated with data from the newly imported file.

- Importing Aspen Zyqad XML data files allows greater flexibility when importing data into the software. All three reports, Equipment, Pumps, and Exchangers, are included in a single XML report so that only one file has to be imported.

- You can use the Equipment List, Pipe Run List, and Equipment Nozzle List Reports from SmartPlant to define properties of items already placed in the drawing, too. Items must have an assigned item tag prior to definition or modification of properties. In addition, these reports can create occurrences of items in the stockpile.

- See the *SmartPlant P&ID Programmer's Guide Help* file for more information on import code and other importing parameters. Click **Help > Programming with SmartPlant P&ID**.

Using Drawing Revisions: An Overview

You can create revisions for drawings consisting of major and minor revisions and include the revision information in the drawing title block if desired. You can also associate a version with the latest revision if you need to archive drawings for viewing or comparing. You can compare two archived drawings or you can compare an archived drawing with the current drawing.

📝 **Notes**:

- The combination of the major and minor revision values must be unique in the plant, or when working As-Built, in the project.

- When working in an integrated environment, when you select the **New Revision** command, the software opens the SmartPlant Foundation **Revisions** dialog box instead of the SmartPlant P&ID **New Revision** dialog box. If you want to associate a version with a new revision in an integrated environment, you must select the drawing and run the **Revisions** > **Associate Version** command after creating the revision.

New Revision Command

The **Revisions** > **New Revision** command opens the **New Revision** dialog box, allowing you to create a new revision for the selected drawings and to enter values for the revision properties. At the time you create a Revision you can also associate a version with a new revision.

New Revision Dialog Box

The New Revision Dialoog Box allows you to enter values of properties pertaining to the creation of a new revision that applies to one or more drawings. This dialog box opens when you select the drawings and click **Revisions** > **New Revision**.
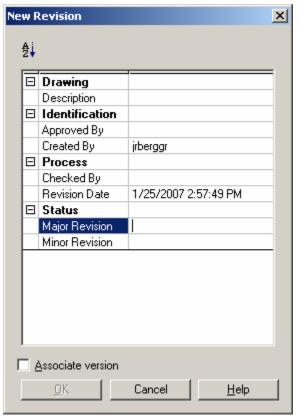
Alphabetized Button– Displays drawing properties alphabetically. This button acts as a toggle and is available when properties are displayed categorically.

Categorized Button– Displays drawing properties in categories. This button acts as a toggle and is available when properties are displayed alphabetically.

(Properties and Values List) – Displays a list of shipped properties used for the revision. You can specify additional properties for display on this dialog box by defining them in the Data Dictionary Manager. Note that **Major Revision** is a mandatory property and that the combination of the **Major Revision** and **Minor Revision** values must be unique in the plant or project.

Associate version – On clicking **OK**, opens the **New Version** dialog box, which allows you to create a new version and associate it with the revision.

OK – Generates the new revision, and if specified, an associated version. This command only becomes available when you enter a value for **Major Revision**.

Revision History Command

The **Revisions** > **Revision History** opens the **Revision History** dialog box, which displays a record of the revisions of the selected drawing.

Revision History Dialog Box

The **Revision History Dialog Box** lists all available revisions of a drawing. You can open this dialog box by clicking **Revisions** > **Revision History**.

History
Lists all the revisions of the drawing in the current plant or project. Note that when you check out or fetch a drawing from As-Built into a project, only the last revision record is displayed on this dialog box, regardless of whether that revision is associated with a version or not.

Major Revision – Indicates the number or other designation of the major revision. This is a required property, so a value will always appear in this column.

Minor Revision – Indicates the number or other designation of the minor revision. This is not a required property, so a value may or may not appear in this column.

Revision Date – Shows the date on which the revision was created.

Approved By – Shows the name of the person who approved the revision.
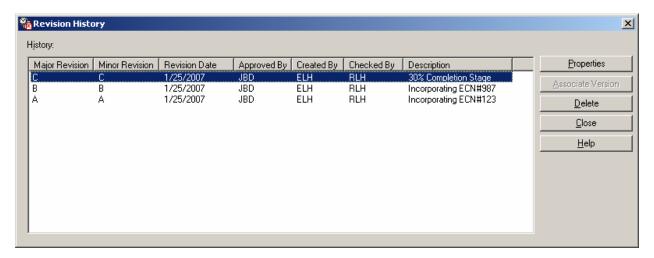
Created By – Shows the name of the user who created the revision.

Checked By – Shows the name of the person who checked the revision.

Revision Properties – Opens the **Revision Properties** dialog box, allowing you to view or edit all of the revision properties.

Associate Version – Opens the **New Version** dialog box, allowing you to create and associate a new version with this revision. This option is only available for the last revision.

Delete – Removes the selected drawing revision. To be able to delete revisions, you must have appropriate permissions assigned in SmartPlant Engineering Manager. If the revision has an associated version, the software deletes the version together with the revision.

Using Drawing Versions: An Overview

Drawing versions allows you to create, compare, and recover versions of your drawings. Using drawing versioning is helpful in the following situations.

- Restoring a drawing after it has been deleted

- Restoring a drawing after items have been deleted from the drawing

- Restoring a drawing after making design errors

- Archiving a drawing before making major design changes

Notes:

- If utilizing As Built/Projects, when a drawing is checked in, all versions in the project are deleted.
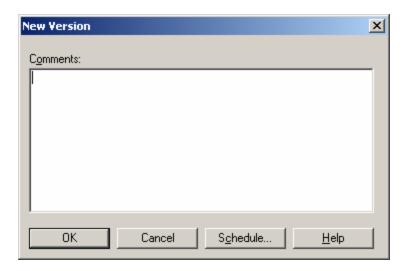
Creating a New Version of a Drawing: An Overview

The current state of a drawing and all associated data can be saved using the New Version command. Previously saved drawing versions are available to be viewed, compared, and fetched, if necessary. If drawing versions are saved on a regular basis, the sequence of drawing versions in a project becomes a sort of archive that shows the development of the drawing over the life of the project.

Each time a drawing is checked into the Plant database, a new version of the drawing is automatically created in the Plant and all versions of that drawing are deleted from the project. The drawing versions in the Plant show the changes to the drawing over the life of the Plant.

Save a Version of a Drawing

1. From Drawing Manager, in the List view, select a drawing.

2. Select **Versions** > **New Version** or right click and select **New Version.**

3. On the **New Version** dialog box, enter any comments that you want to attach to the new version.

4. Select **OK** to create the version.

   📝 **Notes:**

- Click **View Log** on the **Creating Version of Drawings** dialog box to open the log file and review notes on the version creation operation.

- **New Version** skips open drawings, notes them in the log, and then continues.

- Select **View Log**

************************* NewVersion *************************
04/29/2005 08:39:58 - INGRPPO\jrberggr Performed NewVersion.
   Operation : NewVersion
   Drawing :   Lab5
   Status   : **Failed. Error:** Drawing may be currently opened or require a recreate from database. Failed to create new version. Failed to create New Version.

```
            Operation  : NewVersion
            Drawing :   41-PI-0001
            Status    : Completed successfully.
            ----------------------------------------------------------------------------
            Operation  : NewVersion
            Drawing :   38-PI-0001
            Status    : Completed successfully.
            ----------------------------------------------------------------------------
```

- If no changes have been made to the drawing since the last version was created, no new version is created.

```
************************* NewVersion *************************
04/29/2005 08:43:33 - INGRPPO\jrberggr Performed NewVersion.
            Operation  : NewVersion
            Drawing :   38-PI-0001
            Status    :Nothing to do.
```

- Use the Schedule button to create a task for creating drawing versions at a later time or on a regular interval. Follow the instructions on the Schedule Task Wizard.

- When a drawing is checked in to the Plant, all versions are deleted from the project.

Save Versions of All Drawings

1. In the Tree view, select the site, plant, or project containing the drawings you want to save versions of.

2. Select **Versions** > **Incremental New Versions**.

3. Follow the steps and directions on the **Schedule Task Wizard**, and select **Finish** to schedule the operation at another time or on a regular interval.

   📝 **Notes:**

   - When saving versions of all drawings in a Plant that has projects, new versions of the drawings belonging to those projects are created also.

   - This procedure saves new versions of only those drawings whose time stamp shows that they have changed since the last version was saved.

Recovering Drawings: An Overview

The following areas require special consideration with regard to saving new drawing versions and how retrieving a drawing effects the actions that can occur in the Plant between saving a version and retrieving it. These actions have serious implications when recovering (using the Fetch Deleted Drawings command) a drawing. In all drawing recovery activities, a log file is created in which you can review notes on any recovery activity that you are undertaking.

Multiple Representations

After a drawing is recovered, there are situations where multiple representations of piping and equipment items can spontaneously occur. For example, you place a piece of equipment on drawing A and then you save a version of drawing A. After creating a version of drawing A, you move the equipment from drawing A to the Plant Stockpile and then to drawing B. When drawing A is retrieved, the following message is added to the log file:

```
Item (item tag ItemTag, internal ID SP_ID) is being restored as a
multiple representation because another representation of the same
item was found in drawing Drawing Name.
```

Encountering this situation does not cause the retrieval to fail; the retrieval process continues as normal.

If an equipment item already exists as a multiple representation in another drawing, the following message is added to the log file:

```
Restoring multi representation item (item tag ItemTag, internal ID
SP_ID).
```

Encountering this situation does not cause the retrieval to fail; the retrieval process continues as normal.

If other valid stockpile items are moved from drawing A to drawing B, the following message is added to the log file:

```
Error! Item (item tag ItemTag, internal ID SP_ID) has been moved to
drawing Drawing Name.
```

To resolve this conflict, you must either delete the indicated item or restore the indicated drawing first. If you delete the item, then the item can either be deleted to the Plant Stockpile or deleted from the model. This error message is created for each moved item, and the retrieval process quits without restoring the archive.

Off Page Connectors (OPCs)

If you save a new version of a drawing that includes an OPC, and its mate is deleted from the database afterward, both OPCs are restored to the drawing or drawing stockpile, as appropriate, when the drawing is retrieved.

For example, an OPC is placed on a drawing A, its mate is placed on drawing B, and both drawings saved in versions. When drawing A is retrieved, the OPC is restored to drawing A, and the mate is placed in the Plant Stockpile. The mated OPC has the same item tag as the OPC restored to drawing A. Once drawing B is retrieved, the OPC mate of the OPC in drawing A is placed in drawing B and removed from the Plant Stockpile.

Pipe and Signal Lines

If all the line runs belonging to a line are deleted from the model after a drawing version is saved, the line is restored back to the database after the drawing is recovered.

To restore a deleted line, Drawing Manager searches the database for a line that has the same key property values as the line that is being restored. If such a line is found, it is used as the line for the restored runs. If a suitable line is not found one is created for the restored runs.

Plant Group Joins

Plant Group Joins, which relate items in plant groups, are restored from a version only if the plant groups, such as the unit or area, exist in the current database.

For example, a piece of equipment belongs to a plant group and a drawing version is saved. If the plant group is deleted and then the drawing is recovered, the equipment is restored, but because the plant group does not exist, the Plant Group Join is not restored.

If the plant item group is found in the archived drawing, but the Plant Group Join does not exist in the current database, Drawing Manager restores the Plant Group Join.

Plant Item Groups

Plant Item Groups placed in the drawing stockpile are considered part of the drawing; therefore, Drawing Manager restores them to the drawing stockpile when the drawing is recovered.

Plant Item Groups that are moved to the Plant Stockpile after a drawing version has been saved are restored back to the drawing stockpile when the drawing is recovered. The software searches for the corresponding Plant Item Group in the saved stockpiles, and if it is not found there, searches in the active database for the same.

Plant Item Group Joins

A Plant Item Group Join is a relationship created when an plant item, such as an instrument or a piece of equipment, is linked to a Plant Item Group, such as an instrument loop, a package, or the like. Plant Item Group Joins are saved as part of the drawing version.

During a drawing recovery, if a Plant Item Group Join exists in the saved drawing version, the software searches for the corresponding Plant Item Group in the archived stockpiles. If the corresponding Plant Item Group is not found in those stockpiles, the database is also searched.

If the Plant Item Group is found in the saved version, and the Plant Item Group Join is not found in the database, then the Plant Item Group Join is restored. If the Plant Item Group is not found in the saved stockpiles, the Plant Item Group and the Plant Item Group Join are restored to the Plant Stockpile. If the Plant Item Group exists in the current drawing stockpile, Drawing Manager updates the database to reflect the archived Plant Item Group Join.

For example: An instrument is associated with a Loop, LP1, in the drawing stockpile and a version is saved. Afterward, a new Loop, LP2, is placed in the drawing stockpile and the instrument is associated with LP2. When the drawing is restored, the Plant Item Group Join indicates a relationship between the instrument and LP1. If LP1 has since been deleted from the drawing stockpile, it is restored to the drawing stockpile. If LP2 exists in the current Plant Stockpile at the time of drawing recovery, LP2 is left as is. However, if LP2 is in the drawing stockpile, Loop LP2 is deleted from the database along with any other corresponding representations and histories of Loop LP2.

Miscellaneous

If a drawing is deleted after a version is saved and a new drawing is created using the same name and drawing number as the deleted drawing, retrieval of the deleted drawing fails. Changing drawing properties, such as name, number, and so forth, after saving a version of a drawing result in the original values being restored when the drawing is recovered. If this situation occurs, the following message is added to the log file:

```
Warning! Drawing drawing name1 has been renamed to new drawing name2.
```

Drawing Manager changes the drawing back to its original name, *drawing name1*, in the database. The original .pid file *pathname\drawing name1* is also replaced. You must delete the .pid file for *pathname\drawing name2*.
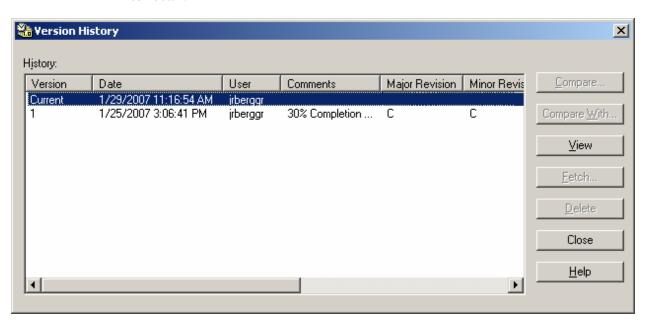
Notes:

- You must have either site administrator or modify privileges to save versions or recover drawings.

You cannot restore hierarchy items by using drawing recovery. For example, if a unit is deleted, an archived drawing belonging to that unit can never be retrieved

Recover a Version of a Drawing
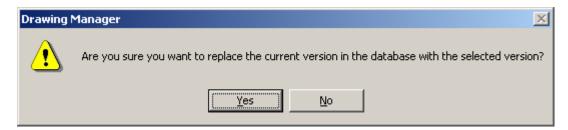
1. In the List view, select the drawing that you want to recover.

2. Select **Versions** > **Show History** or click the **Show History** button.

3. On the **Show History** dialog box, select the version of the drawing you want to recall.
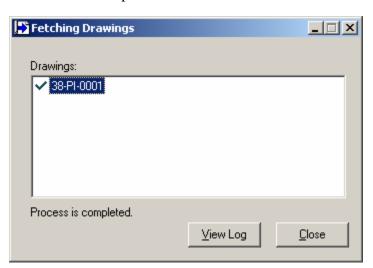
4. Select **Fetch** and enter comments on the **Fetch Comments** dialog box.

Fetch Comments

Comments:

OK   Cancel   Help

5. On the message box, select **Yes** to confirm that you want to overwrite the current version with the saved version you have selected.

Drawing Manager

⚠ Are you sure you want to replace the current version in the database with the selected version?

Yes   No

6. Click **View Log** on the **Fetching Drawings** dialog box if you want to see notes about this operation.

Fetching Drawings

Drawings:

✓ 38-PI-0001

Process is completed.

View Log   Close

**************** FetchVersionFromOtherProject ****************
04/29/2005 09:13:03 - INGRPPO\jrberggr Performed
FetchVersionFromOtherProject.
    Operation : FetchVersionFromOtherProject
    Drawing :   38-PI-0001
    Status   : Completed successfully.

🖉 **Notes:**

- Be sure you are familiar with the ramifications of drawing recovery before you undertake the operation. For more information, see Recovering Drawings: An Overview.

- You can also recover a version of a drawing by clicking Fetch. Using the Fetch command is particularly useful if you want to recover a version of your drawing that resides at the Plant or another project. For more information, see Fetch a Drawing.

Recover a Version of a Deleted Drawing

1. Select the appropriate plant level in the **Tree** view, and select **Versions > Fetch Deleted Drawing**.

2. On the **Fetch Deleted Drawing** dialog box, select the drawing you want to retrieve.

3. Select **OK** to retrieve the drawing now or click **Schedule** to open the **Schedule Task Wizard**, which allows you to schedule retrieval at a later time or on a regular interval.

4. On the **Fetching Deleted Drawing Status** dialog box, select **View Log** to review notes about this retrieval process.

Notes:

- All the saved versions of the selected drawing are retrieved. You can view the various versions by selecting the drawing in the List view and selecting **Versions** > **Show History**.

- If you retrieve a drawing that was new to your Plant or project, then the drawing icon for a new drawing will be applied; otherwise, the drawing icon for a fetched drawing is used in the **List** view.

Comparing Versions: An Overview

When more than one version of a drawing exists, the **Compare** command on the **Show History** dialog box allows you to view two versions side-by-side and examine their differences. You can compare two versions from inside your own Plant or project database, or you can compare a version in your database to a version in the Plant or another project database. Keep in mind that you can only compare a drawing against a version of itself; that is, you cannot compare one drawing to another drawing.

Differences between drawing versions are assigned to logical "change" groups, which are listed on the **Compare** dialog box.

Differences also belong to one of three possible categories:

- **Data** - Refers to a mismatch in the properties assigned to an item that exists in both drawings, namely a change, addition, or deletion of a property in the **Properties** window, in the **Engineering Data Editor** in SmartPlant P&ID, or through automation.

- **Graphic** - Refers to an item that has changed only in its graphical representation in the design (for example, the item is moved or otherwise graphically manipulated in the drawing).

The following differences are ignored: claim status, select list strings, linked or embedded objects, symbology, and inconsistency indicators.

Adding or deleting an item is also a connectivity change because the relationship between the item and the drawing has changed. Every change grouping and every changed item is assigned a category, and if more than one category applies (for instance, if you move an item and change one of its properties) then the highest priority category is displayed. The order of priority, from high to low, for the categories is connectivity then data then graphic.

The two versions are displayed in two **Drawing** views, described only as left and right. The relationship between the two views depends on whether you are comparing two versions in your own database or comparing your version to a version in another database.

- If the two versions are in the active database, then the left-hand view is the older version, and the right-hand view is the newer version. That is, they are displayed in time-order from left to right.

- If the two versions exist in different databases, you cannot be assured that time-order is the logical order to display the versions; therefore, the right-hand view is reserved for the version in your active Plant or project database, and the left-hand view belongs to the version in another database.

The **Compare** dialog box in Drawing Manager is useful reviewing differences between versions only. If you want to reconcile anything about the two versions you review, then you must do so inside SmartPlant P&ID. The **Compare and Refresh** command in SmartPlant P&ID exists for this purpose. If you started with the left-hand drawing version and applied every change listed on the **Compare** dialog box, then you would end with a drawing that is identical to the right drawing version.

Compare Versions of Drawings in Same Database

1. In the List view, select the drawing.

2. Select Show History.

3. In the drawing list on the Show History dialog box, select two versions of the drawing.

4. Select Compare. If you want to compare your drawing to a version in a different project or the Plant, see Compare Versions of Drawings in Separate Databases.

5. On the Compare dialog box, you can view the differences between the two versions, but you cannot make changes to the designs. To change the design, you must use SmartPlant P&ID.

   Notes:

   - You can manipulate the views and navigate through the listed changes by using the commands on the **Compare** dialog box toolbar. Each **Drawing** view also has its own shortcut menu, which includes manipulation commands that apply only to that view.

   - You can select an item in either **Drawing** view. The item is then located in the appropriate group in the **Change details** list. If you select an item in the **Change details** list, then you can use the **Find in Drawings** button on the toolbar to locate the item in one or both **Drawing** views.

- You can select an item in the **Drawing** view or in the **Change details** list. Properties for that item appear in the **Properties** window. Selecting multiple items is not possible on the **Compare** dialog box.

- The following differences are ignored: claim status, select list strings, linked or embedded objects, symbology, and inconsistency indicators.

- You can only compare a drawing against a version of itself; that is, you cannot compare one drawing to another drawing.

- You can also compare versions when you are checking in a drawing.

- If at any point you attempt to compare two versions that are actually identical to each other, the **Compare** dialog box does not open and a confirmation message alerts you as to why.

Compare Versions of Drawings in Separate Databases

1. In the **List** view, select the drawing.

2. Select **Show History**.

3. In the drawing list on the **Show History** dialog box, select the version in the active project that you want to compare against a version elsewhere.

4. Select **Compare With**.

5. On the **Compare With** dialog box, select the correct target project or the Plant database from the **Available Databases** list.

6. In the **History** list, choose the version of the drawing that you want to compare against your version and Select **OK**.

7. On the **Compare** dialog box, you can view the differences between the two versions, but you cannot make changes to the designs. To change the design, you must use SmartPlant P&ID.
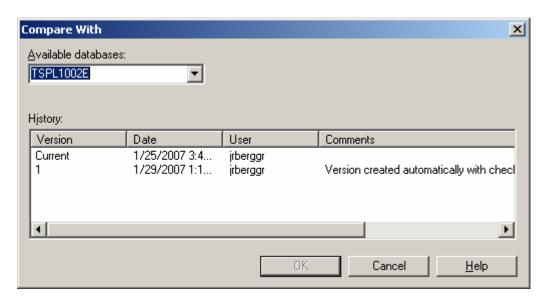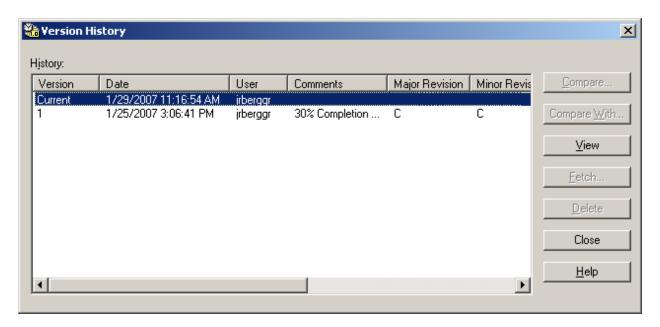
   📝 **Notes:**

   - You can manipulate the views and navigate through the listed changes by using the commands on the **Compare** dialog box toolbar. Each **Drawing** view also has its own shortcut menu, which includes manipulation commands that apply only to that view.

   - You can select an item in either **Drawing** view. The item is then located in the appropriate group in the **Change details** list. If you select an item in the **Change details** list, then you can use the **Find in Drawings** button on the toolbar to locate the item in one or both **Drawing** views.

   - You can select an item in the **Drawing** view or in the **Change details** list. Properties for that item appear in the **Properties** window. Selecting multiple items is not possible on the **Compare** dialog box.

   - The following differences are ignored: claim status, select list strings, linked or embedded objects, symbology, and inconsistency indicators.

   - You can only compare a drawing against a version of itself; that is, you cannot compare one drawing to another drawing.

   - You can also compare versions when you are checking in a drawing

   - If at any point you attempt to compare two versions that are actually identical to each other, the **Compare** dialog box does not open and a confirmation message alerts you as to why.

Show History Dialog Box

Lists all available versions of a drawing. You can compare two versions of the drawing, or view a version of the drawing without opening **SmartPlant P&ID**, or fetch a version from the list. You can open this dialog box by selecting **Versions** > **Show History** or by selecting the **History** button on the **Fetch** dialog box.

History - Lists all the versions of the drawing in the current plant or project.

Compare - Opens the **Compare** dialog box, allowing you to compare two versions in the **History** list. This button is not available unless two versions are selected in the list or if you open this dialog box by clicking **History** on the **Fetch** dialog box. Use the **Compare With** button to compare one version in the active project to a version in another project or the Plant.

Compare with - Opens the **Compare With** dialog box, allowing you to find a drawing version in a different project or in the Plant to compare to the drawing version you select in the **History** list. The **Compare With** button is not available if you open this dialog box by clicking **History** on the **Fetch** dialog box.

View - Opens the **View** dialog box, which displays a read-only view of the selected drawing version without opening SmartPlant P&ID. You can manipulate the view or select drawing items and review their properties.

Fetch - Opens the **Fetch-Option** dialog box. This button is available only if you have selected one, and only one, version in the **History** list and that drawing is not the current version. The **Fetch** button is not available if you open this dialog box by clicking **History** on the **Fetch** dialog box.
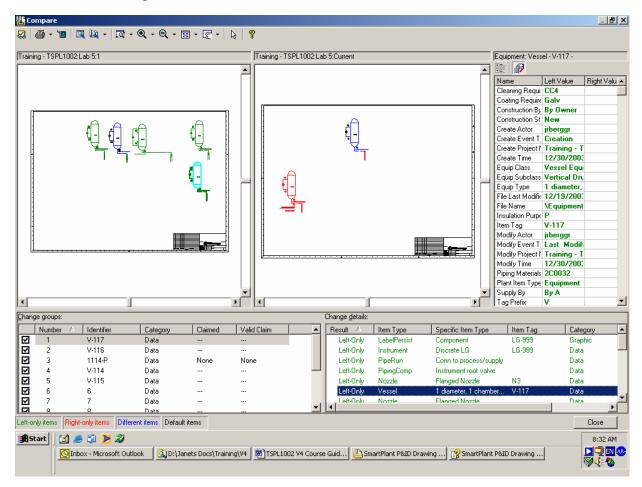
Delete - Removes the selected drawing version. You must have the appropriate permissions, assigned in SmartPlant Engineering Manager, to delete versions. You cannot delete the current version of a drawing by using this button. However, you can delete the current version of a drawing by using the **Delete** command on the **Edit** menu on the main menu bar.

Compare Dialog Box

Opens when you click **Compare** on the **Show History** dialog box, displaying two versions of the same drawing, and illuminating the differences between them. At the top of each **Drawing** view, the Plant or project, the name of the drawing, and the version is displayed explicitly. You can move the bars between the different views according to your needs. If you click on the divider between the left and right **Drawing** views, then the two views automatically same-size.

📝 Notes:

- You cannot do anything else in Drawing Manager while this dialog box is open.

Toolbar Commands - Apply to the **Drawing** views.

- **Compare Options** - 🔲 Opens the **Compare Options** dialog box, which allows you to customize the colors that the various comparison states are displayed in. That color-coding is then displayed in the status bar of the **Compare** dialog box as a static reminder.

- **Print** - 🖨 Prints the **Drawing** view or views.

- **Generate Report** - 📇 Opens Microsoft Excel and creates a report of the information contained in this comparing versions session. This report lists the same information that is displayed in the **Change Groups** and **Change Details** lists (change groups, item types, claim statuses, and so forth).

- **Find in List** - 🔍 Zooms to the **Change details** list entry that corresponds to an item you select in either **Drawing** view. Generally speaking, if you select a drawing item that exists in the list, then the list display automatically zooms to that entry.

- **Find in Drawings** - 🔍 Manipulates the **Drawing** views so that the selected element is listed in the **Change details** and the **Change groups** list is centered in the appropriate **Drawing** view.

- **Zoom Area** - 🔍 Enlarges the display of an area in one or both **Drawing** views by allowing you to draw a fence around that arbitrary area of the view.

- **Zoom In** - 🔍 Enlarges the display of items around a specified point in one or both **Drawing** views.

- **Zoom Out** - 🔍 Reduces the display of items around a specified point in one or both **Drawing** views.

- **Fit** - 🔲 Fits all visible items in one or both **Drawing** views.

- **Pan** - 🖐 Allows you to move the display in any direction from a specific point in one or both **Drawing** views in order to see other areas of the view by dragging the pointer across the display.

- **Select** - 🔺 Activates the select tool.

- **Help** - ❓ Opens Drawing Manager online Help.

Left Drawing View - Displays one version of your drawing. If you compare two versions from different projects, the version that belongs to the other project appears in the left-hand **Drawing** view. If you compare two versions from your active project, then the older version appears in the left-hand **Drawing** view.

Right Drawing View - Displays the other version of your drawing. If you compare two versions from different projects, the version that belongs to your active Drawing Manager project appears in the right-hand **Drawing** view. If you compare two versions from your active project, then the latest version appears in the right-hand **Drawing** view.

Properties Window - Displays two columns of properties for an item selected in a **Drawing** view or in the **Change details** list. The left-hand and right-hand column corresponds to the left-hand and right-hand **Drawing** views. If a deleted item is selected (the item exists in left-hand view, but not the right-hand), the properties for that item are listed in the left-hand column and the right-hand column is empty. If a modified item is selected, values from both versions show in their respective columns in the **Properties** window. If a new item is selected, that is, the item exists in right-hand view, but not the left-hand, the properties for that item are listed in the right-hand column and the left-hand column is empty.

Properties Commands - Allows you to customize the properties that are displayed in the **Properties** window.

- **Alphabetical** - Lists properties in alphabetical order. This button acts as a toggle and is available when properties are displayed categorically.

- **Categorized** - Displays properties grouped by specific categories. Categories are defined and properties are assigned to those categories in Data Dictionary Manager. This button acts as a toggle and is available when properties are displayed alphabetically.

Show Modified - Toggles the display of only those properties that are different between the properties that belong to a selected drawing item. This button applies only to modified items that exist in both versions; for added and deleted items, all properties are listed.

Change groups - Lists logical groupings of differences between the two versions. Each group contains the smallest number of items possible while preserving logic of the group. You can sort this list by clicking on any one of the following column headings.

- **Number** - Displays an arbitrary number that is assigned to a logical change group when this dialog box is opened. The number has no intrinsic meaning and may apply to a different group the next time you open this dialog box.

- **Identifier** - Lists item tags for the principal member of the change group, if an item tag is assigned to that object. For instance, if a change group centers on data differences for a vessel and its nozzles, then the item tag for the vessel is displayed in this column.

- **Category** - Displays the highest priority of categories that appear in the **Change details** list for this group. The possible categories are connectivity, data, and graphic, listed in highest to lowest priority.

- **Claim** - Displays an overview of the claim status of the individual items in the group. Possible values are all, some, or none.

Change details - Lists all the individual items that belong to the group that you select from the **Change groups** list. You can sort this list by clicking on any one of the following column headings.

- **Result** - Displays one of three possible values: Left-Only, Right-Only, and Different. Left-Only denotes an item that exists in the left-hand version only, implying that the item is deleted from the right-hand version. Right-Only denotes an item that exists in the right-hand version only, implying that the item was added to the right-hand version. Different denotes a difference between the properties, graphics, or connectivity of an item that exists in both versions.

- **Item Type** - Displays the item type of the individual item in question.

- **Item Tag** - Displays the item tag of the individual item in question if a tag has been assigned to the item.

- **Category** - Displays the highest priority category of change that applies. The possible categories are connectivity, data, and graphic, listed in highest to lowest priority. That is to say, that an item that is moved graphically (graphic category) but also includes a property change (data category) lists "data" in its category column.

- **Claimed** - Displays the claim status of the object: valid claim, invalid claim, and not claimed.

Status bar - Displays the currently defined colors for illustrating comparison status. You can change the color scheme by clicking the **Compare Options Error! Objects cannot be created from editing field codes.**button on the toolbar and defining options on the **Compare Options** dialog box.

Compare Options Dialog Box

The **Compare Options Dialog Box** opens when you click **Compare Options** on the **Compare** dialog box toolbar and allows you to customize the colors that the various comparison states are displayed in. The active color scheme is displayed in the **Compare** dialog box status bar.

Left-only items - Allows you to choose a color for displaying objects that exist only in the left-hand Drawing view. Dark green is the default color for this option.

Right-only items - Allows you to choose a color for displaying objects that exist only in the right-hand Drawing view. Red is the default color for this option.

Different items - Allows you to choose a color for displaying items that exist in both views but differ from each other for any number of reasons (for example, modified

properties, changed connectivity, and so forth). Blue is the default color for this option.
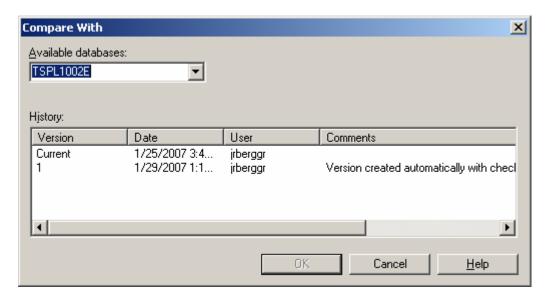

Default items - Allows you to choose a color for displaying drawing items that are identical in the two views. Black is the default color for this option.

Highlight items - Allows you to choose a color to denote that a drawing object is highlighted, for instance, when an item is within your locate zone.

Selected items - Allows you to choose a color to denote items that are selected in one or both of the **Drawing** views.

Compare With Dialog Box

Opens when you click **Compare With** on the **Show History** dialog box and allows you to select a drawing version from a database other than the current active database to compare against the version you choose on the **Show History** dialog box in the active database.

Available Databases - Lists all the different databases that currently have a version of the drawing you chose on the **Show History** dialog box.

History - Lists all the versions of the chosen drawing in the database you selected in the **Available Databases** list.

Compare and Refresh Command

The **Tools > Compare and Refresh** allows you to refresh the active drawing with data from another version of the drawing. The differences display in what is known as a change group. If you accept the changes, the drawing refreshes to display the changes.

To refresh a project drawing, you use the drawing in the plant. Change groups are marked if it affects items you have claimed. Be sure to refresh all of the changes that do not involve claimed items because these are the changes that have been checked in by other projects. You want to include these changes in your drawing so they will not be lost when you perform a check in.

Compare With Dialog Box

Opens when you click **Tools > Compare and Refresh** allowing you to select a drawing version to compare against the version that you currently have open and active.

Available Databases - Lists all the different databases that currently have a version of the drawing you chose on the **Show History** dialog box.

History - Lists all the versions of the chosen drawing in the database you named in the **Available Databases** list.

Compare and Refresh Dialog Box

Compare Options - Displays the **Compare Options** dialog box that allows you to customize the display colors for items that differ between the two drawings being compared.

Print - Prints the entire **Compare and Refresh** dialog box contents. You can also specify to print either the right or left view by using the drop-down arrow to select either **Right View** or **Left View**.

Generate Report - Displays a report in Microsoft Excel. The report contains the details of the compared drawings.

Find in List - Highlights the line in the **Change groups** and **Change details** areas. You must first select an item in a drawing.

Find in Drawing - Zooms to the selected item. You must first select an item in either the **Change groups** or **Change details** area.

Zoom Area - Enlarges the selected area by allowing you to draw a fence around the area.

Zoom In - Enlarges the selected area where you click.

Zoom Out - Reduces the display of the selected area where you click.

Pan - Allows you to move the display in any direction by dragging the pointer across the view.

Select - Changes the pointer to an arrow allowing you to select an item.

Alphabetic **Categorized** - Displays the properties in either an alphabetic list or by specific category.

Show Modified - Displays modified properties.

Change groups - Area that lists the changed items in groups. A listed item contains all the items this change effects.

- **Checkbox** column - Displays colors in the drawings as defined using **Compare Options**. If not selected, the **Default items** color displays.

- **Number** - Displays the number assigned to each change group.

- **Action** - Defines which action is to be taken involving the selected change. Options include:

    - **No Action** - Validates Invalid Claims for the selected group (no drawing changes are applied).

- **Refresh** - Applies any drawing changes to the open drawing version; and **No Action** - performs no action on the selected drawing group.
- **Category** - Displays the category of the change. Options include:
  - **Data** - Indicates that a property value has changed (for example, a property value for a vessel).
  - **Graphic** - Indicates that a change has been made to an item in the drawing (for example, a vessel has been moved).
- **Claimed** - Indicates that the items in the group have been claimed.
- **Valid Claim** - Indicates that the claimed item is a valid claim.

Change details - Area that lists the details of the item in each change group.

- **Result** - Defines the results of the compare. The column indicates any differences in the two versions and which version contains the change.
- **Change** - Describes what action is required to make the drawing in the right frame match the drawing in the left frame. Actions include:
  - **Add** - Adds the listed item to the version on the right.
  - **Delete** - Removes the listed item from the version on the right.
  - **Modify** - Changes the listed item in the version on the right.
- **Item Type** - Describes the type of item.
- **Specific Item Type** - Displays the specific type of item.
- **Item Tag** – Displays the value of the **Item Tag** property.
- **Category** - Displays the category of the change.
  - **Data** - Indicates that a property value has changed (for example, a property value for a vessel).
  - **Graphic** - indicates that a change has been made to an item in the drawing (for example, a vessel has been moved)**.**
- **Claimed** - Indicates if an item is a valid or invalid claim.
- **Source Representation Location** - Displays the source item location if the item exists in either the **Drawing** or the **Plant Stockpile**.
- **Target Representation Location** - Displays if the target item location if the item exists in either the **Drawing** or the **Plant Stockpile**.

Compare and Refresh Dialog Box Properties Window Toolbar

The **Properties Window** in the **Compare and Refresh Dialog Box** allows you to customize the properties that are displayed in the **Properties** window of the **Compare** dialog box.

Alphabetical - Lists properties in alphabetical order.

Categorized - Displays properties grouped by specific categories. Categories are defined and properties are assigned to those categories in Data Dictionary Manager.

Show Modified - Toggles the display of only those properties that are different between the properties that belong to a selected drawing item. This button only applies to modified items that exist in both versions; for added and deleted items, all properties are listed.

Compare Options Dialog Box

The **Compare Options Dialog Box** opens when you click **Compare Options** on the **Compare** dialog box toolbar and allows you to customize the colors that the various comparison states are displayed in. The active color scheme is displayed in the **Compare** dialog box status bar.

Left-only - Allows you to choose a color for the display of objects that exist in the left-hand Drawing view only. Dark green is the default color for this option.

Right-only - Allows you to choose a color for the display of objects that exist in the right-hand Drawing view only. Red is the default color for this option.

Different items - Allows you to choose a color for the display of items that exist in both views but differ from each other for any number of reasons (for example, modified properties). Blue is the default color for this option.

Identical items - Allows you to choose a color for the display of drawing items that are identical in the two views. Black is the default color for this option.

Highlight items - Allows you to choose a color to denote that a drawing object is highlighted, for instance, when an item is within your *locate zone*.

Selected items - Allows you to choose a color to denote items that are selected in one or both of the Drawing views.

Compare and Refresh Drawing Versions

Select **Tools > Compare**.

On the **Compare With** dialog box, select a database using the **Available databases** drop-down list box.

In the **History** list box, select the drawing you want to compare your current drawing with.

Select **OK**.

On the **Compare and Refresh** dialog box, review the information in the **Change groups** and **Change details** areas. Your current drawing displays on the right side of the screen. The version you are comparing it to displays on the left.

In the **Change groups** area, click in the **Action** column.

Using the drop-down list, select **No Action**, **Refresh**, or **Validate**. **Validate** will display as an option only if you have an **Invalid Claim**.

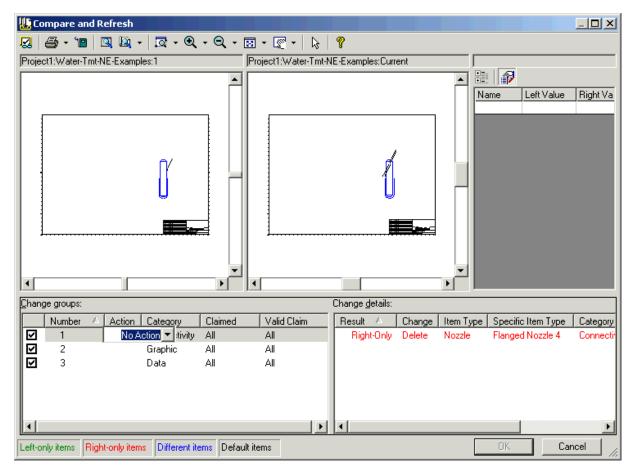Select **OK** to refresh the drawing and accept any changes or **Cancel** to dismiss the dialog box.

Compare and Refresh Examples

The following examples provide details for reconciling data and graphic differences between your current drawing and a previously created version of that drawing.

Data Example

In this example, a jacketed tower exists in the version to be selected for the compare. The current version contains the same jacketed tower but a change has been made to the **Cleaning Requirements** property. When the current version is compared to the selected version, a data change is found during the compare.

The current version appears on the right and the version it is compared to appears on the left. **A change group indicates** there is only one group and a data change was located. All items in the group are valid claims. **A change detail indicates** the compare located a different property value (**Cleaning Requirement**) in the current version. If you **Refresh** the version, the property value will be modified as shown in the **Change** column. Clicking in the **Action** column of the **Change groups** allows you to take no action or refresh (accept the new property value).

Graphic Example

In this example, a jacketed tower exists in the version to be selected for the compare. The current version contains the same jacketed tower but it has been moved to a new location. When the current version is compared to the selected version, a graphic change is found during the compare.

The current version appears on the right and the version it is compared to appears on the left. **A change group indicates** there is only one group and a graphic change was located. All items in the group are valid claims. **A change detail indicates** the compare located a graphic modification to a vessel (**Jacketed Vessel**) in the current version. If you **Refresh** the version, the graphic modification will be accepted as shown in the **Change** column. Clicking in the **Action** column of the **Change groups** allows you to take no action or refresh (accept the previous location of the vessel). The settings are not applied to a version until you click **OK**.