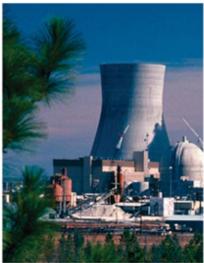
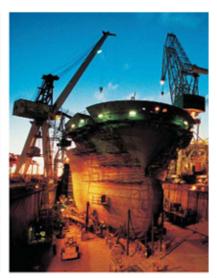
Drawings and Reports *User's Guide*

Process, Power & Marine









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Preface

This document is a user's guide for the SmartPlant[®] 3D Drawings and Reports task and provides command reference information and procedural instructions.

SmartPlant 3D Documentation Set

The SmartPlant[®] 3D documentation set is available as Adobe[®] PDF files. The content of the PDF files is the same content as online Help. To access these PDF documents in the software, click **Help > Printable Guides**.

The documentation set is divided into four categories:

- Administrative guides contain information about installing, configuring, customizing, and troubleshooting SmartPlant 3D.
- User's guides provide command reference and how-to information for working in each SmartPlant 3D task.
- Reference data guides define the reference data workbooks. Not all tasks have reference data.
- Third-party guides from other vendors for software that works with SmartPlant 3D.

Administrative Guides

Project Management User's Guide - Provides instructions for setting up the databases, creating permission groups, backing up and restoring project data, assigning access permissions to the model, managing interference detection, defining and managing locations for Global Workshare, controlling duplication and consolidation of plants, tools for synchronization, regeneration of report databases, and version upgrade.

SmartPlant 3D Database Integrity Guide - Provides information about the error messages in the database integrity reports, including meaning, cause, and possible corrective action.

SmartPlant 3D Installation Guide - Provides instructions on installing and configuring the software on both the client and server computers.

SmartPlant 3D/IntelliShip Programmer's Guide - Provides information about custom commands, naming rules, and symbol programming.

SmartPlant 3D Interference Checking Guide - Provides information on installing, configuring, and using the interference detection service.

SmartPlant 3D Plant Design System (PDS) Guide - Provides all information needed to use PDS with SmartPlant 3D. Topics include referencing active PDS projects in SmartPlant 3D, exporting PDS data and importing that data into SmartPlant 3D, converting PDS reference data to SmartPlant 3D reference data, and converting EDEN symbols to Visual Basic symbols.

SmartPlant 3D Release Bulletin - Provides what's new, hardware/software requirements, and support information for the current release.

SmartPlant 3D The Engineering Framework Reference Guide - Provides information about installing, configuring, and using The Engineering Framework with SmartPlant 3D.

SmartPlant 3D Troubleshooting Guide - Provides information on how to resolve errors that you may encounter in the software by documenting troubleshooting tips, error messages, and to do list messages.

User's Guides

Catalog User's Guide - Provides information about viewing, editing, and creating reference data and select lists (codelists).

Common User's Guide - Provides information about defining workspaces, navigating in the model, precision input, filtering, manipulating views, and running reports.

Drawings and Reports User's Guide - Provides information about creating drawing and report deliverables.

Electrical User's Guide - Provides information about routing electrical cable, cableway, cable tray, and conduit.

Equipment and Furnishings User's Guide - Provides information about placing equipment.

Grids User's Guide - Provides instructions for creating coordinate systems, elevation grid planes, vertical grid planes, radial cylinders, radial planes, grid arcs, and grid lines.

Hangers and Supports User's Guide - Provides instructions on placing piping, duct, cableway, and conduit supports in the model.

HVAC User's Guide - Provides instructions for routing HVAC duct.

Piping User's Guide - Provides instructions for routing pipe and placing valves, taps, and pipe joints.

Space Management User's Guide - Provides instructions for placing volumes (such as drawing volumes, obstruction zones) in the model.

Structural Analysis User's Guide - Provides instructions for defining loads, load cases, load combinations, and the importing and exporting of analytical data.

Structure User's Guide - Provides instructions for placing structural members such as: beams, columns, braces, slabs, openings, stairs, ladders, equipment foundations, and handrails.

Systems and Specifications User's Guide - Provides instructions for creating systems and their hierarchies and selecting which specifications are available for each system type.

SmartPlant 2D Symbols User's Guide - Provides instructions for creating cross section symbols.

Reference Data Guides

Drawings and Reports Reference Data Guide - Provides information about reports reference data.

Electrical Reference Data Guide - Provides information about electrical cable, cableway, cable tray, and conduit reference data.

Equipment and Furnishings Reference Data Guide - Provides information about equipment reference data and name rules.

Hangers and Supports Reference Data Guide - Provides information about hangers and supports reference data.

HVAC Reference Data Guide - Provides information about HVAC reference data.

Piping Reference Data Guide - Provides information about piping reference data including piping specifications, piping specification rules, piping parts, piping symbols, and name rules.

SmartPlant 3D Reference Data Guide - Provides instructions about the Bulkload utility, codelists, and the reference data common to several disciplines.

SmartPlant 3D Symbols Reference Data Guide - Provides information about the Visual Basic Part Definition Wizard and the three-dimensional symbols used in all tasks.

Space Management Reference Data Guide - Provides information about space management reference data.

Structure Reference Data Guide - Provides information about structural reference data and name rules.

Third-Party Guides

AText Reference Guide - Provides information about alternative text for isometric drawings. This guide is from Alias, the makers of ISOGEN[®].

Option Switches Reference Guide - Provides information about the ISOGEN option switches for isometric drawings. This guide is from Alias, the makers of ISOGEN.

Symbol Keys Reference Guide - Provides information about the symbol keys for isometric drawings. This guide is from Alias, the makers of ISOGEN.

Documentation Comments

Send documentation comments or suggestions to PPMdoc@intergraph.com.

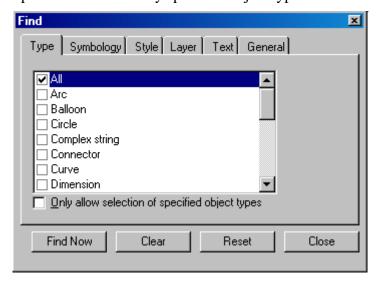
What's New in Drawings and Reports

The following changes have been made to the Drawing and Reports task.

Version 6.1

- You can create ship drawings based on orientations derived from userdefined Local Coordinate Systems. This is for Manufacturing Pin Jig drawings and Shell Profile Plan drawings.
- The new **Generic Module Folder** component documentation was missing from Version 6.0. The **Generic Module Folder** component provides a way to use custom VB modules within the Drawings and Reports task. For more information, see *Generic Module Folder: An Overview*, page 247.
- For Isometric drawings, the PreserveUserAnnotation option is no longer available. The software now preserves your manual markups on the Default layer and any layer prefixed with User (for example, a layer named UserAnnotationLayer) when you update drawings. This behavior is true for both Isometric and Orthographic drawings. For more information, see Working with Layers: An Overview, page 223 for Isometric drawings.

• In the **2D Drawing Editor**, the **Find** command is now delivered on the **Edit** menu. The **Find** command also has a new option to restrict the operation to select only specified object types.



- Other improvements in the **2D Drawing Editor** are the ability to use the **Shift** key to move elements in horizontal and vertical directions and the ability to use macros to customize the drawing templates.
- Drawings support graphical comparison so that you can update drawing document revision descriptions accurately when registered with The Engineering Framework.

Drawings and Reports: An Overview

This task creates orthographic drawings, isometric drawings, and reports from the model. This task provides an update feature to increase productivity for your company. When the 3D model changes, you can update the drawings and reports.

The Drawings and Reports task is also responsible for publishing. When registered with The Engineering Framework (TEF), you can publish volume and snapshot drawings, orthographic drawings, isometric drawings, and reports. You can also publish 3D model data using the 3D Model Data component. For more information, see Publishing Documents: An Overview, page 373.

The **Management Console** organizes the different document types into a customizable hierarchy. Using the component functionality of the console, you can create, edit, update, print, save, and publish the deliverables.

Before you can create components for drawings and reports, your administrator must organize the **Management Console** hierarchy with folders for each component type. Then, the administrator must complete several setup steps, including setting up drawing and report templates, creating view styles, creating appropriate filters, and specifying isometric drawing options. Default templates and view styles are delivered with the software, and you can customize them to suit your needs.

It is possible to customize templates and view styles before any objects exist in the model. However, to create drawings and reports, objects must exist in your model. For example, if you want to generate isometric drawings, you must have piping in your model.

Volume and Snapshot Drawings

Volume and Snapshot Drawings are useful for creating general arrangement or construction drawings of areas within the model. In the Volume and Snapshot Drawing workflows, you or your administrator must create or edit border templates. You can place drawing property labels in the title block of the template to fit your company or project. You also must configure the view styles, which are sets of rules that determine how the graphics in the three-dimensional model are represented on the drawings. View styles use filters. You can create a folder of drawing filters, with new, existing, and future filters for each discipline.

For Volume Drawings, you place drawing volumes in the Space Management task. For Snapshot Drawings, you take snapshots of the model using the **Drawing View** command in any of the 3D design tasks.

You can publish both Volume and Snapshot Drawings when they are up-to-date. For Volume and Snapshot Drawings, a viewable graphic file is created; no physical data is published.

For more information on snapshot drawings, see *Snapshot Drawings: An Overview*, page 153. For more information on volume drawings, see *Volume Drawings: An Overview*, page 161.

Orthographic Drawings by Query

The Orthographic Drawing by Query component, in conjunction with the Drawings by Query Manager component, creates drawings in mass by specifying a filter-based query to collect objects for drawings. This drawing type is appropriate for creating detail drawings of particular objects within the model. They are especially useful when creating drawings that use the same style or format for large numbers of similar objects, such as hangers or supports.

Just like Snapshot and Volume Drawings, you can print, update, save into MicroStation or AutoCAD formats, or publish Orthographic Drawings by Query to The Engineering Framework (TEF). When you publish Orthographic Drawings, a viewable graphic file is created; no physical data is published.

For more information on orthographic drawings by query, see *Orthographic Drawings by Query: An Overview*, page 181.

Piping Isometric Drawings by Query

Like Orthographic Drawings by Query, you create Piping Isometric Drawings by Query by specifying a filter-based query to collect the objects. The workflow requires that you create or edit border templates to fit your company or project. You or your administrator also must set the isometric options for each of the isometric styles that you need in your project.

When you publish Piping Isometric Drawings by Query, they are published as viewable graphics; no physical data is published.

For more information on piping isometric drawings, see *Piping Isometric Drawings* by *Query: An Overview*, page 193.

Reports

In the Spreadsheet Reports workflow, you create report templates, which control the content and format of reports. The default file format of reports in the software is Microsoft Excel[®] format. The Report Editor provides the ability to configure your reports to use queries and special formatting.

You can publish Spreadsheet Reports just like drawings. However, the Spreadsheet Reports are published as Excel spreadsheets; no physical data is published.

For more information on reports, see *Spreadsheet Reports: An Overview*, page 251.

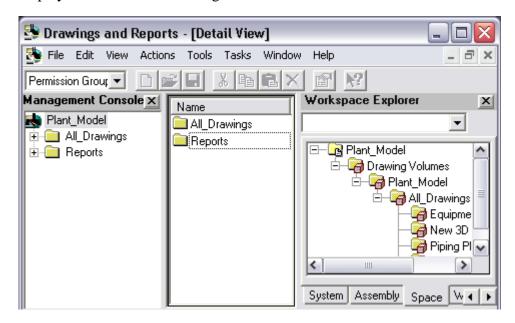
Note

 To ensure that Microsoft Excel spreadsheets embedded in a drawing are displayed properly, users of Office 2000 are required to install Service Pack 3 for that product. For users of Office XP, in Microsoft Excel under Tools > Macro > Security > Trusted Publishers tab, check the Trust Access to Visual Basic Project option. For more information about Office 2000, Office XP, and service packs, refer to the Microsoft® web site (http://www.microsoft.com/).



Understanding the Windows: An Overview

This task includes different windows or views within its interface. You can toggle the display of these windows using commands on the **View** menu.



The **Management Console** contains a hierarchy of folders and components that you create. If you right-click an item in the **Management Console**, the available menu commands vary, depending on the item and your permissions. For more information on managing folders and components in the **Management Console**, see *Managing Documents: An Overview*, page 29.

The **Detail View** shows the children of the selected item in the **Management Console**. You can sort the list in the **Detail View** just like sorting files in Windows Explorer. You can select multiple components or documents by pressing **Ctrl** or **Shift** while selecting. To specify the columns in the **Detail View**, right-click a column heading and click **More**. The **Detail View** is overlaid by other windows depending on the current operation. For example, when reviewing drawings using the **Open** command, a 2D viewer appears. When you edit report templates, a tabular editor appears. For more information on setting the appearance of the Detail View, see *Detail View Command (View Menu)*, page 22. For information on using commands associated with templates, see *Template Commands: An Overview*, page 122.

The **Workspace Explorer** is the tabbed view of systems, assemblies, spaces, and Work Breakdown Structure (WBS) items in the software. For more information about the **Workspace Explorer**, see the *Common User's Guide* available from the **Help** > **Printable Guides** command in the software.

Note

• Another window you use while working in this task is the **2D Drawing Editor**, which appears as a separate application window. It allows you to edit border templates, drawing templates, and backing sheets for all types of drawings.

- Viewing Icons: An Overview, page 19
- Viewing the Menus and Toolbars: An Overview, page 17

Viewing the Menus and Toolbars: An **Overview**

In this task, the menus and toolbars are dynamic. Commands change according to the active window and workflow. For example, if the Management Console and Detail View are active, the menus and toolbars are appropriate for actions such as adding components to the hierarchy or viewing properties of documents.

When you edit a template or open a drawing, the **2D Drawing Editor** menus and toolbars are available. When you edit report templates, the report menus and commands are available.

In addition, the shortcut menu that appears when you right-click an item in the Management Console differs according to the type of item. For example, some of the commands on the shortcut menu for a piping isometric drawing are different from the commands on the shortcut menu for a snapshot drawing component.

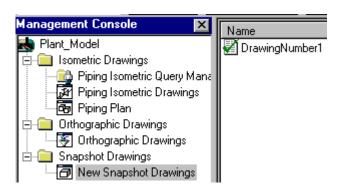
Also, keep in mind that the main menu bar available in SmartPlant 3D varies by task. Some commands available in other tasks may not be available in this one.

- Specify Columns in the Detail View, page 24
- Understanding the Windows: An Overview, page 15

Viewing the Menus and Toolbars:	An Overview	

Viewing Icons: An Overview

The Management Console and Detail View display different icons to show the type and status of documents.



Drawing Type Icons

- **Solution** Root of the hierarchy
- i Folder
- 👊 Query Manager
- de Generic Module Folder
- Volume Drawing type
- Snapshot Drawing type
- Orthographic Drawing type
- Piping Isometric Drawing type
- Spreadsheet Report
- **-** 3D Model Data type
- ☑- MicroStation 3D DGN drawing type

Document Status Icons

These icons appear superimposed on the document icon and indicate document status.

- Submitted or scheduled for batch processing
- **a** Updating or publishing to The Engineering Framework
- X Out-of-date

✓ - Up-to-date

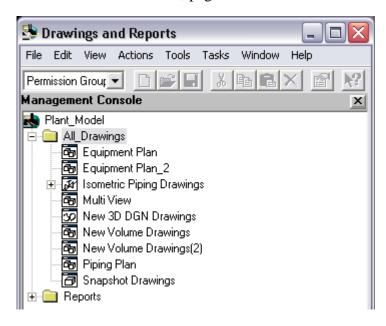
♣ - Error status

- Understanding Components: An Overview, page 33
- Understanding the Windows: An Overview, page 15
- Viewing the Menus and Toolbars: An Overview, page 17

Management Console Command (View Menu)

Toggles the display of the **Management Console**. By default, the **Management Console** is visible when you enter this task.

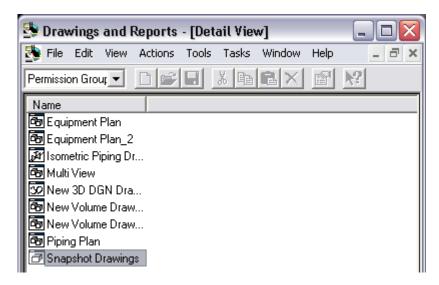
The Management Console contains a hierarchy of folders and components that you create. If you right-click an item in the **Management Console**, the available menu commands vary, depending on the item and your permissions. For more information on managing folders and components in the **Management Console**, see *Managing* Documents: An Overview, page 29.



- Detail View Command (View Menu), page 22
- Understanding the Windows: An Overview, page 15
- Workspace Explorer Command (View Menu), page 26

Detail View Command (View Menu)

Turns the display of the **Detail View** on and off. This command is located on the **View** menu. When checked, the **Detail View** is visible in the application window. When you right-click folders or documents in the **Detail View**, shortcut menus display. The items on the shortcut menu vary depending on the selected item. For more information on the commands, see *Managing Documents: An Overview*, page 29.



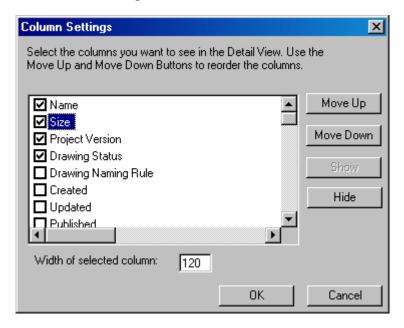
To modify the headings in the **Detail View**, right-click the column-heading area. Select **More** on the shortcut menu to display the **Column Settings** dialog box.

For more information, see *Specify Columns in the Detail View*, page 24.

- Management Console Command (View Menu), page 21
- Understanding the Windows: An Overview, page 15
- Workspace Explorer Command (View Menu), page 26

Column Settings Dialog Box

Specifies the columns you want to see in the **Detail View**. You also can specify the order and width of the columns. You access this dialog box when you right-click in the column heading area of the Detail View and select **More** on the shortcut menu.



Move Up - Moves the selected column up one position. The column appears one position to the left in the **Detail View**.

Move Down - Moves the selected column down one position. The column appears one position to the right in the **Detail View**.

Show - Displays the column in the **Detail View**.

Hide - Hides the column in the **Detail View**.



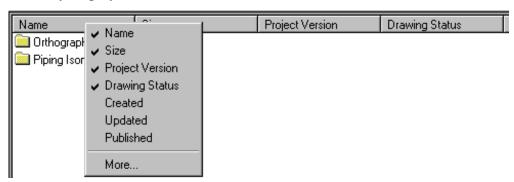
In addition to the **Show** and **Hide** commands, you can use the checkboxes beside the column names to add and remove them from the **Detail View**. Checked indicates that the column appears in the **Detail View**.

Width of selected column - Specifies the width of the column in pixels. You can specify a different column width for each column.

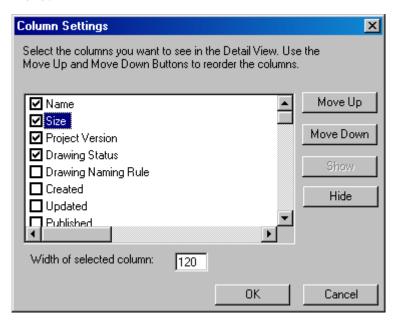
- Detail View Command (View Menu), page 22
- Specify Columns in the Detail View, page 24
- Understanding the Windows: An Overview, page 15
- Viewing the Menus and Toolbars: An Overview, page 17

Specify Columns in the Detail View

1. Right-click a column heading in the **Detail View**. The shortcut menu shows the currently displayed columns with a checkmark ✓.



- 2. Add and remove columns automatically by checking and unchecking them on the shortcut menu.
- 3. To modify the appearance and order of the columns, click **More** on the shortcut menu.



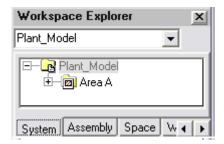
- 4. On the **Column Settings** dialog box, select the columns you want to include in the **Detail View**. Clear, or uncheck, the ones you do not want to include. You can also use the **Show** and **Hide** buttons to add and remove columns.
- 5. To change the order of the columns, click **Move Up** and **Move Down** on the **Column Settings** dialog box.

6. Specify the width of a column by selecting it and typing an integer in the **Width** of selected column box. You can also resize columns by dragging the edge of the column in the **Detail View**.

- Column Settings Dialog Box, page 23
- Detail View Command (View Menu), page 22
- Understanding the Windows: An Overview, page 15
- Viewing the Menus and Toolbars: An Overview, page 17

Workspace Explorer Command (View Menu)

Toggles the display of the **Workspace Explorer** as a viewer only in the Drawings and Reports task. By default, the **Workspace Explorer** is visible when you enter this task. The **Workspace Explorer** displays the contents of the workspace in a classification hierarchy that reflects the various relationships defined for the design objects. The content represents the current objects loaded from the database into the active workspace.



Viewing in 3D

When using the **View in 3D** command, the **Workspace Explorer** works as it does in other 3D tasks, allowing you to identify or select objects or sets of objects by name. Icons appear at the left of the window objects to indicate the type of the object. For example, a file folder icon represents the plant, an equipment icon represents equipment, an I-beam icon represents a structural system, and so forth. Point to objects in the **Workspace Explorer** to highlight them, and click objects to select them.

If you pause the pointer over an object in the **Workspace Explorer**, the object is highlighted with the highlight color in the graphic view and in the **Workspace Explorer**. If you pause the pointer over an object in a graphic view or in the **To Do List** dialog box, the object is highlighted with the highlight color in the **Workspace Explorer**. However, if you select certain objects such as features in a graphic view, they do not highlight in the **Workspace Explorer** (because features do not appear in the **Workspace Explorer**). Foundation ports also do not appear in the **Workspace Explorer**, but nozzles do.

If you select an object in the **Workspace Explorer**, it appears highlighted with the select color in the graphic view. If you select an object in a graphic view, it is highlighted with a gray background in the **Workspace Explorer**. The **Workspace Explorer** automatically scrolls when objects are highlighted if the object is outside the **Workspace Explorer** window. Automatic scroll is not available if the object is already in the **Workspace Explorer** window.

If you select a set of objects, but the set contains objects you do not want, press CTRL and click to cancel the selection of unwanted objects. To select multiple objects, press CTRL while selecting to add individual object to the set or press SHIFT

to add all objects between the selected object and the last highlighted object above it. The **Select Nested** command on the right-click menu of any **Workspace Explorer** object allows you to select the parent and all nested children of the parent.

In the **Workspace Explorer**, you can select or type a name in the box at the top of the view. The list displays the last 15 objects you selected that satisfy the current select filter. Also, you can use the wildcard * (asterisk character) when typing the name in the box at the top of the view.

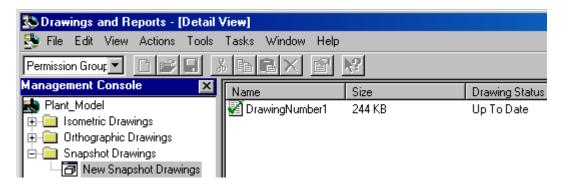
- Detail View Command (View Menu), page 22
- Management Console Command (View Menu), page 21
- Understanding the Windows: An Overview, page 15
- View in 3D Command, page 104



Managing Documents: An Overview

You can organize and manage your drawings and reports using the **Management** Console and Detail View.

The Management Console and Detail View work together similar to Windows Explorer. You can select multiple items for processing, sort items, and add columns in the **Detail View**.



When you right-click nodes in the **Management Console** and nodes or documents in the **Detail View**, shortcut menus display. The items on the shortcut menu vary depending on the selected item. The commands available include, but are not limited to:

New Command, page 34 Save Package Command, page 38 Copy Command, page 41 Paste Command, page 43 Edit Command, page 46 Open Command, page 47 Delete Command, page 48 Rename Command, page 49 Print Command, page 50 View Log Command, page 52



By default, the **Management Console** and **Detail View** appear when you enter the task. You can turn the views on or off on the **View** menu.

- Add a Component, page 36
- *Understanding Components: An Overview*, page 33



Using Permissions: An Overview

Your site administrator sets permissions and creates permission groups in the Project Management task. These permissions are used in the different tasks in the software to control user access.

You can see your current permission group in the dropdown box in the upper lefthand corner of the window when in the Drawings and Reports task.



The permission group to which an item belongs can affect the actions allowed against that item. For example, the propagation of properties down the hierarchy, from parent to child, is interrupted when a node or document in a read-only permission group is encountered.

The following list shows the actions relating to drawings and reports that are affected by permission groups:

- Accessing shortcut menu commands in the **Management Console** and **Detail View**
- Creating items, such as drawings, drawing views, and drawing volumes
- Propagating properties down through the hierarchy
- Deleting items
- Updating items, such as re-extracting drawings

In addition, access to the Symbols share on the server computer affects actions such as creating and editing view styles and graphic rules.

- *Understanding the Windows: An Overview*, page 15
- Viewing the Menus and Toolbars: An Overview, page 17

Using Permissions: An Overview

Understanding Components: An Overview

Several specialized components are provided, and they access commands for configuring templates and generating drawings and reports. The various types of components can be divided into two groups: application components and folder components. Many of the application components correspond to specific types of drawings, such as volume drawings and snapshot drawings. The Spreadsheet Reports component provides access to report-related commands. You organize drawings and reports in folders. You can add folders to the root and to other folders. Each component has a different icon and right-click menu.

You can copy and paste components with some restrictions. For example, application components cannot contain folders or other application components. If you copy a folder, you can paste it under a folder but not under another component type. If you copy a Volume Drawing component, you can paste it under a folder. If you copy other types of components, you can paste them under folders but not under other types of components.

Your administrator can assign permissions to the different components using commands in the Project Management task. For example, the administrator can set permissions so that only the piping designers have write privileges on Piping Isometric Drawings. For more information, see the *Project Management User's* Guide.

There are several types of delivered components. Their names reflect the type of drawing or report they create. When you right-click the root or a folder, then select New, the Add Component dialog box appears. The dialog box includes a General Tab for general types of drawings or reports and an Imperial Drawing Types Tab that contains drawing packages based on a fully-defined template. For more information on the **Add Component** dialog box, see *Add Component Dialog Box*, page 34. For more information on the Imperial Drawing Types, see Imperial Drawing Types: An Overview, page 149.

Note

Windows® 2000 and XP use similar components in the Microsoft® Management Console (MMC). For more information, see the help for your operating system.

- Add a Component, page 36
- Managing Documents: An Overview, page 29
- New Command, page 34
- Viewing Icons: An Overview, page 19

New Command

Adds new components or packages to the **Management Console**. Select the model root or a folder to place the new components or packages. If you have previously saved a package, the package is available to add to the **Management Console**.

If you select the model root, the **New** command creates a new folder in which to place components and packages. You cannot create an application component directly under the model root.

Related Topics

• Understanding Components: An Overview, page 33

Add Component Dialog Box

Lists the available components, folders, and packages you can create at the selected level in the **Management Console** hierarchy. You access this dialog box when you select **New** on the shortcut menu for a folder or the model root. There are four tabs delivered by default. The **General** tab shows a list of delivered components. The **Imperial** tab shows delivered packages to assist in the production of piping isometric drawings and orthographic drawings, respectively. The **Isometric** (**By Query**) and **Orthographic** (**By Query**) tabs show delivered Drawings by Query packages to assist in the production of piping and isometric drawings and orthographic drawings, respectively. If you access this dialog box from the root node in the **Management Console**, the only tab available by default is the **General** tab so you can create a new folder.

You can add tabs to this dialog box when you save a package. For more information, see *Save Package Command*, page 38.

This dialog box also provides a brief description of each component or package. Select a component or package to view its description.

- Imperial Drawing Types: An Overview, page 149
- New Command, page 34
- Understanding Components: An Overview, page 33
- Viewing Icons: An Overview, page 19

General Tab (Add Component Dialog Box)

Lists the delivered components and folders you can create at the selected level in the Management Console hierarchy.

This tab also provides a brief description of each component. Select a component to view the description.

Root Node Selected

If you right-click the root node in the **Management Console** and select **New**, the General tab allows you to create a new folder.

Folder Selected

If you right-click a folder in the Management Console and select New, the General tab includes both folders and application components.

Related Topics

- Imperial Drawing Types: An Overview, page 149
- New Command, page 34
- *Understanding Components: An Overview*, page 33
- Viewing Icons: An Overview, page 19

Imperial Tab (Add Component Dialog Box)

Lists the Imperial drawing type packages delivered with the software.

This tab also provides a brief description of each package. Select a package to view the description.

For more information on imperial drawing types and the delivered packages, see Imperial Drawing Types: An Overview, page 149.

- New Command, page 34
- Understanding Components: An Overview, page 33
- Viewing Icons: An Overview, page 19

Isometric (By Query) Tab (Add Component Dialog Box)

Lists the Piping Isometric Drawing by Query packages delivered with the software.

This tab also provides a brief description of each package. Select a package to view the description.

Related Topics

- New Command, page 34
- Understanding Components: An Overview, page 33
- Viewing Icons: An Overview, page 19

Orthographic (By Query) Tab (Add Component Dialog Box)

Lists the Orthographic Drawing by Query packages delivered with the software.

This tab also provides a brief description of each package. Select a package to view the description.

Related Topics

- New Command, page 34
- Understanding Components: An Overview, page 33
- Viewing Icons: An Overview, page 19

Add a Component

1. Right-click an item in the **Management Console** or the **Detail View**.



- To add a folder, right-click the root or another folder.
- To add an application component, right-click a folder.
- 2. On the shortcut menu, select **New**.
- 3. Select a component or package on one of the tabs of the **Add Component** dialog box, and click **OK**.
- 4. To rename the component, right-click it, then select **Rename** on the shortcut menu, or press **F2** on the keyboard.

Notes

- Right-click a component to access the available commands for that component.
- You can place a folder, application component, or a package under a folder in the hierarchy. For example, you can add a Piping Isometric Drawings by Query component to a folder.

• You can save a package and have it listed on the **Add Component** dialog box. When adding the package to the hierarchy, the software adds the components in the package to the active permission group. For more information, see *Save a Package*, page 39.

- Managing Documents: An Overview, page 29
- New Command, page 34
- Understanding Components: An Overview, page 33

Save Package Command

Saves the console hierarchy from the selected component down. The package saves the setup information and any template definitions that may exist on nodes within the selected hierarchy. You can access this command by right-clicking a folder or application component in the **Management Console**. You must have at least write permissions on the component to access the **Save Package** command.

When a package is placed back into the **Management Console**, it will recreate the hierarchy that was saved with the package.

Packages are also used in the setup of a Drawings by Query Manager component for the creation of orthographic and piping isometric drawings.

Notes

- Output documents are not saved in a package.
- If the topmost component saved in the package is a folder, then the package can be placed under the model root or a folder. If the topmost component saved in the package is an application component, then the package can only be placed under a folder.
- When you place a package, the software adds all the components to the active permission group.
- To save drawings or reports externally, see *Save As Command*, page 105.

Related Topics

- Managing Documents: An Overview, page 29
- Save a Package, page 39

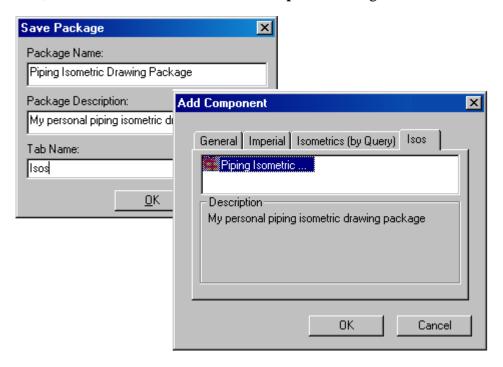
Save Package Dialog Box

Saves all or a portion of the console hierarchy as a package for import later. You can open this dialog box by right-clicking a folder or component in the **Management Console** and selecting the **Save Package** command.

Package Name - Specifies a name for the package.

Package Description - Describes the package.

Tab Name - Specifies the tab of the **Add Component** dialog box on which the package appears. You can pick an existing name or type a new tab name in this field. The next time you access the **Add Component** dialog box from an existing folder, the software adds the new tab and lists the new package on the tab. For more information, see *Add Component Dialog Box*, page 34. For example, if you saved a package called Piping Isometric Drawing Package and added it to a new tab called Isos, an **Isos** tab is added to the **Add Component** dialog box:



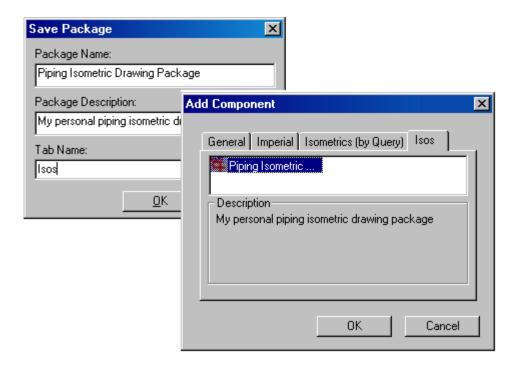
Related Topics

- Managing Documents: An Overview, page 29
- Save a Package, page 39

Save a Package

- 1. Right-click a component in the **Management Console** or the **Detail View**.
- 2. Select **Save Package** on the shortcut menu.
- 3. On the **Save Package** dialog box, specify a name, description, and tab name. For example, you could save a Piping Isometric Drawing Package to a new tab called **Iso**.
- 4. Click **OK** to save the package.

The package is added to the **Add Component** dialog box on the specified tab. If a new tab name was specified, a new tab is added to the dialog box.



You can add the new package to the hierarchy in the **Management Console** by using the **New** command. Select the package on the **Add Component** dialog box. When you add a package, the software adds all the components to the active permission group.

- Managing Documents: An Overview, page 29
- Orthographic Drawings by Query Common Tasks, page 183
- Piping Isometric Drawings by Query Common Tasks, page 195
- Save Package Command, page 38
- Understanding Components: An Overview, page 33

Copy Command

Copies the console hierarch from the selected component down. The copy command does not copy documents. After you copy an item, you can paste it at another location in the hierarchy. If you copy a component, the software enforces the following rules regarding pasting:

- If you copy a folder, the **Paste** command will only work if a folder is selected. An error message will display if you use the **Paste** command on application components in this situation. This is because application components cannot have a folder beneath them in the hierarchy.
- If you copy an application component and the **Paste** command is selected on a folder, the copied application component will be pasted under the folder.



- If the application component you copy is a volume drawings or MicroStation 3D DGN component, a **Paste Special** dialog box displays if either component has a template or drawing volume associated with it. The dialog box gives the option to paste the component with or without the template or volume of that component.
- If you copy a volume drawings component with a template or a drawing volume associated with it, the software displays a **Paste Special** dialog box if you paste onto another volume drawings component. This dialog box give you the option to paste (1) only the template, (2) only the volume, or (3) both the template and volume. The volumes associated with the current volume drawings component being pasted are copied and associated with the new volume drawings component. The **Paste** command is not available under any other type of component.

Note

If you copy a MicroStation 3D DGN component with a drawing volume associated with it, the software also displays the Paste Special dialog box if you paste onto another MicroStation 3D DGN component.

Related Topics

Copy an Item, page 42

Copy an Item

- 1. Select an item in the **Management Console** or **Detail View**.
- 2. Right-click the item, then select **Copy** on the shortcut menu.

Notes

- You can paste the copied item at another location in the **Management Console** hierarchy. For more information, see *Paste an Item*, page 44.
- If you copy a component, the software enforces rules regarding pasting. For more information, see *Copy Command*, page 41.

- Managing Documents: An Overview, page 29
- Understanding Components: An Overview, page 33

Paste Command

Either inserts the last-copied contents into the hierarchy, modifies the template information, or creates drawing volumes, depending on the component you have selected. You must copy an item using the **Copy** command before you can paste it.

The software enforces the following rules regarding pasting:

- If you copy a folder, the **Paste** command will only work if a folder is selected. An error message is displayed if you use the **Paste** command in this situation on application components. This is because application components cannot have a folder beneath them in the hierarchy.
- If you copy an application component and the **Paste** command is selected on a folder, the copied application component will be pasted under the folder.



- If the application component you copy is a volume drawings or MicroStation 3D DGN component, a **Paste Special** dialog box displays if either component has a template or drawing volume associated with it. The dialog box gives the option to paste the component with or without the template or volume of that component.
- If you copy a volume drawings component with a template or a drawing volume associated with it, the software displaces the **Paste Special** dialog box if you paste onto another volume drawings component. This dialog box gives you the option to paste (1) only the template, or (2) only the volume, or (3) both the template and volume.

Note

 If you copy a MicroStation 3D DGN component with a drawing volume associated with it, the software also displays the **Paste Special** dialog box if you paste onto another MicroStation 3D DGN component.

Related Topics

• Paste an Item, page 44

Paste Special Dialog Box

Specifies the items to paste if you copied a volume drawings or MicroStation 3D DGN component that has a template or drawing volume defined. The options provided to you depend on the component selected.

If you are pasting a hierarchy containing one or more volume drawings or MicroStation 3D DGN components into a folder, you can select one of the following options:

Copy Nodes(s), Template(s), and Volume(s) - Inserts the new components, including their respective template and drawing volumes, under the selected folder.

Copy Node(s) and Templates(s) - Inserts the new components, including their respective templates, under the selected folder.

Copy Node(s) Only - Inserts the new components under the selected folder.

If you are pasting a volume drawings component onto another volume drawings component or a MicroStation 3D DGN component onto another MicroStation 3D DGN component, you can select one of the following options:

Copy Template(s), and Volume(s) - Copies the template settings and drawing volumes to the selected component.

Copy Template(s) only - Copies only the template settings to the selected component.

Copy Volumes(s) only - Copies only the drawing volumes to the selected component.

Related Topics

- Paste an Item, page 44
- Paste Command, page 43

Paste an Item

Before pasting an item, you must copy the item using the **Copy** command. For more information, see *Copy an Item*, page 42.

- 1. Select a location in the **Management Console** or **Detail View**.
- 2. Right-click the location, and click **Paste** on the shortcut menu. The software pastes the item under the selected location.

Note

The software enforces rules regarding pasting. For more information, see Paste Command, page 43.

- Managing Documents: An Overview, page 29
- Understanding Components: An Overview, page 33

Edit Command

Activates the selected drawing for editing. This command is available on the right-click menu for a drawing in the **Detail View**. The drawing opens in the 2D Drawing Editor with the addition of the **Manually Place Labels** command.



If you use other native 2D Drawing Editor commands (such as Place Line or Place Dimension) to add manual markups to the template, put them on the **Default** or a layer with "User" as the prefix (for example, a layer named **UserAnnotationLayer**) to preserve the changes when you update drawings.



Your access permissions, defined in the Project Management task, affect whether or not you can edit documents.

- Edit Template Command (Volume Drawings Component), page 169
- Snapshot Drawings: An Overview, page 153
- Volume Drawings: An Overview, page 161

Open Command

Activates the selected document for viewing within this task. This command is available on the shortcut menu for all document types except 3D Model Data. You also can open the document by double-clicking it.

Note

• Opening a MicroStation 3D DGN document requires that the MicroStation application be loaded on the workstation.

Related Topics

- Edit Template Command (Volume Drawings Component), page 169
- Open a Document, page 47
- Snapshot Drawings: An Overview, page 153
- Volume Drawings: An Overview, page 161

Open a Document

- 1. In the **Detail View**, double-click a document. You can also right-click the document, then select **Open**.
- 2. Close a document by clicking **File > Exit**.

Note

• To edit the document, right-click the document, the select **Edit**. If the document is a drawing, you can annotate it.

- Managing Documents: An Overview, page 29
- *Open Command*, page 47
- Understanding Components: An Overview, page 33

Delete Command

Removes an item and its sub-items from the hierarchy and the database. You access this command on the right-click menu for any node or document in the hierarchy. **Undo** is not available for this action. A confirmation message appears.

You can delete an individual document by right-clicking the document and selecting **Delete** on the shortcut menu.

When you delete a drawing, its associated template and its component remain unchanged. Any associated physical volumes are deleted.

Note

• You can select multiple components or documents in the **Detail View** and use the **Delete** command from the right-click menu to remove those items.

- *Delete an Item*, page 361
- Understanding Components: An Overview, page 33

Rename Command

Activates the name of an item in the hierarchy. You can type a different name. The shortcut key for this command is **F2**. The following restrictions on naming exists:

- Names of items, including components and documents, cannot exceed 40 characters.
- You cannot have duplicate names at the same level in the tree.

Related Topics

- Rename an Item, page 49
- Understanding Components: An Overview, page 33

Rename an Item

- 1. Select an item in the **Management Console** or **Detail View**.
- 2. Right-click the item, then select **Rename** on the shortcut menu or press **F2** on the keyboard.
- 3. Type a new name for the item.

Notes

- Names of items, including components and documents, cannot exceed 40 characters.
- You cannot have duplicate names at the same level in the tree.

- Managing Documents: An Overview, page 29
- Rename Command, page 49
- Understanding Components: An Overview, page 33

Print Command

Sends a print request for the selected documents to the default printer. This command is not available until you have created and updated documents. For more information, see *Updating Documents: An Overview*, page 337.

Related Topics

- Print a Document, page 50
- Understanding Components: An Overview, page 33

Select Printer Command (File Menu)

Specifies a printer for documents. The command lists all printers available to your computer.

Related Topics

- Print a Document, page 50
- Understanding Components: An Overview, page 33

Select Printer Dialog Box

Specifies a printer for documents.

Name - Specifies a printer name.

Status - Displays the current status of the specified printer.

Type - Displays the type of printer.

Where - Displays the port or location the printer uses.

Related Topics

- Print a Document, page 50
- Understanding Components: An Overview, page 33

Print a Document

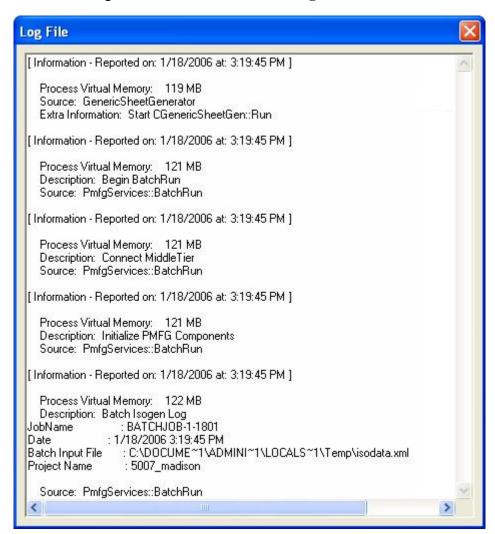
- 1. To specify a printer, select **File > Select Printer.**
- 2. Select a folder, application component, or the root node in the Management Console to print all of the drawing and report documents beneath the selected level. You can also select a single document or multi-select documents in the Detail View. You can select multiple documents to print by pressing Ctrl or Shift and then clicking each document in the Detail View.

3. Right-click and select **Print** on the shortcut menu.

- Managing Documents: An Overview, page 29
- Print Command, page 50
- Understanding Components: An Overview, page 33

View Log Command

Displays the log information for the selected drawing. To access this command, rightclick a drawing document and select **View Log** on the shortcut menu.



- Orthographic Drawings by Query Common Tasks, page 183
- Piping Isometric Drawings by Query Common Tasks, page 195
- Snapshot Drawings Common Tasks, page 155
- Volume Drawings Common Tasks, page 165

Drawing Tools: An Overview

Several drawing tools are provided to make customization of your drawings within the Drawings and Reports task easier. Some of these tools appear on the **Tools** menu in the Drawings and Reports task. Others are found on the right-click shortcut menu associated with the **Management Console** components and documents.

The tools available are:

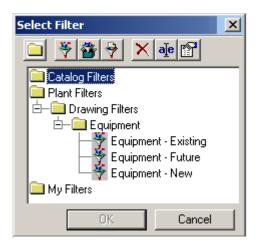
Define View Style Command (Tools Menu), page 54 Edit Border Template Command (Tools Menu), page 96 Define Key Plan Style Command (Tools Menu), page 97 View in 3D Command, page 104 Save As Command, page 105

Define View Style Command (Tools Menu)

Defines view styles for drawings and MicroStation 3d DGN documents. This command displays the **Select View Style** dialog box, which lists the available volume and snapshot view styles.

Each drawing view and MicroStation 3D DGN document in the software has an associated view style. A view style contains several rules for filtering, labeling, and creating the graphics in the output document.

You can set up the filters for view styles by using the **File > Define Workspace** or **Tools > Select by Filter** commands in any 3D task. One way to organize the drawing filters is by discipline, such as Piping, Equipment, or Structure. The following example shows Equipment filters.



You can also define filters within the **View Style Properties** dialog box by selecting **More** in the **Filter** dropdown.

For orthographic drawings, including snapshot and volume drawings, you can create and edit view styles using the **Tools > Define View Style** command. For isometric drawings, you can modify drawing output using the **Isometric Style Options Browser** dialog box. For more information on style for isometric drawings, see *Edit Options Command*, page 212.

The software interprets view styles differently depending on the component (volume or snapshot). For a volume type, the view style is applied only to objects that meet the criteria specified by filters in the view style. If the associated volume contains an object that is not matched in a row on the **View Style Properties** dialog box, then the software excludes the object from the resulting view.

For a snapshot component, the software processes all objects in the associated volume, and then applies the rows listed on the **View Style Properties** dialog box to the objects that match the filters.

The software comes with default view styles for a number of drawing types. These view styles can be used as delivered or copied and modified to suit your needs.

You can change the order of the rows on the **View Style Properties** dialog box by selecting a row and then clicking the up and down arrows on the left side of the dialog box. The software applies the graphic rules in a top-down manner. If an object finds its match in several filters, the last graphic rule applied (the bottom-most style) overrides any preceding graphic rules. Label rules and dimension rules accumulate.

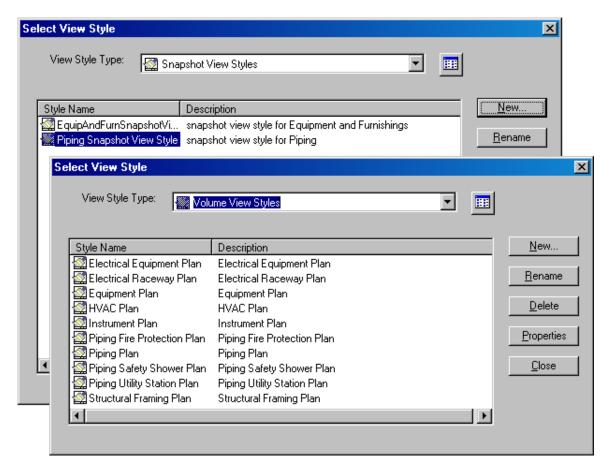
Note

• Within regard to the graphic rules specified as part of the view style properties, the software honors only the last aspect applied.

- Apply Grid Labels to Elevation and Section Views, page 70
- Create a View Style, page 66
- Dimensions: An Overview, page 109

Select View Style Dialog Box

Creates and edits view styles for drawings and MicroStation 3D DGN documents. You can open this dialog box by clicking **Tools > Define View Style**. When you create view styles, the software stores them as files in the **Symbols** share of the model.



View Style Type - Specifies a type of view style, either Volume View Styles or Snapshot View Styles. The type of view style selected populates the table with the available styles. There are several delivered Volume View Styles, but there are no delivered Snapshot View Styles. You need to create Snapshot View Styles before you can create Snapshot drawings.

Style Name - Lists the names of view styles.

Description - Lists the descriptions of the view styles.

New - Creates a new view style of the selected type. For more information, see *Create a View Style*, page 66.

Rename - Renames the selected view style.

Delete - Deletes the selected view style.

Properties - Displays the properties for the selected view style for editing. For more information, see View Style Properties Dialog Box, page 57.

Close - Dismisses the dialog box.

Related Topics

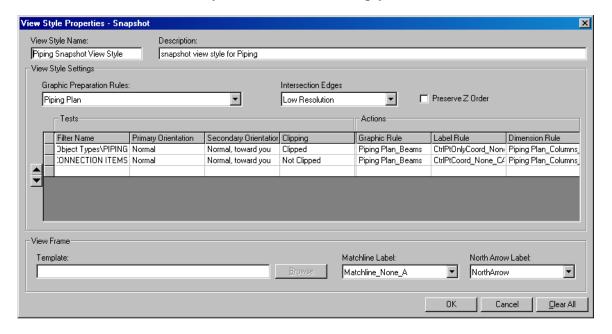
Define View Style Command (Tools Menu), page 54

View Style Properties Dialog Box

Defines view styles for drawings and MicroStation 3D DGN documents. You can open this dialog box by clicking **New** or **Properties** on the **Select View Style** dialog box.

The software stores the view styles and rules as files in the **Symbols** share.

If you are editing the properties of an existing view style, the current properties are shown. If this is a new view style, all the fields are empty.



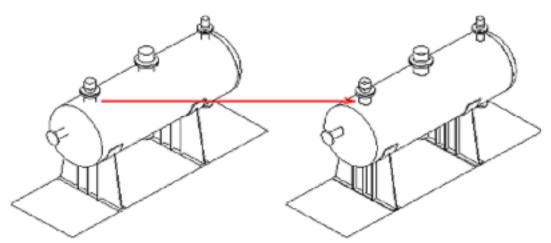
View Style Name - Specifies the name of the view style. The view style name can have up to 128 characters. If you are creating a new view style, you need to type a new name.

Description - Describes the view style.

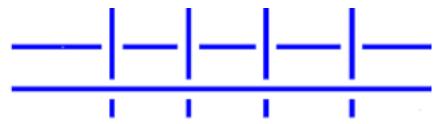
View Style Settings - Lists graphic preparation rules and resolution settings for intersections between objects. This section also lists tests and actions for the view style. Each row in the grid must contain a filter. If all the tests are true in a row, then the software executes the actions in that row.

Graphic Preparation Rules - Specifies or defines a graphic preparation rule. One rule may contain several modules (.dll files). A module can replace the threedimensional graphics in the model with other three-dimensional graphics. For more information, see Create or Edit a Graphic Preparation Rule, page 68.

Intersection Edges - Specifies the resolution at the intersection edges between objects. At High Resolution, more detail is shown in the drawing for the representation of intersecting surfaces. For example, changing from Off to High **Resolutions** would produce the following effect for your graphics:



Preserve Z Order - Maintains the three-dimensional order of resymbolized graphics. This setting overrides the top-down ordering in the view style for the graphic rules. For example, pipe gaps display according to their layering in the 3D model. A cleaner-looking drawing results if this option is selected.



Filter Name - Specifies a filter in the current model. You can choose existing filters or define new filters by selecting **More** in the dropdown.

Note

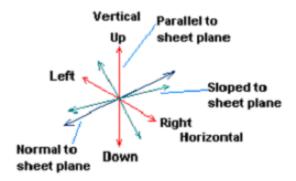
Asking or parameterized filters are not supported in view styles.

Primary Orientation - Lists the primary linear axis for items.

Secondary Orientation - Lists the secondary rotation axis, or roll, of items. An example is a structural stiffener that is parallel to the sheet plane but rotated 90 degrees about its primary axis.

You can resymbolize the model graphics for each filter based on special orientation cases. For example, you might want to represent piping items that are parallel to the sheet plane as single lines while representing the items coming straight out of the sheet as special symbols.

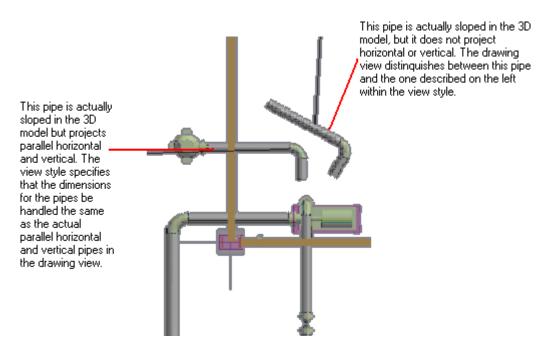
The following example shows the orientation options. The options in the **Primary Orientation** box are the same as the options in the **Secondary Orientation** options.



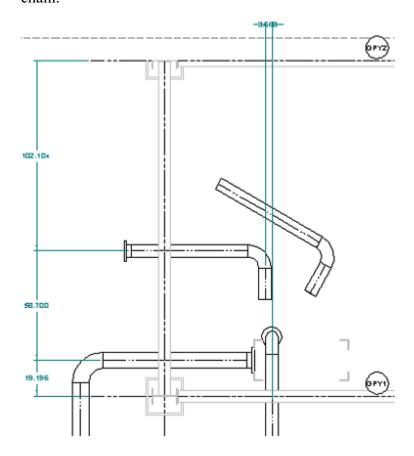
The options are as follows:

- Normal; Normal, toward you; and Normal, away from you
- Sloped; Sloped, toward you; Sloped, away from you; Sloped, projects horizontal; Sloped, projects vertical
- Parallel, vertical; Parallel, pointing up; Parallel, pointing down; Parallel, horizontal; Parallel, pointing right; Parallel, pointing left

The **Sloped, projects horizontal** and **Sloped, projects vertical** options provide special orientation for situations where the drawing needs to distinguish between sloped pipe that projects orthogonally on the sheet and those that project non-orthogonally on the sheet. You may, for example, prefer orthogonally-projected pipe be dimensioned in the drawing with the pipes that are parallel and orthogonal to the sheet. For example, the following graphic shows examples of a 3D model with pipe for both cases:



As shown in the following graphic, for dimensioning purposes, the **Parallel**, **horizontal** and **Sloped**, **projects horizontal** pipes are part of the same dimensioning chain.



Clipping - Lists the various clipping options. For example, you might want to put a cap on a pipe that is clipped to the right of the sheet.

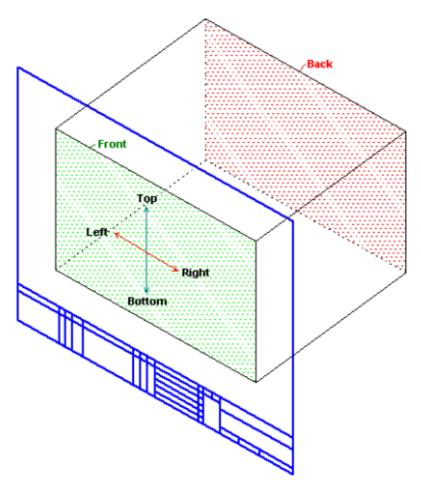
Use the **Clipped** option when testing for objects that are clipped by one of the sides (Top, Bottom, Right, or Left) of the volume. Dimensioning pipe that is clipped by the volume can take advantage of this option setting.

Use the **Not Clipped** option to test for objects that are not clipped at all.

Note

• The **Clipped** and **Not Clipped** options ignore front and back clipping.

The following picture illustrates the various clipping options - Left, Right, Front, Back, Top, and Bottom.

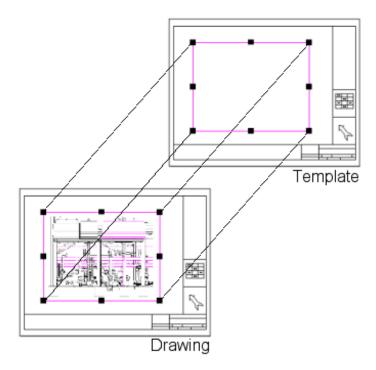


Graphic Rule - Determines how objects passing the tests will display. Objects can be displayed as Vector Hidden Line (VHL) or resymbolized. If you click **More**, you can define a new graphic rule. For more information on graphic rules, see *Graphic Rules*, page 78.

Label Rule - Determines the labeling to be applied to objects that pass the tests. Click **More** in the field to select a label rule. For more information, see *Label Rules*, page 92.

Dimension Rule - Controls the placement of dimensions on objects that pass the tests. Click **More** in the field to select a dimension rule. For more information, see *Dimensions: An Overview*, page 109.

Template - This field is currently not used.



Browse - This button is currently not available.

Matchline Label - Provides a selectable list of delivered matchline labels. Click **More** in the field to select a matchline rule. For more information, see *Matchline Rules*, page 94.

North Arrow Label - Provides a selectable list of delivered north arrow labels. Click **More** in the field to select a north arrow label. For more information, see *North Arrow Rules*, page 95.

Clear All - Removes all data in the grid, but not the name or description of the view style.

Notes

- You can change the order of the rows on the View Style Properties dialog box by selecting a row and then clicking the up and down arrows at the left side of the dialog box. If an object finds its match in several different filters, the last graphic rule applied (bottom most row) wins.
- To delete a row from the table, select the row and press DELETE on the keyboard.

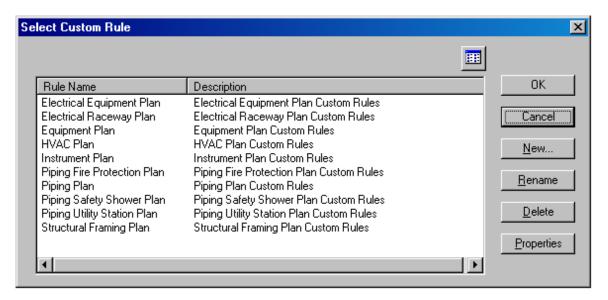
• If any text string is longer than the width of its box on the grid, a ToolTip provides the entire string.

Related Topics

- Create a View Style, page 66
- Define Key Plan Style Command (Tools Menu), page 97
- Define View Style Command (Tools Menu), page 54

Select Custom Rule Dialog Box

Lists the available custom graphic rules. You can open this dialog box by clicking **More** in the **Graphic Preparation Rules** box on the **View Style Properties** dialog box.



Views - Switches the view from **List** to **Details**. When you choose the List mode, descriptions of each rule do not appear.

Rule Name - Lists the names of the graphic rules.

Description - Lists the descriptions of the graphic rules.

New - Creates a new graphic rule of the selected type.

Rename - Renames the selected graphic rule.

Delete - Deletes the selected graphic rule.

Properties - Displays the properties for the selected graphic rule for editing.

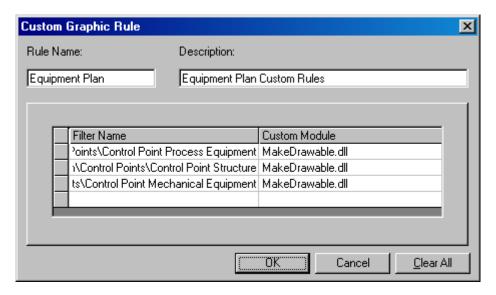
When you create a new custom rule or edit the properties for an existing rule, the Custom Graphic Rule dialog box displays. For more information, see Custom Graphic Rule Dialog Box, page 65.

Related Topics

- Create or Edit a Graphic Preparation Rule, page 68
- View Style Properties Dialog Box, page 57

Custom Graphic Rule Dialog Box

Sets options for a custom graphic preparation rule.



Rule Name - Specifies the name of the custom graphic rule.

Description - Describes the rule.

Filter Name - Specifies a filter. The filter sets the criteria to meet for the custom rule to be applied.

Custom Module - Specifies the module (a .dll file) that contains the changes you want to incorporate to objects meeting the criteria determined by the filter.

- Create or Edit a Graphic Preparation Rule, page 68
- Select Custom Rule Dialog Box, page 64
- View Style Properties Dialog Box, page 57

Select Dimension Rule Dialog Box

Lists the available dimension rules. You can open this dialog box by clicking **More** in the **Dimension Rule** box on the **View Style Properties** dialog box.

Views - Switches the view from **List** to **Details**. When you choose the List mode, descriptions of each rule do not appear.

Rule Name - Lists the names of the rules. For more information, see *Use Dimension Rules*, page 112.

Description - Lists the descriptions of the rules.

New - Creates a new rule. This button is not currently available.

Rename - Renames the selected rule. This button is not currently available.

Delete - Deletes the selected rule.

Properties - Allows you to edit the properties for the selected rule. This button is not currently available.

Related Topics

- Dimensions: An Overview, page 109
- View Style Properties Dialog Box, page 57

Create a View Style

- 1. Select **Tools > Define View Style**.
- 2. In the **View Style Type** box, select either the volume or snapshot style type.
- 3. Click New.
- 4. In the **View Style Name** box, type the style name.
- 5. In the **Description** box, type an optional description of the style.
- 6. Under **Graphic Preparation Rule**, select or define a rule, if necessary. The rule can contain multiple modules (.*dll* files).

Create or Edit a Graphic Preparation Rule, page 68

- 7. Specify the resolution in the **Intersection Edges** box.
- 8. Select the **Preserve Z Order** option if necessary. This option maintains the three-dimensional order of the resymbolized graphics.
- 9. In the **Filter Name** column, select **More** in the dropdown list to choose an existing filter or to create a new filter.
- 10. Select additional tests, if necessary, in the **Primary Orientation**, **Secondary Orientation**, and **Clipping** columns.

- 11. In the **Graphic Rule** column, select an existing graphic rule or create a new graphic rule by clicking **More**. Graphic rules control the line style and formatting of the output.
- 12. In the **Label Rule** column, select a labeling rule.
- 13. In the **Dimension Rule** column, select a dimension rule.
- 14. Select a matchline label, if applicable.
- 15. Select a north arrow label, if applicable.

Notes

- Test your view style by creating volume and snapshot drawings.
- You can edit a view style by selecting the style on the **Select View Style** dialog box and clicking **Properties**.
- The rules and associated files are saved in the **Symbols** share in the *Drawings\Catalog* folder.
- You can change the order of the rows on the **View Style Properties** dialog box by selecting a row and then clicking the up and down arrows at the left side of the dialog box. If an object finds its match in several different filters, the last graphic rule applied (bottom most row) wins.
- To delete a row in the table, select it and press DELETE on the keyboard.

Related Topics

- Create a Snapshot Drawing, page 158
- Create a Volume Drawing, page 167
- Define View Style Command (Tools Menu), page 54

Create a Custom Symbol for Volume or Snapshot Drawings

The software provides sample symbols to use in your drawings. Some of these symbols resize based on the size of the object they are replacing. You can create your own symbols or modify copies of existing ones to suit your needs. It is not necessary to have administrator privileges on the computer on which you perform this procedure. However, it is necessary to have write permissions at least to the Symbols share on the server computer.

- 1. Open the **Drawing Editor** by browsing to the [Product Directory]\Common2D\Shape2D\Bin folder and double-clicking **shape2dserver.exe**.
- 2. Draw your symbol using the various commands and tools in the software.
- 3. Use the **Select Tool** to draw a fence around the symbol graphics.
- 4. Click Create Symbol 🔟.

💡 Tip

- If the Create Symbol icon is not visible, click Tools > Customize, and drag the Create Symbol icon (from the Catalogs category) into the application window.
- 5. Click to define the origin of the new symbol.
- 6. On the **Save As Symbol** dialog box, browse to the \Drawings\Catalog\Symbols folder under the **Symbols** share, name the symbol, and save it.
- 7. Exit the **Drawing Editor** without saving the document.

Note

 You can incorporate your custom symbol into a graphic rule for a view style for use in any orthographic drawings. For more information, see *Graphic Rule - Symbol Dialog Box*, page 89.

Related Topics

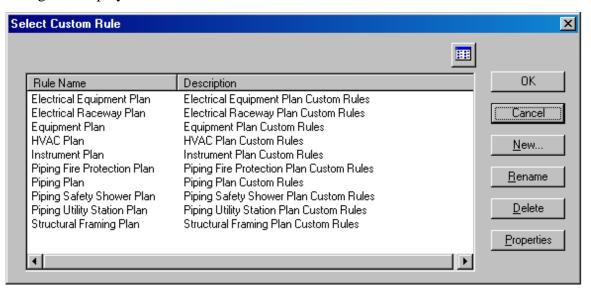
- Create a View Style, page 66
- Understanding the View Style Rules: An Overview, page 76

Create or Edit a Graphic Preparation Rule

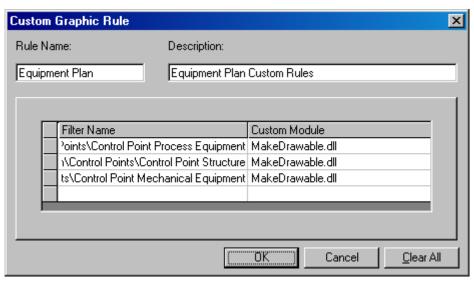
Graphic preparation rules allow you to customize the graphics included in your view style. You can use a graphic preparation rule to implement functionality that does not generally support interface that the Drawings environment requires. For example, the MakeDrawable custom module forces objects that cannot normally appear in a drawing, such as piping runs objects, to be drawable.

- 1. Click **Tools > Define View Style**.
- 2. In the **View Style Type** box, select either the volume or snapshot style type.
- 3. Click New.

4. In the Graphic Preparation Rule box, select More. The Select Custom Rule dialog box displays.



5. Click **New** to create a new graphic rule, or select an existing rule and click **Properties** to edit the properties. The **Custom Graphic Rule** dialog box displays.



- 6. In the **Rule Name** box, type a name.
- 7. In the **Description** box, type a description.
- 8. In the **Filter Name** column, select a filter, such as Equipment.
- 9. In the **Custom Module** column, select a .dll file.
- 10. Click **OK** on the **Custom Graphic Rule** dialog box.
- 11. On the **Select Custom Rule** dialog box, select the new or modified rule, and click **OK** to return to the View Style Properties dialog box. You can continue specifying parts of the view style on this dialog box.

Related Topics

- Apply Grid Labels to Elevation and Section Views, page 70
- Create a View Style, page 66

Apply Grid Labels to Elevation and Section Views

The following procedure describes how to apply grid labels to elevation and section views on drawings. Custom rules are required along with filters for Grid Plane objects.

The example workflow uses the *GridLinesDrawingWrapperEntity.dll* custom rule to resymbolize the grid plane as a set of lines. The lines draw at all perpendicular plane locations and extend along the elevation plane from (lowest Z-1)m to (highest Z+1)m of the coordinate system to which the grid plane belongs. For example, if the coordinate system has X=0,10,20, Y=0,5,10 and EL=0,10, then for the grid plane at X=0, there will be three lines:

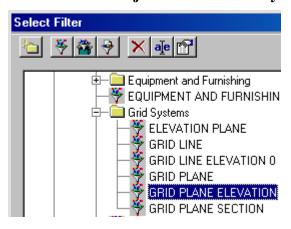
- Y=0 from **Z-1** to **Z+11**
- Y=5 from Z-1 to Z+11
- Y=10 from Z-1 to Z+11

Check for the Required Filters

If the catalog you are using does not have filters for Grid Plane Elevation and Sections, you need to create them. You need one for "Elevations" looking North/South and another for "Sections" looking East/West.

Note

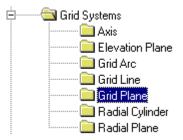
• Check the Catalog for the filters. If you are using the delivered default filters in version 6.1 or later, these two filters are delivered under **Catalog Filters** > **SP3D Object Filters** > **Grid Systems**.



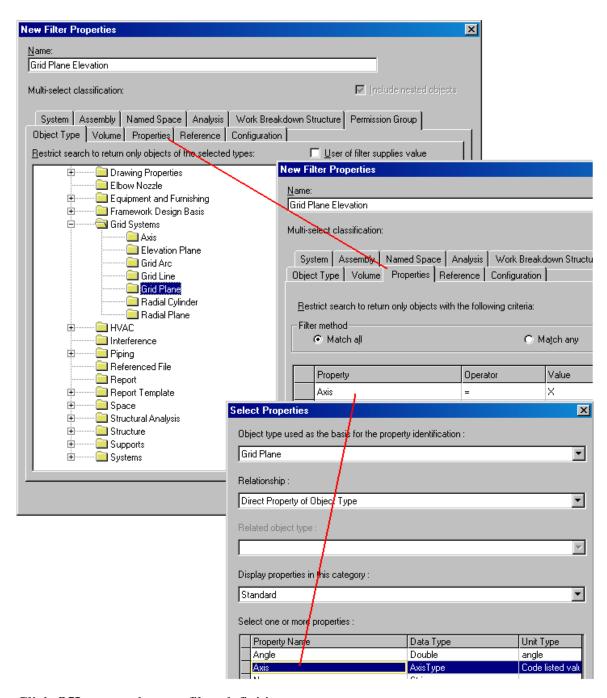
Create the Required Filters

If the Grid Plane Elevation and Sections filters do not exist, you can create them. For more information on creating filters, see the Common User's Guide available from **Help > Printable Guides.**

- 1. In the **Common** task, use **File > Define Workspace** to create two new filters. On the Define Workspace dialog box, select Create new filter in the Filter dropdown. The **New Filter Properties** dialog box displays.
- 2. Go to the **Object Type** tab and select **Grid Systems**. Expand **Grid Systems** and select Grid Plane.



3. Go to the **Properties** tab to further restrict the filter. For example, select **Grid Planes** and restrict to Axis = X for the Elevation filter and Axis = Y for the Sections filter. Select More in the Property dropdown. The Select Properties dialog box displays so you can specify the property definition for the **Grid Plane** object. Click **OK** to return to the **Properties** tab to set the value for the selected property. Refer to the graphic below for more information.



4. Click **OK** to save the new filter definitions.

Create a View Style

After checking to make sure you have the required filters or creating new filters, you need to create a view style that uses the filters to create the grid labels on your drawings. Go to the **Drawings and Reports** task to perform the steps below.

- 1. Select **Tools > Define View Style**. The **Select View Style** dialog box displays.
- 2. Select a **View Style Type** and click **New**. The **View Style Properties** dialog box displays.

- 3. Specify a name and description for the new view style. For example, you could create a view style called Elevation Grid Bubbles.
- 4. In the Graphic Preparation Rule box, select More. The Select Custom Rule dialog box displays.
- 5. Click **New** to create a new graphic rule. The **Custom Graphic Rule** dialog box displays.
- 6. Select the **Grid Plane Elevation** filter by selecting **More** in the **Filter Name** dropdown.
- 7. Apply the *GridLinesDrawingWrapperEntity.dll* custom rule to resymbolize the vertical grid planes as lines.
- 8. Click **OK** on the **Custom Graphic Rule** dialog box.
- 9. On the **Select Custom Rule** dialog box, select the new rule, and click **OK** to return to the View Style Properties dialog box.
- 10. Specify a graphic rule to apply color or style to the graphics in the drawing.

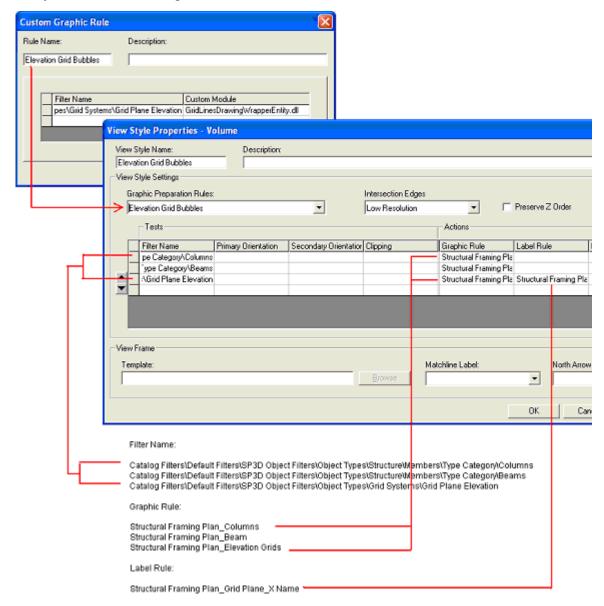
Note

- We recommend the "invisible" style to prevent the lines for Grid Planes from appearing. The software labels them even when invisible.
- 11. Apply a label rule that returns the name as its value.

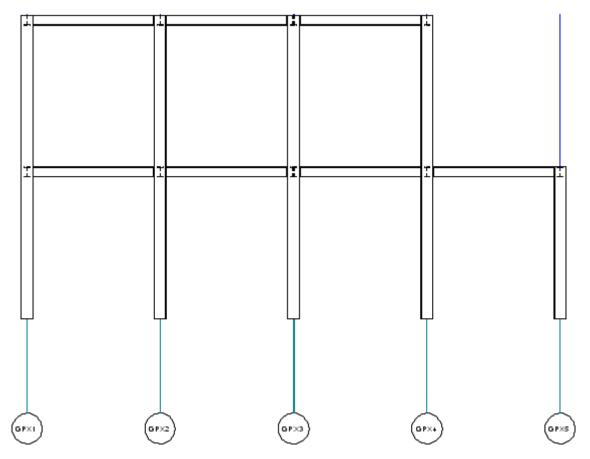
Note

We recommend copying one of the delivered Grid labels and its supporting files in the \Symbols\Drawings\Catalog\Labels\Templates folder and the \Symbols\Drawings\Catalog\Rules\LabelRules folder. Rename the new label and substitute one of the delivered name label .rge files for the new labels query.

12. After defining the view style, click **OK** to save the settings. The view style is now ready to use with drawings.



The example workflow above uses columns and beams to give the results of the view style application and meaning. The row referencing Grid Plane Elevations is the only required row. In a drawing using the new view style, the Grid Planes are blue based on the settings from the view style:



Notes

- Some modeling situations have intersections that cause unwanted Grid Planes to appear and be labeled. You may need to edit the drawing document and delete unwanted labels. Be sure to save changes to make to the drawing document so that subsequent updates are remembered.
- Use the procedure above to create separate view styles for Elevation and Section views. You can then create Elevation and Section view drawing type packages using the appropriate view styles. For more information, see Save Package Command, page 38.

- Create a View Style, page 66
- Create or Edit a Graphic Preparation Rule, page 68
- Define View Style Command (Tools Menu), page 54

Understanding the View Style Rules: An Overview

View styles consist of tests and actions. One of the tests must be a filter, and the actions are rules that control the drawing results. Each rule is an XML file, which is used as an input to the view style XML file. The rule XML files are located on the **Symbols** share in the \Drawings\Catalog\Rules folder.

Graphic Rules

Graphic rules control the representation of the model graphics on drawings. For example, you can represent pipe runs as single lines while representing equipment as control point symbols. You can also resymbolize structural members as single lines with detailed widgets placed along the lines. To specify the resymbolization, you create a custom symbol and incorporate the symbol into a graphic rule for a view style. For more information, see *Graphic Rules*, page 78.

Label Rules

Label rules control automatic label placement on drawings. For more information, see *Label Rules*, page 92.

Before creating a new label rule, you must first create the label query using the **Define Label** command in the Catalog task.

Dimension Rules

Dimension rules control the placement of dimensions while dimension styles control the appearance, including the units, of dimensions on orthographic drawings. You can specify the granularity, offsets, style, text size, and object types for the dimensions. For more information on working with dimensions, see *Dimensions: An Overview*, page 109.

Matchline Rules

The matchline rule uses labels to specify the drawing extents and the names of adjoining drawings. For more information, see *Matchline Rules*, page 94.

North Arrow Rules

The north arrow rule uses a label to specify the direction toward north on drawings. For more information, see *North Arrow Rules*, page 95.

Note

The drawing dimension and label rules are discussed further in the SmartPlant 3D/IntelliShip Programmer's Guide under Extending the Capabilities of the Software. Contact your administrator or Intergraph Support if you need the *Programmer's Guide*.

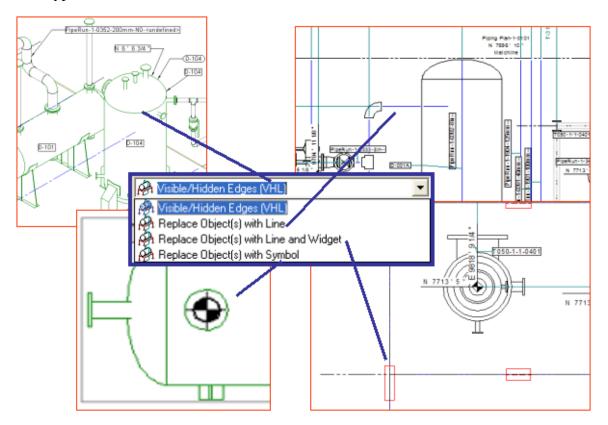
- Create a View Style, page 66
- Define View Style Command (Tools Menu), page 54

Graphic Rules

Graphic rules control the representation of the model graphics on drawings. Each filter in the view style can have its own graphic rule. For example, you can represent pipe runs as single lines while representing equipment as control point symbols. You can also resymbolize structural members as single lines with detailed widgets placed along the lines.

You specify graphic rules when you edit an existing view style or create a new view style with the **Tools > Define View Style** command. In the **Graphic Rule** dropdown on the **View Style Properties** dialog box, select **More** to display the **Select Graphic Rule** dialog box.

You have several types of rules available. The following graphic shows examples for each type.



Note

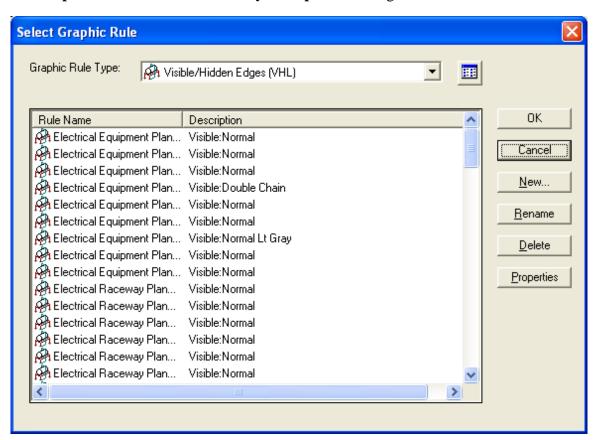
• If an object is filtered in more than one row of the view style, the graphic rule in the bottom-most row applies to the object.

Related Topics

• Understanding the View Style Rules: An Overview, page 76

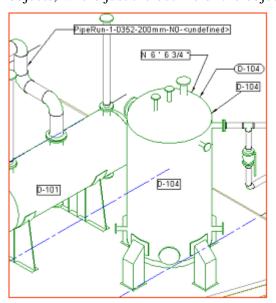
Select Graphic Rule Dialog Box

Lists the available graphic rules. You can open this dialog box by clicking **More** in the **Graphic Rule** box on the **View Style Properties** dialog box.

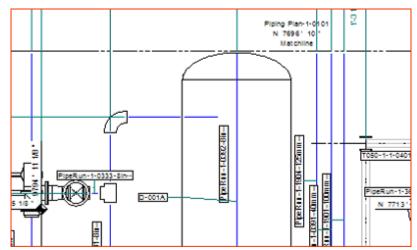


Graphic Rule Type - Specifies a type of graphic rule. The rule types available include:

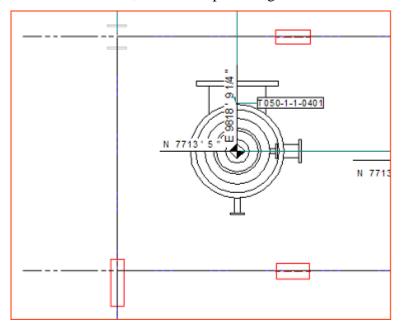
• **Visible/Hidden Lines (VHL)** - Creates a "wireframe" representation of objects, where just the outline of the object is drawn.



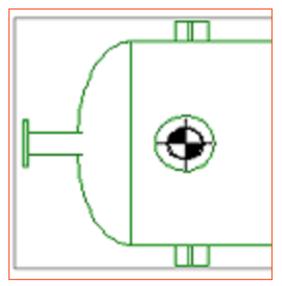
• Replace Objects with Line - Replaces objects, such as pipes, with lines.



Replace Objects with Line and Widget - Replaces objects, such as structural members, with a line plus widget combination.



Replace Objects with Symbol - Replaces objects with symbols.



Wiews - Switches the view from **List** to **Details**. When you choose List mode, descriptions of each rule to not appear.

Rule Name - Lists the names of the graphic rules.

Description - Lists the descriptions of the graphic rules.

New - Creates a new graphic rule of the selected type.

Rename - Renames the selected graphic rule.

Delete - Deletes the selected graphic rule.

Properties - Displays the properties for the selected graphic rule for editing.

When creating a new graphic rule or editing the properties of an existing graphic rule, see the following for more information:

Graphic Rule - VHL Dialog Box, page 82 Graphic Rule - Line Dialog Box, page 84

Graphic Rule - Line with Widget Dialog Box, page 86

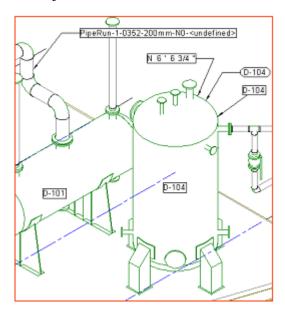
Graphic Rule - Symbol Dialog Box, page 89

Related Topics

- Graphic Rules, page 78
- View Style Properties Dialog Box, page 57

Graphic Rule - VHL Dialog Box

Sets options for a specific Visible/Hidden Lines (VHL) graphic rule. The VHL graphic rules create a "wireframe" representation of objects, were just the outline of the object is drawn.



Rule Name - Specifies the name of the graphic rule. The rule name can be up to 128 characters long.

Description - Describes the rule.

Graphic Module - Specifies a module (.*dll* file) that allows you to customize the graphics included in you graphic rule.

Note

This module affects only those items that meet the criteria of the filter on the selected row of the table in the View Style Properties dialog box. To apply a module to all items in the view style, choose a **Graphic** Preparation Rule on the View Style Properties dialog box.

Visible Line Style - Specifies the line style for visible edges of objects.

Hidden Line Style - Specifies the line style for edges of an object hidden by other objects.

Hidden By Self Line Style - Specifies the line style for edges of an object that are hidden by itself.

Occluded Line Style - Specifies a line style for edges of an object that are aligned with and completely hidden by edges of another object. For example, you may use this style when you have two pipes of the same diameter but at different elevations and the drawing is looking down at them.

Layer - Indicates the MicroStation level on which the graphic is placed. This setting is currently only used with the MicroStation 3D DGN component.

! Important

Only whole numbers between 1 and 63, inclusive, are accepted in this field.

Visible Fill Style - Specifies the fill style for visible surfaces of objects. This setting is currently not supported.

Hidden Fill Style - Specifies the fill style for hidden surfaces of objects. This setting is currently not supported.

Aspect (Multi-Select) - Specifies the envelope around the objects for VHL processing. Examples are maintenance or insulation envelopes. You can select multiple aspects by holding the CTRL or SHIFT keys. This setting is currently not supported.

Show Centerline - Specifies whether the centerline appears in addition to the 3D graphics. If you reset this property to No, the Centerline Visible Line Style and Centerline Hidden Line Style boxes display <Not Drawn>.

Centerline Visible Line Style - Specifies the line style for a visible centerline.

Centerline Hidden Line Style - Specifies the line style for a hidden centerline.

Clipping - Indicates whether or not the graphics are clipped within the drawing. For example, in plan views, certain objects such as stairs or ladders are best represented unclipped by the drawing volume. You would specify a filter row in the view style for stairs or ladders and set this option to unclipped to achieve the desired graphic representation.

Note

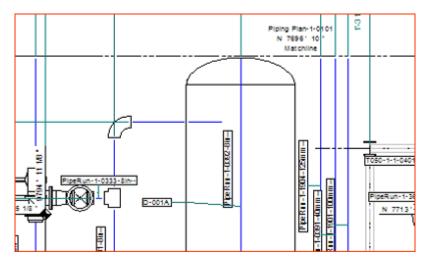
• All of the line styles in the pull down lists are found in the *Styles.sha* file, located in the **Symbols** share in the \(\Drawings\Catalog\Templates\) folder. You can add new line styles to this file.

Related Topics

- *Graphic Rules*, page 78
- Graphics Tab (Graphic Rule Line with Widget Dialog Box), page 87
- Select Graphic Rule Dialog Box, page 79
- View Style Properties Dialog Box, page 57
- Widget Tab (Graphic Rule Line with Widget Dialog Box), page 88

Graphic Rule - Line Dialog Box

Sets options for a single line graphic rule. The single line graphic rules replace linear objects, such as pipes and structural members, in the graphic view with single lines. The graphic below shows pipes replaced with a blue line.



Rule Name - Specifies the name of the graphic rule. The rule name can be up to 128 characters long.

Description - Describes the rule.

Graphic Module - Specifies a module (.*dll* file) that allows you to customize the graphics included in your graphic rule.

Note

This module affects only those items that meet the criteria of the filter on the selected row of the table in the **View Style Properties** dialog box. To apply a module to all items in the view style, choose a **Graphic** Preparation Rule on the View Style Properties dialog box.

Visible Line Style - Specifies the line style for visible lines.

Hidden Line Style - Specifies the line style for lines hidden by other objects.

Occluded Line Style - Specifies a line style for lines that are aligned with and completely hidden by the lines or edges of other objects.

Layer - Indicates the MicroStation level on which the graphic is placed. This setting is currently only used with the MicroStation 3D DGN component.

! Important

Only whole numbers between 1 and 63, inclusive, are accepted in this field.

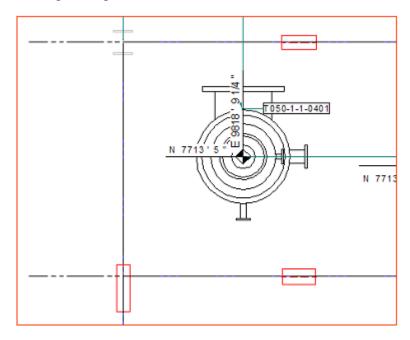
Note

All of the line styles in the pull down lists are found in the *Styles.sha* file, located in the **Symbols** share in the \Drawings\Catalog\Templates folder. You can add new styles to this file.

- Graphic Rules, page 78
- Graphics Tab (Graphic Rule Line with Widget Dialog Box), page 87
- Select Graphic Rule Dialog Box, page 79
- View Style Properties Dialog Box, page 57
- Widget Tab (Graphic Rule Line with Widget Dialog Box), page 88

Graphic Rule - Line with Widget Dialog Box

Sets options for a line with widget graphic rule. The line with widget graphic rules replace objects, such as pipes, in the graphic view with a line plus widget combination. A widget is a symbol placed at a user-defined spacing along the line. The graphic below shows a line plus widget combination of a black line with a red rectangle widget.



Rule Name - Specifies the name of the graphic rule. The rule name can be up to 128 characters long.

Description - Describes the rule.

To set the line and fill style properties, go to the *Graphics Tab (Graphic Rule - Line with Widget Dialog Box)*, page 87.

To set the widget properties, go to the Widget Tab (Graphic Rule - Line with Widget Dialog Box), page 88.

- *Graphic Rules*, page 78
- Graphics Tab (Graphic Rule Line with Widget Dialog Box), page 87
- Select Graphic Rule Dialog Box, page 79
- View Style Properties Dialog Box, page 57
- Widget Tab (Graphic Rule Line with Widget Dialog Box), page 88

Graphics Tab (Graphic Rule - Line with Widget Dialog Box)

Sets line and fill style properties for a line with widget combination graphic rule. To set the widget properties, go to the Widget Tab (Graphic Rule - Line with Widget Dialog Box), page 88.

Graphic Module - Specifies a module (.dll file) that allows you to customize the graphics included in your graphic rule.



This module affects only those items that meet the criteria of the filter on the selected row of the table in the **View Style Properties** dialog box. To apply a module to all items in the view style, choose a **Graphic** Preparation Rule on the View Style Properties dialog box.

Visible Line Style - Specifies the line style for visible lines.

Hidden Line Style - Specifies the line style for lines hidden by other objects.

Occluded Line Style - Specifies a line style for lines that are aligned with and completely hidden by the lines and edges of other objects.

Layer - Indicates the MicroStation level on which the graphic is placed. This setting is currently only used with the MicroStation 3D DGN component.

• Important

Only whole numbers between 1 and 63, inclusive, are accepted in this field.

Note

All of the line styles in the pull down lists are found in the Styles.sha file, located in the **Symbols** share in the \Drawings\Catalog\Templates folder. You can add new line styles to this file.

- Graphic Rule Line with Widget Dialog Box, page 86
- Graphic Rules, page 78
- Graphics Tab (Graphic Rule Line with Widget Dialog Box), page 87
- Select Graphic Rule Dialog Box, page 79
- View Style Properties Dialog Box, page 57
- Widget Tab (Graphic Rule Line with Widget Dialog Box), page 88

Widget Tab (Graphic Rule - Line with Widget Dialog Box)

Sets options for widgets (symbols) on a line for the line plus widget combination graphic rule. To set the line and fill style properties, go to the *Graphics Tab (Graphic Rule - Line with Widget Dialog Box)*, page 87.

Widget Graphics - Specifies whether an aspect or symbol should be used as part of the graphic rule.

Aspect Properties

Aspect (Multi-Select) - Specifies one or more envelopes to display around objects. You can select multiple aspects by holding the **CTRL** or **SHIFT** keys.

Aspect Length (%) - Sets the length of the aspect as a percentage of the line on which it is placed.

Aspect Height (%) - Sets the height of the aspect as a percentage of the height of the object.

Widget Position (% along object) - Sets the positioning of the widget as a percentage of total length from the start of the line.

Visible Line Style - Specifies the line style for visible edges of the aspect.

Hidden By Self Line Style - Specifies the line style for edges of an aspect that are hidden by itself.

Show Centerline - Specifies whether the centerline appears in addition to the 3D graphics. If you reset this property to **No**, the **Centerline Visible Line Style** and **Centerline Hidden Line Style** boxes display **<Not Drawn>**.

Centerline Visible Line Style - Specifies the line style for a visible centerline.

Centerline Hidden Line Style - Specifies the line style for a hidden centerline.

Symbol Properties

Symbol - Specifies a symbol file. You can click **More** to open the **Choose Symbol** dialog box. Symbols are saved in the Symbols share in the \Drawings\Catalog\Symbols folder. For more information, see Choose Symbol Dialog Box, page 90.

Widget Position (% along object) - Sets the positioning of the widget as a percentage of the total length from the start of the line.

Show Centerline - Specifies whether the centerline appears in addition to the 3D graphics. If you reset this property to **No**, the **Centerline Visible Line Style** boxes display **<Not Drawn>**.

Centerline Visible Line Style - Specifies the line style for a visible centerline.

Centerline Hidden Line Style - Specifies the line style for a hidden centerline.



All of the line styles in the pull down lists are found in the *Styles.sha* file, located in the **Symbols** share in the \Drawings\Catalog\Templates folder. You can add new line styles to this file.

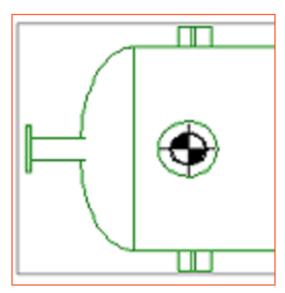
Related Topics

- Graphic Rule Line with Widget Dialog Box, page 86
- Graphic Rules, page 78
- Graphics Tab (Graphic Rule Line with Widget Dialog Box), page 87
- Select Graphic Rule Dialog Box, page 79
- View Style Properties Dialog Box, page 57
- Widget Tab (Graphic Rule Line with Widget Dialog Box), page 88

Graphic Rule - Symbol Dialog Box

Specifies a symbol for a graphic rule. The symbol graphic rules replace objects in the graphic view with a symbol. The graphic below shows an object replaced by a





The symbol files are saved in the **Symbols** share in the \Drawings\Catalog\Symbols folder.

Rule Name - Names of the graphic rule. The rule name can be up to 128 characters long.

Description - Describes the rule.

Graphic Module - Specifies a module (.*dll* file) that allows you to customize the graphics included in your graphic rule.

Note

This module affects only those items that meet the criteria of the filter on
the selected row of the table in the View Style Properties dialog box. To
apply a module to all items in the view style, choose a Graphic
Preparation Rule on the View Style Properties dialog box.

Symbol - Specifies a symbol for objects that are visible. You can click **More** to open the **Choose Symbol** dialog box. For more information, see *Choose Symbol Dialog Box*, page 90.

Hidden Object Symbol - Specifies a symbol for objects that are hidden.

Layer - Indicates the MicroStation level on which the graphic is placed. This setting is currently only used with the MicroStation 3D DGN component.

! Important

• Only whole numbers between 1 and 63, inclusive, are accepted in this field.

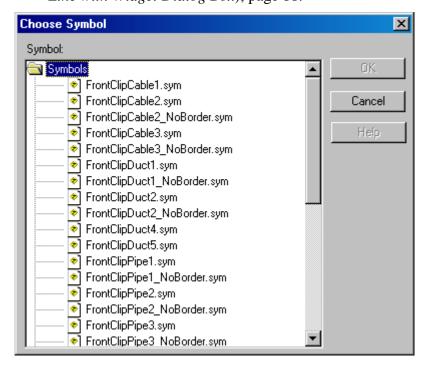
Related Topics

- Graphic Rule Line with Widget Dialog Box, page 86
- *Graphic Rules*, page 78
- Graphics Tab (Graphic Rule Line with Widget Dialog Box), page 87
- Select Graphic Rule Dialog Box, page 79
- View Style Properties Dialog Box, page 57
- Widget Tab (Graphic Rule Line with Widget Dialog Box), page 88

Choose Symbol Dialog Box

Specifies a symbol for the replace with symbol and replace with line and widget graphic rule types. You can open this dialog box in two ways:

 Select More in the Symbol or Hidden Object Symbol fields on the Graphic Rules - Symbols dialog box. For more information, see *Graphic Rule - Symbol Dialog Box*, page 89. Select More in the Symbol field on the Widget tab of the Line with Widget dialog box. For more information, see Widget Tab (Graphic Rule - Line with Widget Dialog Box), page 88.

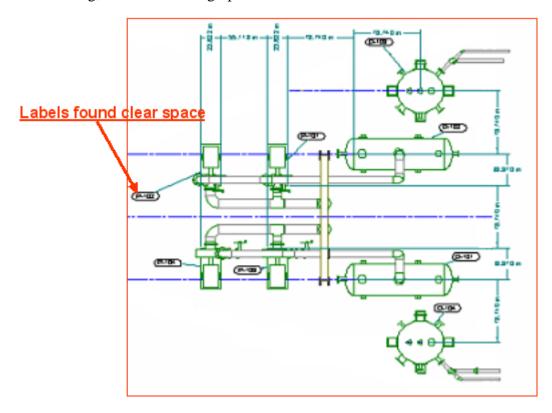


Symbol - Shows a hierarchical list of available symbols from the Symbols share in the \Drawings\Catalog\Symbols folder. Select a symbol from the list and click **OK** to return to the previous dialog box.

- Graphic Rule Line with Widget Dialog Box, page 86
- Graphic Rules, page 78
- Graphics Tab (Graphic Rule Line with Widget Dialog Box), page 87
- Select Graphic Rule Dialog Box, page 79
- View Style Properties Dialog Box, page 57
- Widget Tab (Graphic Rule Line with Widget Dialog Box), page 88

Label Rules

Label rules control the appearance of labels as well as their automatic placement on drawings. For example, you can create label rules that place labels with our without borders and leader lines. You can even specify that the labels find "clear space" on the drawing, as shown in the graphic.



You can use these rules for many purposes. The delivered sample rules include several types of rules, such as control point labels, grid line labels, name labels, name and part labels, and piping labels.

The label rules are saved in the **Symbols** share in the \\Drawings\\Catalog\\Rules\\LabelRules\ folder. The rules have corresponding templates and symbols under \\Drawings\\Catalog\\Labels\\Templates\. For more information, you can open the rule and template XML files to view the comments in the code.

• Understanding the View Style Rules: An Overview, page 76

Select Label Rule Dialog Box

Lists the available label rules. You can open this dialog box by clicking **More** in the **Label Rule** field on the **View Style Properties** dialog box. You can also open this dialog box from the **Matchline Label** or **North Arrow Label** lists.

Views - Switches the view from **List** to **Details**. When you choose the List mode, descriptions of each rule do not appear.

Rule Name - Lists the names of the rules.

Description - Lists the descriptions of the rules.

New - Creates a new rule. This button is not currently available.

Rename - Renames the selected rule. This button is not currently available.

Delete - Deletes the selected rule.

Properties - Allows you to edit the properties for the selected rule. This button is not currently available.

- Label Rules, page 92
- View Style Properties Dialog Box, page 57

Matchline Rules

Matchline rules are used to label adjoining volumes on drawings.

The matchline rules (the standard rule delivered and any new rules you create) are saved in the Symbols share in the \Drawings\Catalog\Rules\MatchlineRules folder. The rules have corresponding templates and symbols under \Drawings\Catalog\Labels\Templates\Matchline. For more information, you can open the rule and template XML files to view the comments in the code.

Related Topics

• Understanding the View Style Rules: An Overview, page 76

North Arrow Rules

North arrow rules place a label that points toward Global North on drawings.

The north arrow rules are saved in the Symbols share in the \Drawings\Catalog\Rules\NorthArrowRules folder. The rules have corresponding templates and symbols under $\Drawings\Catalog\Labels\Templates\NorthArrows$. For more information, you can open the rule and template XML files to view the comments in the code.

Understanding the View Style Rules: An Overview, page 76

Edit Border Template Command (Tools Menu)

Opens a drawing border template in the 2D Drawing Editor for customization. You can place drawing property labels and manual graphics.

When you place drawing property labels, the software automatically makes the **DwgTemplate** layer active. The labels need to be on this layer so that they are preserved when you update the drawing.



If you use other native 2D Drawing Editor commands (such as Place Line or Place Dimension) to add manual markups to the template, put them on the **Default** or a layer with "User" as the prefix (for example, a layer named **UserAnnotationLayer** to preserve the changes when you update drawings.



Related Topics

- Create a Volume Drawing, page 167
- Edit Template Command (Volume Drawings Component), page 169
- Orthographic Drawings by Query: An Overview, page 181
- Piping Isometric Drawings by Query: An Overview, page 193
- Volume Drawings: An Overview, page 161

Select Template Dialog Box

Specifies a template. This dialog box appears when you click the **Edit Border Template** command. It also appears the first time you edit a template for a volume component or create a snapshot drawing. The templates listed in this dialog box are located on the **Symbols** share in the \Drawings\Catalog\Templates folder.

You can select a template on this dialog box and then click \mathbf{OK} , or you can just double-click a template.

The application delivers a set of Imperial and Metric border templates. The names of the templates indicate their size. All of the delivered Imperial and Metric border templates already contain border labels. Some of the border templates also contain a label to display notes. The naming convention indicates which templates contain this label.

- Create a Volume Drawing, page 167
- Edit Border Template Command (Tools Menu), page 96
- Edit Template Command (Volume Drawings Component), page 169
- Volume Drawings: An Overview, page 161

Define Key Plan Style Command (Tools Menu)

Defines a key plan style for use in a volume drawing.

This command allows you to create key plan styles to use when placing key plans in a volume drawing template. The key plan style is specified on the **Key Plan Properties** dialog box when you place a key plan using the **Place Key Plan** command. For more information, see *Place Key Plan Command (Template Toolbar)*, page 140.

There are three types of key plan styles you can create. They are described below.

Normalized Volumes Only

This key plan type specifies that the key plan be arranged in a normalized manner, meaning that each volume has exactly the same amount of space in the key plan. The graphic below represents the layout of the Normalized Volumes Only key plan type.

300	400	500
600	700	800

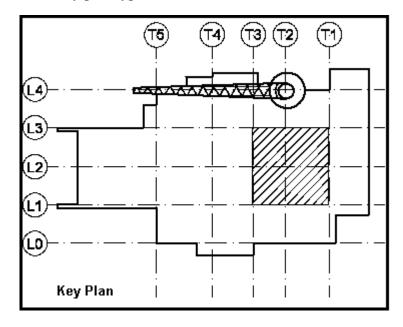
Natural Volumes Only

The Natural Volumes Only type specifies that the key plan displays the volumes by natural size. This means that the key plan layout is based on the size of each volume, which may be irregular if the volumes are of different sizes. The graphic below shows the layout of a Natural Volumes Only key plan type.

400	500
700	800

One Volume with Plant View

This key plan type shows the full plant view with a "you are here" representation for the associated volume. The graphic below is a sample of the One Volume with Plant View key plan type.



The **Define Key Plan Style** dialog box provides the options for creating new key plan styles or modifying existing ones. The type of key plan selected changes the options available. The key plan **View Style** property specifies the view style you want to use within the key plan.

Related Topics

• Define a Key Plan Style, page 102

Define Key Plan Style Dialog Box

Lists available key plan styles and allows you to create new styles or modify existing styles. You can also delete or rename existing styles. You access this dialog box by selecting **Tools > Define Key Plan Style**.

New - Displays the **Key Plan Style** dialog box to create a new style. For more information, see *Key Plan Style Dialog Box*, page 99.

Rename - Renames the selected key plan style.

Delete - Deletes the selected key plan style.

Properties - Displays properties for the selected key plan style for editing.

Rule Name - Lists all currently available key plan styles.

Wiews - Switches the view from **List** to **Details**. When you choose the List mode, descriptions of each rule do not appear.

Related Topics

- Define a Key Plan Style, page 102
- Define Key Plan Style Command (Tools Menu), page 97

Key Plan Style Dialog Box

Sets options for a key plan style. You access this dialog box when you select New or Properties on the Define Key Plan Style dialog box.

Name - Specifies the name of the key plan styles.

Description - Describes the key plan style.

Key Plan Type - Specifies a type for the key plan. There are several types of key plan styles you can create. They are described below. The properties displayed depend on the key plan type you select.

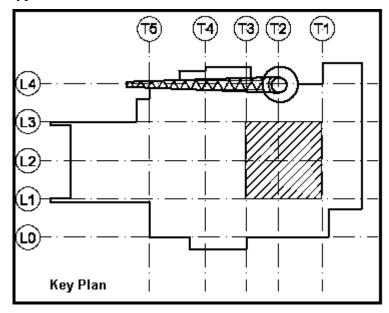
Normalized Volumes Only - This key plan type specifies that the key plan be arranged in a normalized manner, meaning that each volume has exactly the same amount of space in the key plan. The graphic below represents the layout of the Normalized Volumes Only key plan type. The **Default** key plan that is delivered with the software is an example of this type of key plan.

300	400	500
600	700	800

• Natural Volumes Only - The Natural Volumes Only type specifies that the key plan displays the volumes by natural size. This means that the key plan layout is based on the size of each volume, which may be irregular if the volumes are of different sizes. The graphic below shows the layout of a Natural Volumes Only key plan type.

300	400	500
600	700	800

• One Volume with Plant View - This key plan type shows the full plant view with a "you are here" representation for the associated volume. The graphic below is a sample of the One Volume with Plant View key plan type.



Specifying Properties for a Key Plan

The following properties are available for the **Normalized Volumes Only** key plan type:

The Coordinate System property provides a dropdown list of all coordinate systems in the model. It also includes the value Use
 Coordinate System of Related View. Selecting this value will cause the key plan to use the same coordinate system as the associated drawing view.

- The **Search Range in X direction** property determines the horizontal extents the software uses to locate another drawing volume with the same parent as the current drawing. The larger the value, the larger the extents.
- The **Search Range in Y direction** property determines the vertical extents the software uses to locate another drawing volume with the same parent as the current drawing. The larger the value, the larger the extents.
- The **Alignment in X direction** property specifies a tolerance for the software to use when determining whether to align the vertical sides of volumes within the key plan. The larger the tolerance, the more likely the vertical sides will align.
- The **Alignment in Y direction** property specifies a tolerance for the software to use when determining whether to align the horizontal sides of volumes within the key plan. The larger the tolerance, the more likely the horizontal sides will align.
- The **Normalized in X direction** property specifies a tolerance for the software to use when determining whether to represent volumes within the key plan with the same width as the volume of the current drawing. The larger the tolerance, the more likely the volumes will be resized.
- The **Normalized in Y direction** property specifies a tolerance for the software to use when determining whether to represent volumes within the key plan with the same height as the volume of the current drawing. The larger the tolerance, the more likely the volumes will be resized.
- The **View Style** property specifies the style used to display the key plan graphics. Click More to display the Select Key Plan View Style dialog box. For more information, see Define Key Plan Style Dialog Box, page 98.

Notes

- To apply a style to the volume of the current drawing in the key plan, use the term KEY PLAN FOCUS ELEMENT in the Filter column of the key plan view style and then assign a graphic rule. You must use uppercase letters for the term.
- To apply a style to other drawing volumes in the key plan, use the term KEY PLAN ADJACENT ELEMENT in the Filter column of the key plan view style and then assign a graphic rule. You must use uppercase letters for the term.

• Important

Normalized key plan types do not support the use of graphic modules in their key plan view styles.

The following properties are available for the **One Volume With Plant View** key plan type:

- The Coordinate System property provides a dropdown list of all
 coordinate systems in the model. It also includes the value Use the
 Coordinate System of Related View. Selecting this value will cause the
 key plan to use the same coordinate system as the associated drawing
 view.
- The **Range Filter** property specifies the range to display beyond the associated volume. Click **More** to display the **Select Filter** dialog box.
- The **Orientation** property provides a list of orientations available for the key plan type.
- The **View Style** property specifies the style used to display the key plan graphics. Click **More** to display the **Select Key Plan View Style** dialog box. For more information, see *Define Key Plan Style Dialog Box*, page 98.

Note

To apply a style to the volume of the current drawing in the key plan, use
the term KEY PLAN FOCUS ELEMENT in the Filter column of the
key plan view style and then assign a graphic rule. You must use
uppercase letters for the term.

! Important

 One Volume With Plant View key plan types do support the use of graphic modules in their key plan view styles. For example, you can use the VolumeWireFrame.dll graphic module to prevent the volume of the current drawing from obscuring objects within it.

Related Topics

- Define a Key Plan Style, page 102
- Define Key Plan Style Command (Tools Menu), page 97
- Define Key Plan Style Dialog Box, page 98

Define a Key Plan Style

- Click Tools > Define Key Plan Style. The Define Key Plan Style dialog box appears.
- 2. Click **New** . The **Key Plan Style** dialog box appears.

Note

- To modify an existing key plan style, select an existing style in the Rule Name list and click Properties.
- 3. Specify a name and description for the new key plan style.
- 4. Select a **Key Plan Type**. The list of available properties updates.
- 5. Select **More** for the Key Plan View Style property to display the **Select View Style** dialog box.

- 6. Select the **View Style Type**.
- 7. Click **New** to display the **View Style Properties** dialog box.
- 8. In the **View Style Name** box, type the style name.
- 9. In the **Description** box, type a description of the style.
- 10. In the **Filter Name** column, select **More** in the dropdown list to choose an existing filter or to create a new filter.

Note

- To apply a style to the volume of the current drawing in the key plan, use the term **KEY PLAN FOCUS ELEMENT** in the **Filter** column of the key plan view style and then assign a graphic rule. You must use uppercase letters for the term.
- 11. In the **Graphic Rule** column, select an existing graphic rule or create a new graphic rule by clicking **More**.
- 12. In the **Label Rule** column, select a labeling rule, if applicable.
- 13. In the **Dimension Rule** column, select a dimension rule, if applicable.
- 14. Select a north arrow label, if applicable.

Notes

- A single drawing view can have multiple key plans.
- To delete a key plan style, select it and click **Delete**.
- To rename a key plan style, select it and click **Rename**.

Related Topics

• Define Key Plan Style Command (Tools Menu), page 97

View in 3D Command

Displays objects from the drawing in a three-dimensional graphic view.

The **View in 3D** command enables the selection capability in the **Workspace Explorer**. For more information, see *Workspace Explorer Command (View Menu)*, page 26.

Related Topics

- View Items in 3D, page 104
- Volume Drawings: An Overview, page 161

View Items in 3D

- 1. In the **Detail View**, right-click a drawing, then select **Open** on the shortcut menu.
- 2. On the toolbar, click **View in 3D** . The software opens a graphic view to show the items in the drawing.

The **View in 3D** command enables the selection capability in the **Workspace Explorer**. For more information, see *Workspace Explorer Command (View Menu)*, page 26.

Notes

- You can also click **Tools** > **View in 3D** when a drawing is open.
- You can click **Window > Tile Horizontally** or **Window > Tile Vertically** to display the graphic view, drawing, and **Detail View** all at once.

- Orthographic Drawings by Query: An Overview, page 181
- Piping Isometric Drawings by Query: An Overview, page 193
- Volume Drawings: An Overview, page 161

Save As Command

Saves drawings and reports as specified file types to an external location, such as a share on another server. This command is not available until you generate drawings for at least one of the structures in the hierarchy. This command saves only the structures that contain drawings or reports. You can save multiple file types based on the types of documents available. You can specify the target file type for each drawing type you want to save.

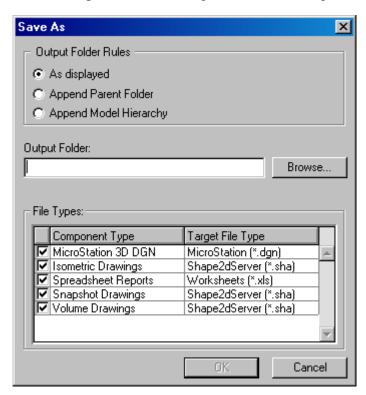
To save the hierarchy as a package, right-click the folder and select **Save Package** on the right-click menu. For more information, see *Save Package Command*, page 38.

Related Topics

- Save As Dialog Box, page 105
- Save As MicroStation or AutoCAD Format, page 120
- Save to a File, page 107
- Snapshot Drawings: An Overview, page 153
- Volume Drawings: An Overview, page 161

Save As Dialog Box

Sets options for exporting drawings. You can open this dialog box by right-clicking a folder, component, or drawing and then selecting **Save As** on the right-click menu.



Output Folder Rules - Specifies the how you want to save the hierarchy. The following options are provided:

- **As displayed** Specifies the selection is saved as displayed.
- **Append Parent Folder** Specifies the selected item is appended to the parent folder.
- **Append Model Hierarchy** Specifies the selected item is appended to the hierarchy starting from the root and including the selected item.

Output Folder - Specifies the location to which to save the package.

Browse - Indicates a folder in which to save the drawings. You can select a local folder or a folder on another computer on the network.

File Type - Specifies the file formats to save for each drawing type. Check the box next to each drawing type you want to save. Use the **Target File Type** dropdown to specify the file type to which the drawing type is saved. The file types available for each drawing type are described in the following table.

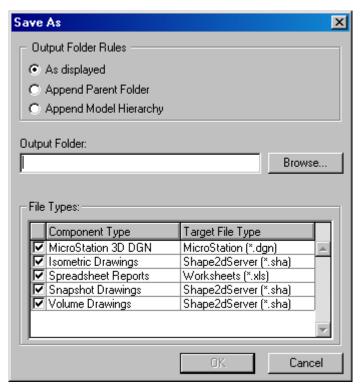
Drawing Type	Target File Types
MicroStation	MicroStation (*.dgn)
Isometric Drawings	Shape2DServer (*.sha)
	PCF file (*.pcf)
	Both (*.sha & *.pcf)
	All Files (*.*) - Includes all .sha and .pcf files, as well as all enabled supplementary files.
	MicroStation (*.dgn)
	AutoCAD (*.dxf)
	AutoCAD (*.dwg)
Spreadsheet Reports	Worksheets (*.xls)
Snapshot Drawings, Volume	Shape2DServer (*.sha)
Drawings, and Orthographic Drawings by Query	MicroStation (*.dgn)
	AutoCAD (*.dxf)
	AutoCAD (*.dwg)

- Save As Command, page 105
- Save to a File, page 107

Save to a File

Prior to following this procedure, you must have generated drawings already for at least one of the items in the **Management Console** hierarchy. The **Save As** command is not available if drawings have not been generated.

- Select a folder or component in the Management Console or documents in the Detail View. You can select multiple documents by holding Ctrl or Shift and clicking each item.
- 2. Right-click your selection, then select **Save As** on the shortcut menu. The **Save As** dialog box displays.



- 3. Specify the **Output Folder Rule** to be used. You can save the item as it appears in the **Management Console**, with its parent folder appended or with the entire model hierarchy appended.
- 4. Specify the **Output Folder** location. Click **Browse** to display a dialog box to locate the appropriate folder location.
- 5. Check the boxes for the **Component Types** you want to save. You can select multiple component types. For more information, see *Save As Dialog Box*, page 105.
- 6. In the **Target File Type** dropdowns, specify the file types you want to save. You can specify a file type for each component type selected. For example, you could use the Iso_Stress style to create a Piping Component File (PCF) file, then when you perform a **Save As** on the document, check the **Isometric Drawings** component type and specify the **Target File Type** as **PCR File**.

7. Click **OK** to save the files as specified.

Note

The saved drawings retain the same names they had in this task.

- Create a Piping Component File (PCF), page 221 Save As Command, page 105

Dimensions: An Overview

In general, dimension rules control the placement of dimensions while dimension styles control the appearance, including the units, of dimensions in orthographic drawings. However, dimension styles and dimension rules interact in complex ways. There are three methods you can use to get dimensions to appear in drawings. The first two methods, *automatic dimensioning* and *manual dimensioning*, place dimensions in native format drawings. The third method uses the **Save As** command to save drawings from the database to files. The files can be native format or a foreign format such as MicroStation or AutoCAD.

For *automatic dimensioning*, the view style controls whether or not dimensions are placed. For *manual dimensioning*, you edit an existing drawing and place dimensions manually. With the **Save As** command, the software attempts to replicate dimensions as they are shown within a file.

Assigning dimension units is different for each method. For more information, see:

Automatic Dimensioning, page 110
Manual Dimensioning, page 119
Save As MicroStation or AutoCAD Format, page 120

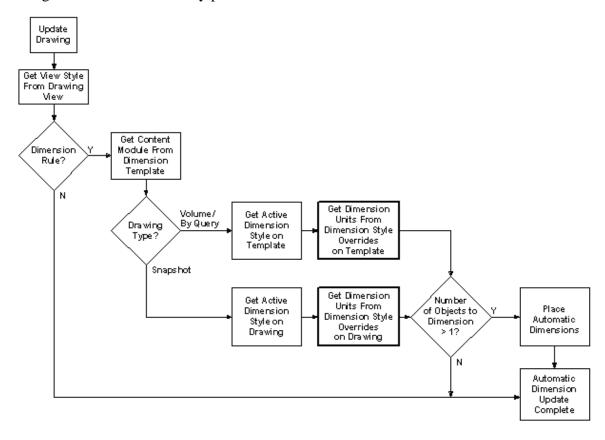
Note

• Isometric drawings use the isometric options settings within their style to determine the dimension appearance, placement, and units. For more information, see *Customizing Isometric Drawing Styles: An Overview*, page 207.

- Define View Style Command (Tools Menu), page 54
- Understanding the View Style Rules: An Overview, page 76

Automatic Dimensioning

When you use *automatic dimensioning*, the view style controls whether or not dimensions are placed. The following flowchart shows how the **Update** command assigns units to automatically placed dimensions.



View Styles

A dimension rule with a view style triggers the automatic placement of dimensions in an orthographic drawing. For more information, see *Use View Styles with Dimension Rules*, page 111.

Dimension Rules

Dimension rules control the placement and appearance of automatic dimensions in the drawing. For more information, see *Use Dimension Rules*, page 112.

Dimension Templates

The dimension rule is not responsible for assigning the dimension units to automatic dimensions. The dimension rule points to a dimension template XML file that influences the unit assignment. For more information, see *Use Dimension Templates*, page 112.

Dimension Styles

You can determine the active style of a dimension by editing the drawing or drawing template in the 2D Drawing Editor. For more information, see *Edit Dimension Styles*, page 114.

Dimension Style Overrides

You can override dimension style settings in the 2D Drawing Editor. For more information, see *Override Dimension Styles*, page 117.

Related Topics

• Dimensions: An Overview, page 109

Use View Styles with Dimension Rules

A dimension rule with a view style triggers the automatic placement of dimensions in a drawing.

- 1. Select **Tools > Define View Style**.
- 2. On the **View Style** dialog box, select **New** to define a new view style with a dimension rule. You can also edit an existing view style by selecting **Properties**.
- 3. On the **View Style Properties** dialog box, use the **Dimension Rule** field dropdown to specify a dimension rule for the view style. For more information on this dialog box, see *View Style Properties Dialog Box*, page 57. Select **More** to display the **Select Dimension Rule** dialog box. For more information, see *Select Dimension Rule Dialog Box*, page 66.



For information on how dimension rules are maintained, see *Use Dimension Rules*, page 112.

Note

• For more information on defining view styles, see *Define View Style Command (Tools Menu)*, page 54.

- Automatic Dimensioning, page 110
- Dimensions: An Overview, page 109

Use Dimension Rules

A dimension rule controls the placement and appearance of automatic dimension in the drawing. When you select **More** in the **Dimension Rule** dropdown on the **View Style Properties** dialog box, the **Select Dimension Rule** dialog box displays the list of rules available in the Drawings catalog. For more information, see *Select Dimension Rule Dialog Box*, page 66.

The Drawings catalog is file-based and located on the Symbols share within the \\Drawings\Catalog\ folder. The available dimension rules are XML files stored in the \\Drawings\Catalog\ Rules\DimensionRules\ folder. Several example dimension rules are delivered with the software.

You edit the dimension rule XML files with a text or XML editor. Each dimension rule should point to its own dimension template. Rename any customized dimension rules. Do not use the delivered rule names for customized rules.

! Important

- We recommend that you maintain a separate set of dimension rules for each dimension style used in drawings.
- The dimension rule is not responsible for assigning the dimension units to automatic dimensions. However, the dimension rule points to another XML file, called the *dimension template*, that influences the unit assignment. For more information, see *Use Dimension Templates*, page 112.
- The drawing dimension XML Files are discussed further in the *SmartPlant 3D/IntelliShip Programmer's Guide* under *Extending the Capabilities of the Software*. Contact your administrator or Intergraph Support if you need the *Programmer's Guide*.

Related Topics

- Automatic Dimensioning, page 110
- Define View Style Command (Tools Menu), page 54
- Dimensions: An Overview, page 109

Use Dimension Templates

The dimension rule is not responsible for assigning the dimension units to automatic dimensions. However, the dimension rule points to another XML file, called the *dimension template*, that influences the unit assignment. The dimension template XML file contains the settings that further control the placement and appearance of dimension in the orthographic drawings.

You edit the dimension template XML files with a text or XML editor. Rename any customized dimension templates. Do not use the delivered rule names for customized dimension templates.

Several example dimension template XML files are delivered with the software and are located on the Symbols share in the $\Drawings\Catalog\Dimensions\Templates$ folder.

The setting in the template that influences dimension unit display is <dimensionContentModules>. In each dimension template, you should set <dimensionContentModules> to the dimension style used to create dimensions in the drawing. The values available for this setting determine whether the software places the dimension vertical, horizontal, radial, or angular. To determine the dimension units, the various content modules refer to the dimension formatting saved either in the drawing template file (for volume and *by query* drawing types) or in the drawing itself (for snapshot drawings).

! Important

- We recommend that you maintain a separate dimension template for each dimension rule you create.
- For *manual dimensioning*, the software looks in the **Linear_A_HV** dimension template for the value of the dimension content module during the update of manually placed dimensions. For more information on *manual dimensioning*, see *Manual Dimensioning*, page 119.
- The **<dimensionStyleSettings>** setting in the dimension templates is not used by the software at this time. However, when creating drawings, we recommend that the active dimension style in the selected border template have the same name as the **<dimensionStyleSettings>** setting in the dimension template to avoid problems in future software releases. Rename any customized border templates. Do not use the delivered border template name for customized border templates.
- The drawing dimension XML Files are discussed further in the *SmartPlant 3D/IntelliShip Programmer's Guide* under *Extending the Capabilities of the Software*. Contact your administrator or Intergraph Support if you need the *Programmer's Guide*.

- Automatic Dimensioning, page 110
- Dimensions: An Overview, page 109
- Use Dimension Rules, page 112

Edit Dimension Styles

The software resolves dimension units for a particular drawing from the active dimension formatting in either the drawing template (for volume and *by query* drawing types) or the drawing itself (for snapshot drawings). The active dimension formatting is a combination of the settings in the active dimension style plus any dimension style overrides that may be set.

You can determine the active style by editing the drawing or the drawing template in the 2D Drawing Editor, selecting one of the placement dimension commands, and checking the style that appears on the resulting ribbon bar. You can view the overrides to the active dimension in the drawing template by selecting **Format** > **Dimension** in the 2D Drawing Editor.

! Important

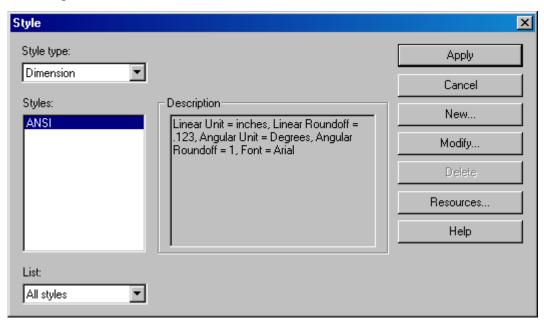
• All of the delivered Imperial drawing types have ANSI as the active dimension style with no overrides set. For more information on the Imperial drawing types, see *Imperial Drawing Types: An Overview*, page 149. The **Imperial** border templates have ANSI as the active dimension style with no overrides set. The **Metric** border templates have DIN as the active dimension style with no overrides set.

You can create and maintain dimension styles with the 2D Drawing Editor.

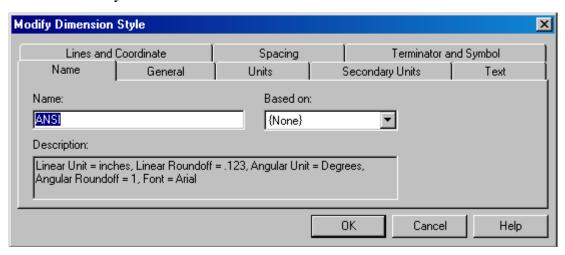
1. In the 2D Drawing Editor, select **Format > Style**.



2. As the **Style** dialog box indicates, dimension styles control the appearance, including the units, of the dimension.



- 3. Click Modify.
- 4. On the **Modify Dimension Style** dialog box, you can modify the appearance of the dimension style as needed.



- 5. Click **OK** to save any changes you made to the dimension style.
- 6. On the **Style** dialog box, you can either save dimension styles with the files being edited or reference them from other files. You can reference files by selecting Resources.

? Tips

- The reference files, also called *resource files*, can be any file with an .igr or .sha extension. While referenced dimension styles are available for dimension placement, they cannot be edited. Therefore, the **Style** dialog box only displays dimension styles that are saved locally to the file.
- In cases where a local dimension style has the same name as a referenced dimension style, the software always uses the local style when placing dimensions. Currently, you cannot override local dimension styles with the same name as a referenced dimension style of the same name. Local dimension styles can be renamed so that the referenced dimension style can be used to place dimensions. Local dimension styles are renamed on the **Modify Dimension Style** dialog box. For more information on overriding dimension style settings, see *Override Dimension Styles*, page 117.

The delivered *Styles.sha* file contains several example dimension styles. This file is located in the Symbols share in the \Drawings\Catalog\Templates folder. All drawings created in the Drawings and Reports task reference the *Styles.sha* file.

• Important

• After updating the drawings, the *Styles.sha* file on the Symbols share is the only file referenced into the drawings. This is true for all orthographic drawings. Any other files previously referenced before the update need to be referenced to the drawing again.

All drawings and drawing templates must have at least one dimension style saved within them. Therefore, if only one dimension style is saved with a file, you cannot delete it. Also, you cannot delete any style from a drawing or drawing template if they are currently used by dimensions in a drawing.

Dimension styles can be added to the list of saved, or local, styles in a drawing or drawing template in one of two ways:

- Create a new dimension style from the Style dialog box using the New command.
- Place a dimension in a drawing or drawing template using a reference dimension style. This method copies the referenced dimension style into the drawing or drawing template.

! Important

- We recommend that you maintain a separate set of border templates for each dimension style used for drawings. The border templates within a particular set should have the same active dimension style with the same style settings.
- The **<dimensionStyleSettings>** setting in the dimension templates is not used by the software at this time. However, when creating drawings, we recommend that the active dimension style in the selected border template have the same name as the **<dimensionStyleSettings>** setting in the dimension template to avoid problems in future software releases. Rename any customized border templates. Do not use the delivered border template name for customized border templates.
- We recommend that you maintain a complete set of your dimension styles in a renamed version of the *Styles.sha* file.

Related Topics

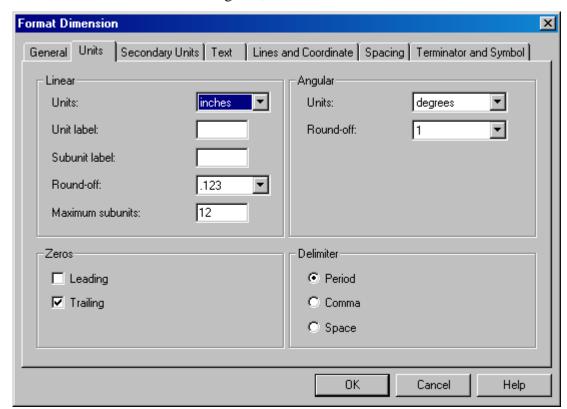
- Automatic Dimensioning, page 110
- Dimensions: An Overview, page 109
- *Use Dimension Rules*, page 112

Override Dimension Styles

You can override dimension styles in a drawing or drawing template.

! Important

- We support dimension style overrides, but we do not recommend their use as a standard practice. Overrides are saved in the drawing or drawing template and can be difficult to maintain as dimension styles change. As an alternative, we recommend creating a new dimension style in a reference resource file. For more information, see *Edit Dimension Styles*, page 114.
- 1. Select **Tools > Edit Border Template** to edit the template. The template opens in the 2D Drawing Editor.
- 2. Select Format > Dimensions.



3. On the **Format Dimension** dialog box, select the **Units** tab.

4. If the settings on the **Units** tab are identical to those on the **Modify Dimension Style** dialog box (for more information, see *Edit Dimension Styles*, page 114), no overrides are applied to the active dimension style. If a dimension style is changed during a dimension placement command, the override values become identical to the dimension style values.

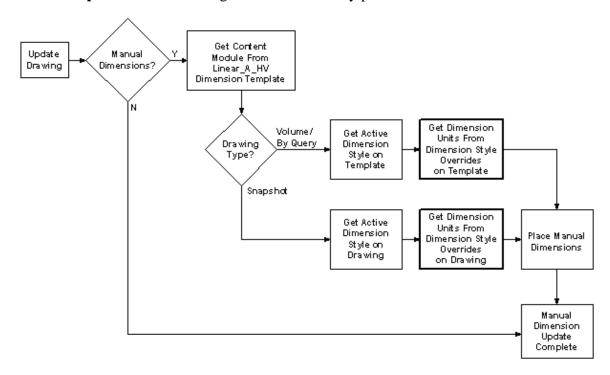
Note

You cannot override local dimension styles with the same name as a
referenced dimension style. Local dimension styles can be renamed so that
the referenced dimension style can be used to place dimensions. Local
dimension styles are renamed on the Modify Dimension Style dialog box.

- Automatic Dimensioning, page 110
- Dimensions: An Overview, page 109
- Use Dimension Rules, page 112

Manual Dimensioning

When you use *manual dimensioning*, the view style and the dimension rules do not apply for display or placement. However, the logic used to determine manual dimensions is similar to that of *automatic dimensions*. The following flowchart shows how the **Update** command assigns units to manually placed dimensions.



Dimension Templates

For *manual dimensioning*, the software looks in the **Linear_A_HV** dimension template for the value of the dimension content module during the update of manually placed dimensions. Aside from this, the behavior is the same as for *automatic dimensioning*. For more information, see *Use Dimension Templates*, page 112.

Dimension Styles

You can determine the active style of a dimension by editing the drawing or drawing template in the 2D Drawing Editor. For more information, see *Edit Dimension Styles*, page 114.

Dimension Style Overrides

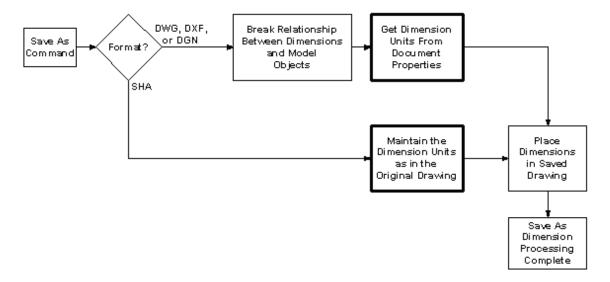
You can override dimension style settings in the 2D Drawing Editor. For more information, see *Override Dimension Styles*, page 117.

Related Topics

• Dimensions: An Overview, page 109

Save As MicroStation or AutoCAD Format

You can use the **Save As** command to assign dimension units and export the drawings to MicroStation or AutoCAD format. The following flowchart shows how the **Save As** command controls dimensioning during export.



Save As Command

The **Save As** command is available from the shortcut menu for any document or node containing documents in the **Management Console**. For orthographic drawings, the **Save As** command supports exporting to DGN, DWG, and DXF formats, as well as the native SHA formats. For more information, see *Save As Command*, page 105.

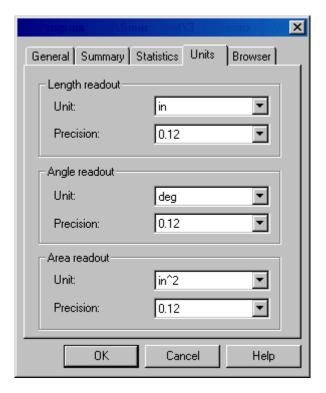
Embedded Object Dimensioning

The model graphics in the drawings are embedded in the drawing file. The dimensions in the drawings are *connected* directly to the embedded objects.

During export to MicroStation or AutoCAD formats using **Save As**, the software opens the drawings in the 2D Drawing Editor and the relationship between the embedded objects and the dimensions is broken. The dimension, however, is not removed and still maintains its original value.

Document Properties

Dimensions in drawings exported to the DGN, DWG, and DXF formats do not use the active dimension formatting for their units. Instead, the software determines dimension units from the document properties for the drawing being exported. This behavior applies whether the drawings is a volume, *by query*, or snapshot. You can view and edit the document units by opening the drawing in the 2D Drawing Editor and selecting **File > Properties**. You can see the current unit settings on the **Units** tab.



! Important

 We recommend that border templates have identical units and precision values in the document properties set in the active dimension style. This maintains dimension unit accuracy if you export the drawing to a foreign file format.

Related Topics

• Dimensions: An Overview, page 109

Template Commands: An Overview

Several commands are provided to help customize your drawing templates. These commands are found in the 2D Drawing Editor environment when editing a drawing template. The availability of the Template Commands depends on the template being edited. For example, the **Manual Place Labels** command is only available when a drawing has been created. It is not available when you use **Tools** > **Edit Border Template**.

To use these commands, select **Tools > Edit Border Template**, or right-click a drawing document and select **Edit**.

The commands are:

Place Drawing Property Label Command (Drawing Labels Toolbar), page 123
Place Drawing View Command (Template Toolbar), page 128
Place Report Command (Template Toolbar), page 136
Place Key Plan Command (Template Toolbar), page 140
Manual Place Labels Command (Manually Place Labels Toolbar), page 144

Place Drawing Property Label Command (Drawing Labels Toolbar)

Positions drawing property labels in the title block of a template. The command allows you to place drawing properties defined in the drawing XML schema file as title block information on the drawing. You access this command by selecting **Tools** > **Edit Border Template** to display a template in the 2D Drawing Editor. The Place Drawing Property Label appears in the toolbar area.

Related Topics

- Edit Border Template Command (Tools Menu), page 96
- Place a Custom Drawing Property Label on a Template, page 125
- Place a Drawing Property Label on a Template, page 124
- Properties Command, page 322

Place Drawing Property Label Ribbon

Sets options for drawing property label placement on a border template. You can access this ribbon when you click the **Place Drawing Property Label** command when editing a border template in the 2D Drawing Editor.

When you place drawing property labels, the software automatically makes the **DwgTemplate** layer active. The labels need to be on this layer so that they are preserved when you update the drawing.



You set options as needed within the **Place Drawing Property Label** ribbon, then place the label in the drawing border title block area.



Label Set - Specifies a property category. This list shows the categories of drawing properties available for the current drawing. The label set controls the fields listed in the **Field** dropdown list and the enabling of other options on the ribbon.

Fields - Lists the properties available in the selected **Label Set**. This is the information you are placing on the title block of the drawing.

Function - Provides positioning functions for the label. The options available are **Index**, **First**, and **Last**. This control works with the **Function Operator** and **Function Argument** fields to set the position of the label within the title block area.

Function Operator - Works with the **Function** and **Function Argument** fields to set the position of the label within the title block area. The value is controlled by the **Function** selection. This field is not editable.

Function Argument Sets a value to add or subtract from the **First** or **Last** settings in the **Function** field. This field works also works with the **Function Operator** field to set the position of the label within the title block area.

Alternative Text Value - Specifies alternative text to use if the selected property has no current value.

More - Expands the ribbon to include additional formatting controls.



The controls on the expanded ribbon include:

Style - Sets the overall style used within the label.

Font - Sets the font for the label text.

Font Size - Sets the font size for the label text.

Textbox Width - Specifies the width of the text box.

Related Topics

- Edit Border Template Command (Tools Menu), page 96
- Place a Custom Drawing Property Label on a Template, page 125
- Place a Drawing Property Label on a Template, page 124
- Place Drawing Property Label Command (Drawing Labels Toolbar), page 123

Place a Drawing Property Label on a Template

If you want to place a custom attribute label on a template, see *Place a Custom Drawing Property Label on a Template*, page 125.

- 1. Click **Tools > Edit Border Template**.
- 2. On the **Select Template** dialog box, select a template, and click **OK**. The template opens in the **2D Drawing Editor**.



 Always place drawing property labels in the **DwgTemplate** layer of the template. Select **Tools** > **Layers**. The **Layer** ribbon appears so you can change the layer if necessary.



3. On the toolbar, click **Place Drawing Property Label** . The **Place Drawing Property Label** ribbon appears.



- 4. In the **Label Set** list, select a label set (set of drawing properties). The list reflects the label sets within the drawing XML schema. The **Label Set** selection controls the contents of the **Fields** list and the enabling of other controls on the ribbon.
- 5. In the **Fields** list, select a field to use as your title block label.
- 6. Specify a Function, setting the Function Argument if needed.
- 7. Provide alternative text for cases when the label property could be blank.
- 8. Click **More** at to expand the ribbon and set formatting options.



You can set options for Style, Font, Font Size, and Textbox Width.

- 9. Click the template to place the label. The label follows your cursor until you click to place the label.
- 10. Continue placing labels on the template as necessary.
- 11. Right-click to end the command.
- 12. Save the changes to the template before closing the **2D Drawing Editor**.

Related Topics

- Place Drawing Property Label Command (Drawing Labels Toolbar), page 123
- Place Drawing Property Label Ribbon, page 123

Place a Custom Drawing Property Label on a Template

The following steps show how to add a custom attribute drawing property label to a drawing template. To add non-custom property labels to the template, see *Place a Drawing Property Label on a Template*, page 124.

- 1. Create a Custom Attribute workbook using Excel. This is the bulkload file for the custom attribute and names the attribute. For example, you could create a file called *CustomAttributes.xls* that contains the definition for a custom attribute.
- 2. Bulkload the Custom Attribute workbook. For more information on populating Excel workbooks and bulkloading, see the *SmartPlant 3D Reference Data Guide* available from **Help > Printable Guides**.
- 3. Create an .xsd file and add a line that defines the attribute in the dropdown list when you edit a template and use the **Place Drawing Property Label** command. For example, using **DrawingCustom1** as the attribute name, the line would be <xs:element name="DrawingCustom1">.

4. Add a line that points to the **AttributeName** property of the bulkloaded attribute. For example, using the above attribute name, the necessary line would be *<pk* name="DrawingCustom1"/>. Your .xsd file would look similar to the following:

- 5. Make sure the .xsd file is in the \Symbols\Drawings\Catalog\Labels\Border\Schema folder.
- 6. In the Drawings and Reports task, click **Tools > Edit Border Template**.
- 7. On the **Select Template** dialog box, select a template, and click **OK**. The template opens in the **2D Drawing Editor**.



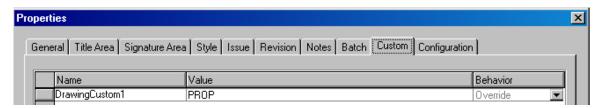
 Always place drawing property labels in the **DwgTemplate** layer of the template. Select **Tools** > **Layers**. The **Layer** ribbon appears so you can change the layer if necessary.



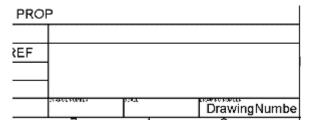
- 8. In the **Label Set** list (the first dropdown on the ribbon), select **Custom**.
- 9. In the **Fields** list (the second dropdown on the ribbon), select the Custom attribute property to use as your title block label.



- 10. Click the template to place the label. The label follows your cursor until you click to place the label.
- 11. Continue placing labels on the template as necessary.
- 12. Right-click to end the command.
- 13. Save the changes to the template before closing the **2D Drawing Editor**.
- 14. Update any drawing documents associated with the modified template. For more information, see *Updating Documents: An Overview*, page 337.
- 15. Right-click the drawing and select **Properties**.
- 16. Go to the **Custom** tab. The new Custom attribute property is shown on the tab.



17. To verify the custom drawing property is added to the drawing title block, rightclick the drawing and select **Open**. The custom drawing property includes the Custom Property attribute in the title block.



- Custom Tab (Properties Dialog Box), page 333
- Place Drawing Property Label Command (Drawing Labels Toolbar), page 123
- Place Drawing Property Label Ribbon, page 123

Place Drawing View Command (Template Toolbar)

Places a two-dimensional view for orthographic drawings. Drawing views contain the two-dimensional representation of items in the model. Both volume and snapshot drawings have drawing views. You use the **Place Drawing View** command in the **2D Drawing Editor** to place views.

When you place drawing views, make sure you are on the **DwgTemplate** layer of the drawing so the software preserves the drawing views when you update the drawing.



Volume Drawing Views

For volume drawing views (including Orthographic Drawings By Query), you must enter at least a name for the view on the **Drawing View Properties** dialog box. You also can enter a description, view style, orientation, and scale for the view.

Volume drawings can have more than one drawing view per template. The drawing views are associated with one drawing volume. You can create volumes by switching to the Space Management task and clicking one of the **Place Drawing Volume** commands.

Snapshot Drawing Views

For snapshot drawing views (Snapshot Components), you use the **Tools > Drawing View** command in one of the 3D tasks (such as Common) to create drawing views, assigning them to a snapshot component with a view name, naming rule, and style. For more information, see the *Common User's Guide* available from **Help > Printable Guides**. After you create all the views for a particular snapshot drawing component, the **Place Drawing View** command becomes unavailable when you use the **Create Drawings** command for the snapshot drawing component.

Snapshot drawings can have multiple views placed on them. The number of snapshot views you can place is determined by the number of drawing views specified for the snapshot component.

View Styles

Each view has a view style that controls the appearance of items in that view. For volume drawings, you can select a view style on the **Drawing View Properties** dialog box. For snapshot drawings, you can select a view style on the **Tools** > **Drawing View** ribbon.

When you want to update drawing views to synchronize with the model, you can click the **Refresh** and **Update** commands in the **Management Console**.

Related Topics

- Place a Drawing View for Snapshot Drawings, page 133
- Place a Drawing View for Volume Drawings, page 131

Drawing View Properties Dialog Box

Sets options for placing a view on a template for drawings. These options include the name, description, and style.

Related Topics

- Place a Drawing View for Volume Drawings, page 131
- Place Drawing View Command (Template Toolbar), page 128

Info Tab (Drawing View Properties Dialog Box)

Provides information about the frame around a drawing view. This information is view-only. You cannot make changes.

Type - Displays the category of the selected element.

Sheet - Displays the name of the drawing sheet that contains the selected element.

Layers - Shows the layer that contains the selected element.

Origin - Specifies the coordinates, or location, of an element along the X- and Y-axes.

Related Topics

- Drawing View Properties Dialog Box, page 129
- Place a Drawing View for Volume Drawings, page 131
- Place Drawing View Command (Template Toolbar), page 128

Format Tab (Drawing View Properties Dialog Box)

Formats the frame around a drawing view.

Show Border - Displays the frame around the object.

Color - Sets the color of the frame.

Line Width - Sets the line thickness on the frame.

Line Type - Overrides a line type for a drawing sheet or embedded object and sets another line style for an element or linked object.

Related Topics

- Drawing View Properties Dialog Box, page 129
- Place a Drawing View for Volume Drawings, page 131
- Place Drawing View Command (Template Toolbar), page 128

View Tab (Drawing View Properties Dialog Box)

Sets the drawing view style and other properties for a selected drawing view.

Name - Specifies a name for the view. You must type a name in order to create a view.

Description - Describes the content of the view. This description is optional.

Style - Specifies a view style, which includes rules for filters, updates, and graphics. The view style controls the output characteristics of the view on the generated drawing.

Orientation - Positions the view in the model. For example, the view can look north, south, east, or west in the model.

Use object coordinate system - Specifies that the drawing takes its viewing direction directly from the object within the drawing view, not from the orientation of the drawing view.

Note

• This option is not available for snapshot drawings.

Scale - Sizes the view as a ratio of drawing size to actual model size. If you select **Custom** for the scale, you must type values that are greater than zero in the boxes at the right to set the scale-to-scale ratio for object-to-drawing view.

Notes

- For **Custom** scale, the default is to the document's unit of measure setting. For example, if you enter values of **1** in to **1** ft, the values are converted to **mm** if that is the default unit of measure.
- Do not use negative values when entering custom scale values.
- For volume drawings, if the volume is *too big* for the drawing view, the software centers the volume in the drawing view.

- Drawing View Properties Dialog Box, page 129
- Place a Drawing View for Volume Drawings, page 131
- Place Drawing View Command (Template Toolbar), page 128

Manual Place View Ribbon

Sets options for placing a view on a snapshot drawing. These options include the view name, style, and scale.

View Name - Displays the name of the view you created in a 3D task such as Common.

View Style - Displays the style for the view.

View Scale - Specifies a scale. You can choose **No Scale** if the view is not to scale. Choose **Custom** if you want to specify your own scale and then type values in the two boxes at the right of the ribbon. If you choose **Custom** for the scale, you must type values that are greater than zero in the boxes at the right.

Related Topics

- Place a Drawing View for Snapshot Drawings, page 133
- Place a Drawing View for Volume Drawings, page 131
- Place Drawing View Command (Template Toolbar), page 128

Place a Drawing View for Volume Drawings

Before placing a drawing view for a volume drawing, you must have a **Volume View Style**. For more information, see *Create a View Style*, page 66.

- 1. Right-click a volume drawing component in the **Management Console**, and then click **Edit Template** on the shortcut menu.
- 2. Select a template on the **Select Template** dialog box and click **OK**. The drawing template opens in the 2D Drawing Editor.



• Always place drawing views in the **DwgTemplate** layer of the template. Select **Tools** > **Layers**. The **Layer** ribbon appears so you can change the layer if necessary.

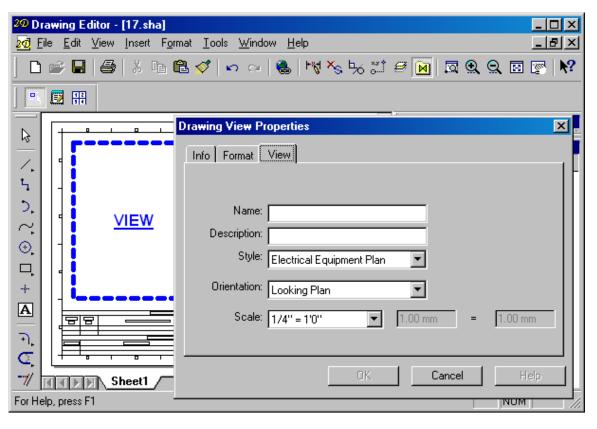


3. Click **Place Drawing View** on the toolbar.

4. Click a point on the drawing sheet to define the first corner of a rectangle, and hold the mouse button down while dragging the mouse diagonally. Release the button at a second point.



The drawing view highlights on the drawing and the **Drawing View Properties** dialog box displays.



- 5. On the **Drawing View Properties** dialog box, enter the necessary information on the **View** tab in the **Name** and **Description** boxes.
- 6. In the **Style** box, select a view style.
- 7. In the **Orientation** box, select an orientation such as **Looking Plan**.
- 8. In the **Scale** box, select a scale, or choose **Custom** and define the values in the boxes to the right.

? Tips

- If you choose Custom, you must type values that are greater than zero in the boxes at the right. For example, if you enter values of 1 in to 1 ft, the values are converted to mm if that is the default unit of measure.
- Do not use negative values when entering custom scale values.
- 9. Specify additional information as necessary on the **Format** tab.
- 10. Click **OK**.
- 11. Click **File > Save** and exit the 2D Drawing Editor.

Notes

- To associate volumes with the view, switch to the Space Management task and click one of the **Place Drawing Volume** commands.
- You can resize a drawing view by dragging its handles.

Related Topics

• Place Drawing View Command (Template Toolbar), page 128

Place a Drawing View for Snapshot Drawings

Before using the following procedure, you must create at least one drawing view in a 3D task using the **Tools > Drawing View** command. When you create the drawing in the Drawings and Reports task, the drawing opens and you can continue with the procedure.

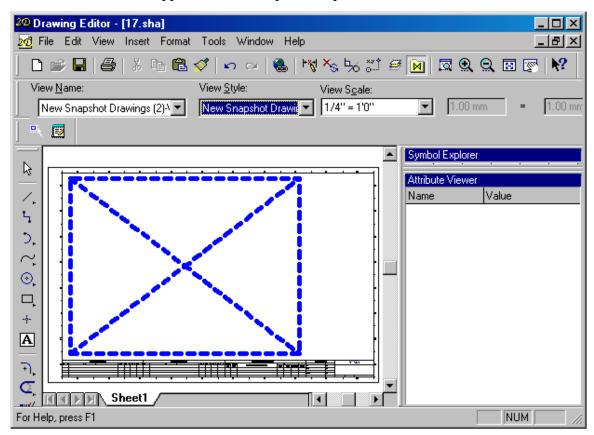
You also must have a **Snapshot View Style** defined to place a drawing view in a snapshot drawing. For more information, see *Create a View Style*, page 66.

- 1. Right-click a snapshot drawing component and select **Create Drawing**. The **Select Template** dialog box displays.
- 2. Select a template and click **OK**. The drawing template opens in the 2D Drawing Editor.



 Always place drawing views in the **DwgTemplate** layer of the template. Select **Tools > Layers**. The **Layer** ribbon appears so you can change the layer if necessary.





→ Tip

- You can use 2D Drawing Editor tools, such as PinPoint, to position the view more accurately in the sheet. The **Align** command on the **Change** toolbar helps position views with regard to each other or other elements on the sheet.
- 4. Click to position the view.
- 5. On the ribbon, select a view in the **View Name** box. This name is the text you typed when creating the drawing view in a 3D task such as Common.

→ Tip

- You can create more than one drawing view in Common for a given snapshot drawing component. Multiple view names display in the View Name box.
- 6. Select a view style in the **View Style** box.
- 7. Select a scale in the **View Scale** box. If you choose **Custom**, you can type values in the two boxes at the far right of the ribbon.

→ Tip

- If you choose **Custom**, you must type values that are greater than zero.
- 8. Click a point on the drawing sheet to place the view.

💡 Tip

- If you click **No Scale** in the **View Scale** box, you must define the size of the drawing view by click-and-drag to define the height and width of the view, and then releasing the mouse button to place the view.
- 9. Continue placing views as needed. The software automatically specifies the next view in the **View Name** box on the ribbon.
- 10. Click **File > Save** and exit the 2D Drawing Editor.

Notes

- After you place all the defined views for a snapshot drawing, the Place
 Drawing View command becomes unavailable. You can define more
 views in a 3D task such as Common by using the Tools > Drawing View
 command.
- You can update the drawings by right-clicking the snapshot drawing component in the Management Console and selecting Update Document(s).

Related Topics

• Place Drawing View Command (Template Toolbar), page 128

Place Report Command (Template Toolbar)

Embeds a report in a drawing view on a volume or snapshot drawing. You must select a drawing view on the drawing before you can embed a report.

The report queries on the items in the drawing view.

Always place reports in the **DwgTemplate** layer of the template. Select **Tools** > **Layers**. The **Layer** ribbon appears so you can change the layer if necessary.



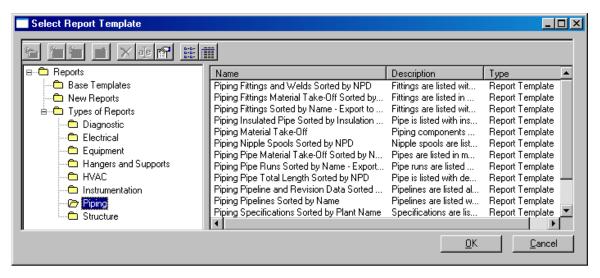
- For embedded reports to run on computers with Office XP, you must modify the security settings in Excel to allow Visual Basic projects to run. To change this setting, open Excel, and click Tools > Macros > Security. On the Trusted Sources tab, select Trust access to Visual Basic Project. This setting must be modified before you update the drawing and generate the report within it.
- When defining embedded report layout (sizing of columns and rows), consider the report usage first. Because of a Microsoft limitation concerning the size of Windows metafile objects within other applications, the data displayed may be incomplete. Therefore, no column should be out of screen when using 100 percent zoom for the report. Otherwise some columns are ignored when the report is embedded within the drawing. The same limitation exists for rows. To preserve the maximum number of rows displayed, the total header row(s) height should be a minimum of the overall report. Using Microsoft Excel default settings, the maximum number of columns is approximately 20 and the maximum number of rows is approximately 75 (including header rows). For more information on setting the defaults in Microsoft Excel, see your Microsoft Excel documentation.

Related Topics

• *Place an Embedded Report*, page 138

Select Report Template Dialog Box

Selects a report template. This dialog box appears when you click the Create Report command for a spreadsheet report component or when you click **Place Report** when a drawing template is open. By browsing through the hierarchy, you can find any report template in the Catalog database. Once you select a template, the software generates the report. You can resize the dialog box and the columns to view the information more clearly.



- **Properties** Displays the properties of the selected item. All properties on the **Properties** dialog box are read-only.
- List View Sets the dialog box to display items in a list view.
- Grid View Sets the dialog box to display items in a spreadsheet-style grid view.

Note

The Create Report and Place Report commands create reports based on a selected report template. You can also select a report template and view its properties. The buttons that are grayed out are not available when using these commands.

Related Topics

- Create and Update a Delivered Report, page 266
- Create Report Command (Report Shortcut Menu), page 262
- Spreadsheet Reports: An Overview, page 251

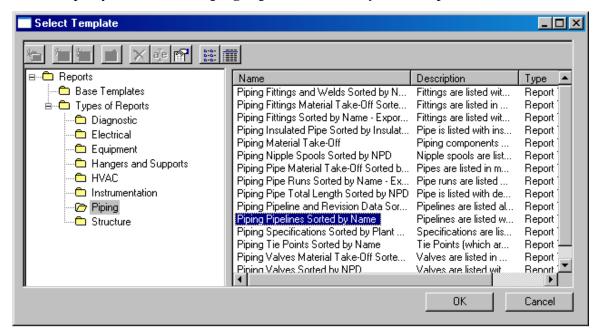
Place an Embedded Report

 For volume drawings, right-click the volume drawing component in the Management Console and select Edit Template. For snapshot drawings, rightclick a drawing in the Management Console and select Edit on the shortcut menu. The drawing template opens in the 2D Drawing Editor.

Always place reports in the **DwgTemplate** layer of the template.
 Select **Tools** > **Layers**. The **Layer** ribbon appears so you can change the layer if necessary.



- 2. Select a drawing view.
- 4. On the **Select Template** dialog box, select a report template from the hierarchy. For example, you can select **Piping Pipelines Sorted by Name** report.



- 5. Click **OK** on the dialog box.
- 6. Back in the 2D Drawing Editor, click to place the report.

Notes

- Only one report per drawing view is permitted.
- For embedded reports to run on computers with Office XP, you must
 modify the security settings in Excel to allow Visual Basic projects to run.
 To change this setting, open Excel, and click Tools > Macros > Security.
 On the Trusted Sources tab, select Trust access to Visual Basic Project.

- This setting must be modified before you update the drawing and generate the report within it.
- When defining embedded report layout (sizing of columns and rows), consider the report usage first. Because of a Microsoft limitation concerning the size of Windows metafile objects within other applications, the data displayed may be incomplete. Therefore, no column should be out of screen when using 100 percent zoom for the report. Otherwise some columns are ignored when the report is embedded within the drawing. The same limitation exists for rows. To preserve the maximum number of rows displayed, the total header row(s) height should be a minimum of the overall report. Using Microsoft Excel default settings, the maximum number of columns is approximately 20 and the maximum number of rows is approximately 75 (including header rows). For more information on setting the defaults in Microsoft Excel, see your Microsoft Excel documentation.

Related Topics

• Place Report Command (Template Toolbar), page 136

Place Key Plan Command (Template Toolbar)

Places a key plan on a volume drawing template. This command is available when you right-click a volume drawing component and select **Edit Template**. The Place Key Plan command is enabled when you select a drawing view. The command displays the **Select Key Plan** dialog box to specify a key plan type to associate with the selected drawing view.

Always place key plans in the **DwgTemplate** layer of the template. Select **Tools** > **Layers**. The **Layer** ribbon appears so you can change the layer if necessary.



A key plan is a small graphical representation of a drawing volume you defined in the geographic area where engineering is taking place. One key plan may serve for an entire project. Multiple key plans may be established per discipline. One drawing view may have multiple key plans.

You can create multiple key plan styles. For more information, see *Define Key Plan Style Command (Tools Menu)*, page 97.



• Key plans are not used with snapshot drawings.

Related Topics

• Place a Key Plan, page 142

Select Key Plan Style Dialog Box

Specifies a style for the selected key plan on a volume drawing template. Select a style from the **Rule Name** hierarchy.

Related Topics

- Key Plan Properties Dialog Box, page 140
- Place a Key Plan, page 142
- Place Key Plan Command (Template Toolbar), page 140

Key Plan Properties Dialog Box

Sets options for a key plan on a volume drawing template.

Related Topics

- Place a Key Plan, page 142
- Place Key Plan Command (Template Toolbar), page 140

Info Tab (Key Plan Properties Dialog Box)

Provides information about a key plan. This information is view-only. You cannot make changes.

Type - Displays the category of the selected element.

Sheet - Displays the name of the drawing sheet that contains the selected element.

Layers - Shows the layer that contains the selected element.

Origin - Specifies the coordinates, or location, of an element along the X- and Y-axes.

Related Topics

- Key Plan Properties Dialog Box, page 140
- Place a Key Plan, page 142
- Place Key Plan Command (Template Toolbar), page 140

Format Tab (Key Plan Properties Dialog Box)

Formats the frame around a key plan.

Show Border - Displays the frame around the object.

Color - Sets the color of the frame.

Line Width - Sets the line thickness on the frame.

Line Type - Overrides a line type for a drawing sheet or embedded object and sets another line style for an element or linked object.

Related Topics

- Key Plan Properties Dialog Box, page 140
- Place a Key Plan, page 142
- Place Key Plan Command (Template Toolbar), page 140

Key Plan Tab (Key Plan Properties Dialog Box)

Sets the properties for a selected key plan.

Name - Specifies a unique name for the key plan.

Description - Describes the key plan contents.

Style - Indicates the key plan view style used. Select **More** to display the **Select Key Plan Style** dialog box. For more information, see *Select Key Plan Style Dialog Box*, page 140.

Scale - Indicates the scale assigned to the key plan with regard to the associated drawing view. If you select **Custom** for the scale, you must type values that are greater than zero in the boxes at the right to set the scale-to-scale ratio for key planto-drawing view.

Notes

- For Custom scale, the default is to the document's unit of measure setting.
 For example, if you enter values of 1 in to 1 ft, the values are converted to mm if that is the default unit of measure.
- Do not use negative values when entering custom scale values.
- Orientation of the key plan graphic is specified as part of the key plan style. For more information, see *Define Key Plan Style Command (Tools Menu)*, page 97.

Related Topics

- Key Plan Properties Dialog Box, page 140
- Place a Key Plan, page 142
- Place Key Plan Command (Template Toolbar), page 140

Place a Key Plan

- 1. In the **Management Console**, right-click a volume drawings component, then click **Edit Template**.
- 2. If no template is defined for the volume drawings component, select a template in the **Select Template** dialog box, and click **OK**.

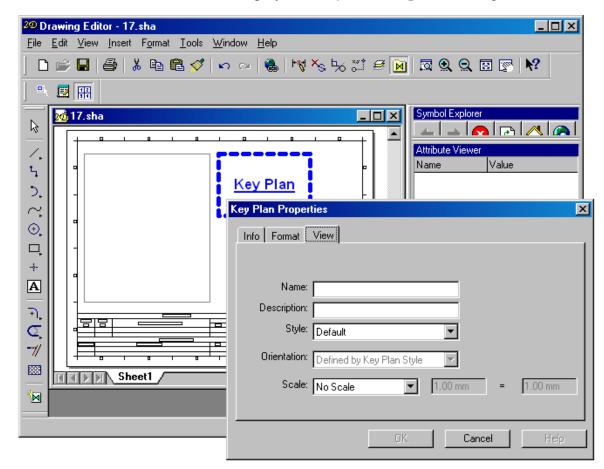


Always place key plans in the **DwgTemplate** layer of the template.
 Select **Tools** > **Layers**. The **Layer** ribbon appears so you can change the layer if necessary.



- 3. Select a volume drawing view on the template.
- 4. On the **Template** toolbar, click **Place Key Plan** [11].
- 5. Click two points to place the key plan view.





On the second click, the software displays the **Key Plan Properties** dialog box.

6. On the dialog box, specify the settings for the key plan. For more information, see *Key Plan Properties Dialog Box*, page 140.

Notes

- Multiple key plans can be associated with a single drawing view.
- For **Custom** scale, the default is to the document's unit of measure setting. For example, if you enter values of **1** in to **1** ft, the values are converted to **mm** if that is the default unit of measure.
- Do not use negative values when entering custom scale values.
- Orientation of the key plan graphic is specified as part of the key plan style. For more information, see *Define Key Plan Style Command (Tools Menu)*, page 97.

Related Topics

• Place Key Plan Command (Template Toolbar), page 140

Manual Place Labels Command (Manually Place Labels Toolbar)

Manually labels items on a drawing. Open the drawing in Edit mode and then click Manual Place Labels [85].

You can choose one of the delivered label rules, or you can customize your own label rule.

You can place labels on drawings after they have been generated. For example, you can place a label manually on a drawing to emphasize certain objects in the drawing view.

When you place labels using this command, make sure you are on the **DwgTemplate** layer of the drawing so the software preserves the manually placed labels when you update the drawing.



Related Topics

• Place a Manual Label on a Drawing, page 145

Manual Place Labels Ribbon

Sets options for label placement on a drawing. To access this ribbon, right-click a drawing and select **Edit** to open the drawing in the 2D Drawing Editor, then click **Manual Place Labels** When you place labels using this command, make sure you are on the **DwgTemplate** layer of the drawing so the software preserves the manually placed labels when you update the drawing.

Label Specification - Specifies a label rule. The label rules are located on the SmartPlant 3D server computer in the $\Symbols\Drawings\Catalog\Rules\LabelRules$ folder.

Font - Overrides the font for the text in the label.

Text size - Overrides the font size for the text in the label.

Text color- Applies a different color to text.

B Bold - Applies bold formatting to text.

I Italic - Applies the italic font to text.

Underline - Underlines the text.

- **Shape** Choose a shape for the enclosure of the label.
- Leaderline on/off Specifies whether you want a leader line pointing to the object.
- ✓ **Leaderline jog on/off** Specifies whether a jog in the leader line is acceptable.
- Orientation Specifies horizontal or vertical text in the label.

As Drawn - Specifies that the label is shown as it was designed in the label rule. Font, text size, text color, and other formatting on the ribbon do not apply.

Related Topics

- Manual Place Labels Command (Manually Place Labels Toolbar), page 144
- Place a Manual Label on a Drawing, page 145

Place a Manual Label on a Drawing

- 1. Right-click a drawing, and click **Edit** on the shortcut menu. The drawing opens for editing in the 2D Drawing Editor.
- 2. Select **Tools > Layers** and change to the **DwgTemplate** layer.



3. On the toolbar, click Manual Place Labels.



- 4. Click an object in a drawing view to label.
- 5. In the **Label Specification** box, select a label rule.

💡 Tip

- The label rules are located on the SmartPlant 3D server in the \Symbols\Drawings\Rules\LabelRules folder.
- 6. In the **Font** box, specify the font.
- 7. In the **Text size** box, specify the size of the text.
- 8. Click **Text Color**, and select a color for the label text.
- 9. Click the **B Bold**, **I Italic**, and **U Underline** buttons to achieve the necessary formatting.
- 10. Click **Shape** flyout to choose the shape of the label enclosure.
- 11. Specify the leader line and jog as necessary.
- 12. Click $\stackrel{\text{\tiny REC}}{\longleftrightarrow}$ **Orientation** to specify horizontal or vertical text.

13. Select **As Drawn** if you want the label definition to reflect the label rule only and not have any formatting overrides.



- If you select **As Drawn**, the label definition is determined by the label rule. You cannot set the font, text size, text color, and so forth for the label.
- 14. Click the drawing to place the label.
- 15. Continue placing labels on the drawing if necessary.
- 16. Right-click to end the command.
- 17. Save your changes before exiting the 2D Drawing Editor.

Related Topics

 Manual Place Labels Command (Manually Place Labels Toolbar), page 144

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Imperial Drawing Types: An Overview

The imperial drawing types are drawing templates that include such things as drawing borders, documentation annotation, note areas, and selection and resymbolization criteria.

Several drawing types are delivered, fully designed to meet particular drawing requirements. There are two versions of each drawing type; one using imperial (or English) units, and the other using metric units. They are accessed when you right-click a folder and select **New**. On the **Add Component** dialog box, go to the **Imperial** tab. For more information, see *Add a Component*, page 36.

You can use the delivered types as the basis for creating new drawing types, modifying the view styles or border templates as needed. You can copy a template from an existing drawing or you can copy volumes only, allowing you to create multiple drawings with the same border graphics. To copy a drawing type component, select the item on the **Management Console** hierarchy, then select **Copy**. To past the item, right-click a location in the hierarchy or in the **Detail View**, then select **Paste**.

All of the delivered drawing types provide customizable templates and view styles. The delivered **Equipment Plan** drawing type is provided as an example below:

- The **Equipment Plan** is a single view drawing plan. It includes the location of equipment, structural columns, building walls, equipment steel, vessel and mechanical steel, roads, and railroads.
- The **Equipment Plan** includes general information for coordinate systems, sheet scales, and modifications. The drawing border provides the border graphics, title block graphics, and the title block labels. For more information on editing the template, see *Edit Border Template Command (Tools Menu)*, page 96.

The document annotation includes the following:

- **North Arrow** The large symbol is used, which is typical for single view drawings. As of version 6.0, the north arrow is placed per the drawing view. Click and drag the symbol to position it within the Note Area if required.
- **Key Plan** The key plan is used to show the geographic position of the single grid relative to the rest of the grids of the same type in the single block. For information on defining key plan styles, see *Define Key Plan Style Command (Tools Menu)*, page 97. To place a key plan, see *Place a Key Plan*, page 142.
- **Drawing Notes** This is a collection of notes consisting of general notes applicable to all drawings, notes applicable to a discipline, notes applicable to a category of drawings, and notes specific to a single drawing (such as a user-defined element or a border report).

Note Area

The Note Area is used to display drawing notes and key plans. The note area on the Equipment Plan is 5" wide on the right-hand side of the drawing border. This area extends from the top of the border down to the top of the title area border. The Note Area is optional. It is not a required element in the template.

View Regions

The View Region defines the drawing view arrangements. The **Equipment Plan** defaults to a single view with a 5" Note Area and 1" margins around the drawing view.

Drawing View

The **Equipment Plan** is a single view plan. The following specifications are set:

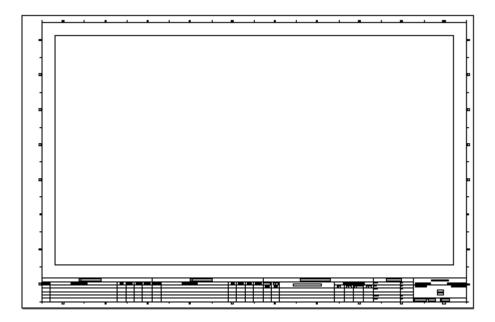
- **Direction** Set to Plan view (Looking Plan).
- **Rotation** Depends on the volume type and the Coordinate System defined, based on the grid section's X and Y size. For example, a volume placed by 4-Points in the Space Management task will rotate the view.
- Scale Set to 1/8'' = 1' or 1 = 100.
- **North Arrow** One symbol included per drawing view. Click and drag the symbol to the Note Area.
- Annotation The plan uses *matchline labels* to indicate the appropriate coordinate of that segment of the grid boundary along with continuations. The *matchlines* are lines that outline the boundaries of a grid by following the exterior boundaries of the collection of sectioning elements defining the grid represented in the view.
- **Drawing Volumes** The drawing volume is the queried 3D volume in the model.

View Styles

The view style specifies the object filters included in the drawing if present in the queried 3D volume. It specifies how objects are displayed, including graphical representation, labeling, and dimensioning. In the **Equipment Plan**, the volume and snapshot view style definitions are the same. For more information on view styles, see *Define View Style Command (Tools Menu)*, page 54.

Title Blocks

The title block generally displays at the bottom of a drawing template. It can include signatures, revision and issue information, and other properties associated with the drawing.



You can add drawing properties to the title block using the **Place Drawing Property Label** command when editing a template. For more information, see *Place Drawing Property Label Command (Drawing Labels Toolbar)*, page 123.

Imperial Drawing Types: An Overview					

Snapshot Drawings: An Overview

Snapshot Drawings are orthographic drawings used in cases where the drawing layout is different for each drawing. Each drawing is independent regarding view arrangement and view properties. Each view is saved and composed as a drawing.

Snapshot Drawings preserve workspace definition at the time the snapshot is created, considering both workspace and range. The view style acts as a style override. Everything in the workspace ends up on the drawing using the default symbology. Only objects specified within the view style are overridden. You can specify the view style using the **Tools > Define View Style** command.

Before you create these types of drawings, an administrator should create or edit drawing view styles and templates for use with the drawings. For example, you can place title block labels and reports on the templates to reflect your project or company standards.

The Snapshot component provides a quick and easy way to convert a three-dimensional graphic view into a drawing. You create views using commands in a three-dimensional task such as Piping, and then choose these views to place onto drawing sheets.

The overall workflow for snapshot drawings is:

- Place snapshot views in a 3D task, such as Electrical, using the Tools > Drawing View command
- Open a template in this task for a snapshot drawing component
- Place drawing views
- Generate the drawings
- Refresh to see if the drawings require an update after the model changes
- As necessary, update the drawings
- Save the drawings and print
- Publish the drawings as a viewable file; no physical data is published

Notes

• A snapshot view is based on the current filter in the model when you create it. It will always be tied to the same filter. The software makes a private copy of the filter and stores it so subsequent changes to the filter definition do not impact previously created snapshot views. You can create new objects or modify existing ones so that they meet the filter active at the time the previous snapshot was made, then you update the snapshot drawing using the Update or Update Now command. Now the snapshot drawing shows all changes affecting the snapshot view.

- If the drawing view you created on the drawing component template is larger than the snapshot view, the software centers the snapshot within the drawing view. If the drawing view is smaller than the snapshot view, the software clips the snapshot view to fit the size of the drawing view.
- If you are zoomed out on a clipped object showing a lot of blank space, the software automatically limits the amount of empty space shown in the snapshot drawing.

Related Topics

• Snapshot Drawings Common Tasks, page 155

Snapshot Drawings Common Tasks

The following tasks are used frequently when you create snapshot drawings. Some of the tasks are performed within the Drawings and Reports task, while others are performed in other 3D tasks.

Create a Snapshot Component

In the Drawings and Reports task, you create a Snapshot component. For more information, see *Add a Component*, page 36.

Create a Snapshot of the 3D View

In any 3D task (such as Piping or Electrical), you perform the following to define the workspace for your snapshots. Refer to task-specific documentation for each of these procedures.

- Define the workspace to include objects desired on the snapshot drawing.
- Hide objects you want to exclude from the snapshot drawing.
- Select objects to be included in the snapshot drawing. You want to do this if you are using select set when creating the drawing view.
- Specify the orientation for your 3D view.
- Create a snapshot drawing view, assigning the drawing type and view style.

Create the Snapshot Drawing Document

In the Drawings and Reports task, create the snapshot drawing document. Right-click a snapshot component and select **Create Drawing**. For more information, see *Create Drawing Command (Snapshot Drawings)*, page 157. The snapshot drawing documents are created based on a template you specify. You perform the following to complete the drawing:

- Specify a template on the Select Template Dialog Box. For more information, see Select Template Dialog Box.
- Place at least one drawing view on the drawing template. For more information, see *Place a Drawing View for Snapshot Drawings*, page 133.
- Update the snapshot drawing document. For more information, see *Updating Documents: An Overview*, page 337.

Viewing the Drawing Log

You can view the drawing log to see any messages associated with the drawing. For more information, see *View Log Command*, page 52.

Set Drawing Properties

You can specify the properties for the drawing component or drawing documents by right-clicking and selecting **Properties**. For more information, see *Edit Document Properties*, page 334.

Manually Add Labels

In many cases, you need to include additional labels or dimensions on the snapshot drawing after it is created. For more information, see *Place a Manual Label on a Drawing*, page 145. This is an optional task and is not required to create snapshot drawings.

Publish the Snapshot Drawing

Publish the Snapshot Drawing to The Engineering Framework (TEF). You can only publish if you are registered with TEF. For more information, see *Publishing Documents: An Overview*, page 373.

Note

• The viewable files created when you publish drawings and reports provide relationship links to the 3D Model Data. You must also publish the 3D Model Data to provide the navigation between the viewable files and the 3D Model Data. For more information, see 3D Model Data: An Overview, page 317.

Create Drawing Command (Snapshot Drawings)

Generates drawings for the snapshot component. Before you can use this command to generate snapshot drawings, you must complete the steps below.

Snapshot Drawings

- Create at least one Snapshot Drawing Type component in the Management Console hierarchy.
- Define a Snapshot view style using the **Tools** > **Define View Style** command in this task.
- Create at least one snapshot view using the Tools > Drawing View command in a 3D task, such as Piping.

Notes

- A snapshot view is based on the current filter in the model when you create it. It will always be tied to the same filter. The software makes a private copy of the filter and stores it so subsequent changes to the filter definition do not impact previously created snapshot views. You can create new objects or modify existing ones so that they meet the filter active at the time the previous snapshot was made, then you update the snapshot drawing using the Update or Update Now command. Now the snapshot drawing shows all changes affecting the snapshot view.
- If you are zoomed out on a clipped object showing a lot of blank space, the software automatically limits the amount of empty space shown in the snapshot drawing.

Related Topics

- Create a Snapshot Drawing, page 158
- Snapshot Drawings: An Overview, page 153

Select Template Dialog Box

Specifies a template. This dialog box appears when you click the **Edit Border Template** command. It also appears the first time you edit a template for a volume component or create a snapshot drawing. The templates listed in this dialog box are located on the **Symbols** share in the \Drawings\Catalog\Templates folder.

You can select a template on this dialog box and then click \mathbf{OK} , or you can just double-click a template.

The application delivers a set of Imperial and Metric border templates. The names of the templates indicate their size. All of the delivered Imperial and Metric border templates already contain border labels. Some of the border templates also contain a label to display notes. The naming convention indicates which templates contain this label.

Related Topics

- Create a Volume Drawing, page 167
- Edit Border Template Command (Tools Menu), page 96
- Edit Template Command (Volume Drawings Component), page 169
- Volume Drawings: An Overview, page 161

Create a Snapshot Drawing

1. In the **Management Console**, verify that at least one snapshot drawing component exists. If none exists, add a snapshot drawing component.

Add a Component, page 36

- 2. Switch to a 3D task, such as Piping or Equipment and Furnishings.
- 3. In the 3D task, create one or more drawing views using the **Tools > Drawing View** command. For more information, see the *Common User's Guide* available from the **Help > Printable Guides** command in the software.
- 4. Switch back to the Drawings and Reports task.
- 5. Right-click a snapshot drawing component in the **Management Console**, and click **Create Drawing**.



- Right-click the same component you chose when you created the drawing views in the 3D task.
- 6. On the **Select Template** dialog box, select a border template for the drawing, and click **OK**.
- 7. In the **2D Drawing Editor**, click **Place Drawing View** .
- 8. On the ribbon, select a view in the **View Name** box. This box contains all the views created for the component.
- 9. If necessary, change the view style. By default, the **View Style** box contains the style specified during the drawing view creation.
- 10. In the **View Scale** box, select or type a scale. If you select **No Scale**, you must drag the view by two points. If you select **Custom**, you can type numbers for the scale in the boxes to the right of the ribbon.
- 11. Click to place the view on the drawing sheet.

∀ Tip

- You can place as many views on the sheet as you defined for the drawing type in the 3D task. Right-click to end the Place Drawing View command.
- 12. If necessary, add symbols to the drawing view. You may need to click **Symbol Explorer** to display the symbol browser.

💡 Tip

- For example, you can place a north arrow symbol. When the drawing
 is generated, the north arrow will orient itself according to the view
 direction.
- 13. Close the drawing editor, and save the changes you made.
- 14. In the **Management Console**, right-click the snapshot drawing component, and select **Update Document(s)** to create the drawings.
- 15. Open the drawings by right-clicking them in the **Detail View** and selecting **Open** on the shortcut menu.

Notes

- After you generate snapshot drawings, you cannot change the view scale.
- You can export drawings to files by right-clicking a folder, component, or single drawing and then selecting **Save As** on the shortcut menu.
- A snapshot view is based on the current filter in the model when you create it. It will always be tied to the same filter. When the filter changes and you update the snapshot drawing using the Update or Update Now command, the snapshot drawing shows all filter changes affecting the snapshot view.
- If you are zoomed out on a clipped object showing a lot of blank space, the software automatically limits the amount of empty space shown in the snapshot drawing.

Related Topics

• Snapshot Drawings: An Overview, page 153

Snapshot Drawings: An Overview				

Volume Drawings: An Overview

You can generate Volume Drawings. Before you create these types of drawings, an administrator should create or edit drawing view styles and templates for use with the drawings. For example, you can place title block labels and reports on the templates to reflect your project or company standards.

After you have your view styles and templates in place, you are ready to start creating drawings. The volume drawings require that the changes occur within the drawing template.

The Volume Drawing component uses a template to create drawings. You can place a view on the template and associate the view with a view style to control the output. To define the contents of the view, you create a drawing volume in the model.

The software allows you to place several views on a template each with independent scale. In the Space Management task, you can place a volume, which is associated to a single or multiple views. The volume you create in Space Management is associated with all the views you create within a volume drawing template. Therefore, in a multiple view volume drawing, you can have multiple views of the same volume, and each view specifies the style, scale, and orientation for the volume associated with the view. If the volume you create is too big for a particular view at its scale, the volume is clipped.

For example, if you place two views in a drawing template with the following property settings:

- **DrawingView1** Size 6" x 6", Scale 1" = 1', and Orientation = Plan
- **DrawingView2** Size 6" x 6", Scale 1" = 1', and Orientation = North.

You use the **Place Volume By 2-Points** command in the Space Management task from coordinates (**0,0,0**) to (**6,6,6**). In this case, the volume is exactly the right size based on scale and orientation. The generated drawing contains the complete contents of the volume.

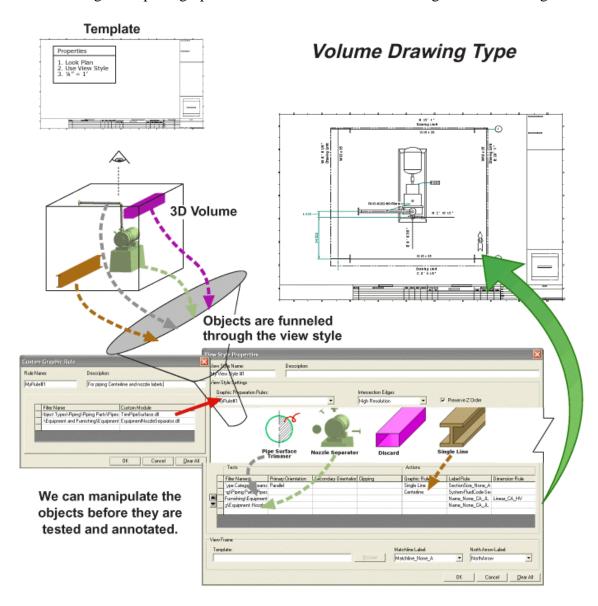
However, if your volume was placed from (0,0,0) to (8,6,6), the volume is *too big* for the drawing view on the template. The maximum volume size the view can accommodate is no more than 6 inches on each side. The software centers the volume in the drawing view, effectively giving you the information from (1,0,0) to <7,6,6 in the output drawing.

The overall processing for volume drawings is:

• Identify a drawing volume. This is a query of all the objects that have been created in the database which reside inside the volume borders. The objects do not have to be visible or even in the current workspace to be included.

- When the drawing is created, the software checks the objects from the volume query against the list of filters in the specified view style.
- If an object in the volume matches one of the specified view style filters, the software applies the rules set for the object filter. If the object does not match a filter, it is not included in the drawing view.

The following conceptual graphic shows the workflow for creating volume drawings:



The software applies the view styles in a top-down manner. If an object finds its match in several filters, the last one applied (the bottom-most style) overrides any preceding style rules.

When a drawing is updated, it can be published by right-clicking the document and selecting Publish on the right-click menu. The document is published as a viewable graphic file; no physical data is published.

Related Topics

- Create a Snapshot Drawing, page 158
- Create a Volume Drawing, page 167

Defining Drawing Volumes: An Overview

Drawing volumes are a critical piece of the volume drawing generation process. Drawing volumes are different from the other drawing components in that they exist in the three-dimensional model as objects.

After you place a drawing view on a volume template, you must define one or more volumes for the view. Volumes clip the results of a query for a drawing view, which displays a two-dimensional representation of the model.

You can define a drawing volume with the **Place Drawing Volume** commands in the Space Management task. It is possible to size the volume based on the grid coordinates in the model. Volumes can be placed by view (if single view), by 2-points, by 4-points, and by object selection (one or more views).

Note

• When you use drawing volumes with key plans, you can specify that the key plan is created with a coordinate system relative to the drawing volume view. For more information on placing key plans, see *Place Key Plan Command (Template Toolbar)*, page 140. For information on setting the properties for the key plan, including the coordinate system property, see *Key Plan Style Dialog Box*, page 99.

Volumes are stored in the Plant database. When a 3D task such as Piping is active, you can see the defined volumes on the **Space** tab of the **Workspace Explorer**. With this solution, you can locate and manipulate drawing volumes quickly and easily. When you create a drawing volume, the software automatically names the volume and places it in the space hierarchy.

! Important

• If you delete a drawing volume in the model, the software deletes any associated drawings and views.

Related Topics

- Create a Volume Drawing, page 167
- Volume Drawings: An Overview, page 161

Volume Drawings Common Tasks

The following tasks are used frequently when you create volume drawings.

Create a Volume Component

You create a Volume component using the **New** command. For more information, see *Add a Component*, page 36.

Edit the Volume Component Template

The template for the volume drawing defines the size and location of the frames that will contain the drawing volumes. For more information, see *Edit a Volume Drawing Template*, page 169. If your volume drawing requires a key plan, you can include it by placing one while in the 2D Drawing Editor. For more information, see *Place Key Plan Command (Template Toolbar)*, page 140.

Place Volumes in Space Management

In the Space Management task, create volumes using one of the **Place Drawing Volume** commands on the vertical toolbar. For more information, see *Defining Drawing Volumes: An Overview*, page 164. For detailed information on these commands, refer to the *Space Management User's Guide*.

Create the Volume Drawing Document

To create the volume drawing document, right-click the component and select **Create Drawing**. The drawing documents are created. For more information, see *Create Drawing Command (Volume Drawings)*, page 167.

Set Volume Drawing Properties

You can set drawing-level properties with the **Properties** command. For more information, see *Edit Document Properties*, page 334.

Updating Volume Documents

Update the volume drawing document. For more information, see *Updating Documents: An Overview*, page 337.

Viewing the Drawing Log

You can view the drawing log to see any messages associated with the drawing. For more information, see *View Log Command*, page 52.

Set Drawing Properties

You can specify the properties for the drawing component or drawing documents by right-clicking and selecting **Properties**. For more information, see *Edit Document Properties*, page 334.

Manually Add Labels

In many cases, you need to include additional labels or dimensions on the volume drawing after it is created. For more information, see *Place a Manual Label on a Drawing*, page 145. This is an optional task and is not required to create volume drawings.

Publish the Volume Drawing

Publish the Volume Drawing to The Engineering Framework (TEF). You can only publish if you are registered with TEF. For more information, see *Publishing Documents: An Overview*, page 373.

Note

• The viewable files created when you publish drawings and reports provide relationship links to the 3D Model Data. You must also publish the 3D Model Data to provide the navigation between the viewable files and the 3D Model Data. For more information, see 3D Model Data: An Overview, page 317.

Create Drawing Command (Volume Drawings)

Generates a drawing for a volume component. Before you can use this command to generate drawings, you must complete the steps below.

- Create at least one Volume Drawing Type component in the **Management** Console hierarchy.
- Define a Spatial view style using the **Tools > Define View Style** command in this task.
- Create at least one volume drawing view with the **Place Drawing Volume** commands in the Space Management task.

Note

If the drawing view you created on the drawing component template is larger than the volume you created using the **Place Drawing Volume** command, the software centers the volume on the drawing view. If the volume is smaller than the drawing view, the software clips the volume to fit the drawing view according to its scale.

Related Topics

- Create a Snapshot Drawing, page 158
- Volume Drawings: An Overview, page 161

Create a Volume Drawing

1. In the **Management Console**, verify that at least one volume drawings component exists. If none exists, add a component for volume drawings.

Add a Component, page 36

- 2. Right-click the volume drawings component, and click **Edit Template** on the shortcut menu.
- 3. On the **Select Template** dialog box, click a template file.



- You can modify the templates in this task by clicking **Tools > Edit Border Template**. For example, place drawing property labels or reports in the title block and save.
- 4. Click **Place Drawing View** to place one or more drawing views on the template.

Place a Drawing View for Volume Drawings, page 131

5. Select a view style for the drawing view.

Tip

- You can create and edit view styles by clicking Tools > Define View
 Style while in this task.
- 6. If necessary, add symbols to the drawing view. You may need to click **Symbol Explorer** to display the symbol browser.

- For example, you can place a north arrow symbol. When the drawing is generated, the north arrow will orient itself according to the view direction.
- 7. Exit and save the template.
- 8. Create one or more drawing volumes to associate with the drawing view by switching to the Space Management task and clicking one of the **Place Drawing Volume** commands.
- 9. In the **Management Console**, right-click the volume drawings component, and click **Update Document(s)** to create the drawings.
- 10. Open the drawings by right-clicking them in the **Detail View** and selecting **Open** on the shortcut menu.

Notes

- The software allows you to place several views on a template. In the Space Management task, you can place a volume, which is associated to all the views
- After you generate drawings, the software does not allow you to edit the view scale.
- You can export drawings to files by right-clicking a folder, component, or single drawing, and then selecting **Save As** on the shortcut menu.
- If volume drawings contain embedded reports and you have Office XP installed, you must modify the security settings in Excel for the embedded reports to run correctly. To change this setting, open Excel, and click Tools > Macros > Security. On the Trusted Sources tab, select Trust access to Visual Basic Project. This setting must be modified before you update the drawing and generate the report within it.
- If the drawing view you created on the drawing component template is larger than the volume you created using the **Place Drawing Volume** command, the software centers the volume on the drawing view. If the volume is smaller than the drawing view, the software clips the volume to fit the drawing view according to its scale.

Related Topics

• Volume Drawings: An Overview, page 161

Edit Template Command (Volume Drawings Component)

Modify a template for a volume drawing component in the 2D Drawing Editor. Use the **Template Toolbar** commands to add a drawing view, report, or key plan.



When you use the **Template Toolbar** commands, make sure you are on the **DwgTemplate** layer of the drawing so the software preserves the placed objects when you update the drawing.



If you use other native 2D Drawing Editor commands (such as Place Line or Place Dimension) to add manual markups to the template, put them on the **Default** or a layer with "User" as the prefix (for example, a layer named **UserAnnotationLayer** to preserve the changes when you update drawings.



Before you click the **Edit Template** command for a volume drawing, you can use the **Tools > Edit Border Template** command to place drawing property labels in the title block of a template. For more information, see *Place Drawing Property Label Command (Drawing Labels Toolbar)*, page 123.

Related Topics

- Create a Volume Drawing, page 167
- Edit Border Template Command (Tools Menu), page 96
- Volume Drawings: An Overview, page 161

Edit a Volume Drawing Template

- 1. Right-click a volume drawing component in the **Management Console**, and select **Edit Template**.
- 2. On the **Select Template** dialog box, select a border template for the drawing, and click **OK**. The template opens in the 2D Drawing Editor.
- 3. In the **Drawing Editor** window, click **Place Drawing View**. Click and drag to place a view. You can place multiple volume drawing views.
- 4. On the **Properties** dialog box for each drawing view, provide a name and description for the drawing view.

- 5. Specify a style and scale on the drawing sheet, and set the view orientation for each drawing view.
- 6. If your volume drawing requires a key plan, you can place one with the **Place Key Plan** command.

Place a Key Plan, page 142

7. Save the changes you made and close the 2D Drawing Editor.

Related Topics

• Volume Drawings: An Overview, page 161

Exporting 3D Drawings to MicroStation: An Overview

You can export 3D drawings directly to MicroStation. The actual export is not a drawing file. It is an actual 3D file translated into MicroStation graphic types. The 3D drawings can be either volume or snapshot drawings.

While the workflow is similar to that of regular volume and snapshot drawings, there are some differences.

- You create a MicroStation component with the same **Add Component** command. Once the component is created, right-click and select **Setup** from the popup menu.
- The **Setup** command allows you to define the MicroStation seed file and style to use in generating the component drawings.
- The view style you select determines which objects are included in the view as well as what layers are placed in MicroStation. The seed file selected determines the MicroStation symbology used within the drawing.

Related Topics

• Create MicroStation DGN Files, page 173

Setup Command (MicroStation DGN Files)

Sets options for creating a MicroStation DGN files. This command is available on the right-click popup menu on the MicroStation 3D DGN component.

Related Topics

• Create MicroStation DGN Files, page 173

Setup Dialog Box (MicroStation DGN Files)

Sets options for creating a MicroStation DGN files.

Seed File - Specifies the seed file to use to create the drawing.

View Style - Selects the defined view style to use in determining which objects appear in the drawing. Click **Properties** to display the **Select View Style** dialog box. For more information, see *Select View Style Dialog Box*, page 56.

Related Topics

• Create MicroStation DGN Files, page 173

Create Drawing Command (MicroStation DGN Files)

Generates a drawing for the MicroStation 3D DGN components. Before you can use this command to generate drawings, you must complete the steps below.

- Create at least one MicroStation 3D DGN component in the Management Console hierarchy.
- Right-click on the component and select **Setup** to define the seed file and view style to use in generating the 3D drawings for export into MicroStation.
- Create at least one volume with the **Place Drawing Volume** commands in the Space Management task.

Notes

- If the drawing view you created on the drawing component template is larger than the volume you created using the **Place Drawing Volume** command, the software centers the volume on the drawing view. If the volume is smaller than the drawing view, the software clips the volume to fit the drawing view according to its scale.
- All MicroStation DGN files are currently created in the location where the symbol share is configured. You can navigate to the symbol share from the **MicroStation Manager** directly.
- The MicroStation DGN data reflects the clipping applied in SP3D and the style information defined in the graphic rule. No data is persisted on any of the objects, but all objects are mapped to levels.

Related Topics

• Create MicroStation DGN Files, page 173

Create MicroStation DGN Files

- In the Management Console, create a MicroStation 3D DGN component.
 Add a Component, page 36
- 2. Right-click the MicroStation 3D DGN component, and select **Setup** from the shortcut menu.
- 3. On the Setup dialog box, select the appropriate seed file to use from the options available.
- 4. Select the view style you want to use to determine what objects are displayed in the drawing.

- 5. Create one or more drawing volumes to associate with the drawing view by switching to the Space Management task and clicking one of the **Place Drawing Volume** commands.
- 6. In the **Management Console**, right-click the MicroStation 3D DGN drawings component, and select **Create Drawing(s)** to create the blank drawing.
- 7. Right-click on the drawing, and click **Update Now** to update the information displayed in the drawing.

♀ Tip

- Click **View in 3D** to open a graphic view of the drawing. This view will show the clipped contents of the volume.
- 8. You can export drawings to DGN files by right-clicking a folder, component, or single drawing, and then selecting **Save As** on the shortcut menu.

Notes

- All MicroStation DGN files are currently created in the location where the symbol share is configured. You can navigate to the symbol share from the **MicroStation Manager** directly.
- The MicroStation DGN data reflects the clipping applied in SP3D and the style information defined in the graphic rule. No data is persisted on any of the objects, but all objects are mapped to levels.
- Once a view style is selected on the Setup dialog box, you can use the view style properties to add, edit, or remove filters that determine what objects are included in the drawing.
- A new Layer property has been added to the graphic rule properties. Its value field can hold a 32-character text string. The translation software will either treat it as a named layer or interpret numbers from 1 to 64.
- After you generate drawings, the software does not allow you to edit the view scale.
- If volume drawings contain embedded reports and you have Office XP installed, you must modify the security settings in Excel for the embedded reports to run correctly. To change this setting, open Excel, and click Tools > Macros > Security. On the Trusted Sources tab, select Trust access to Visual Basic Project. This setting must be modified before you update the drawing and generate the report within it.

- Create Drawing Command (MicroStation DGN Files), page 173
- Exporting 3D Drawings to MicroStation: An Overview, page 171

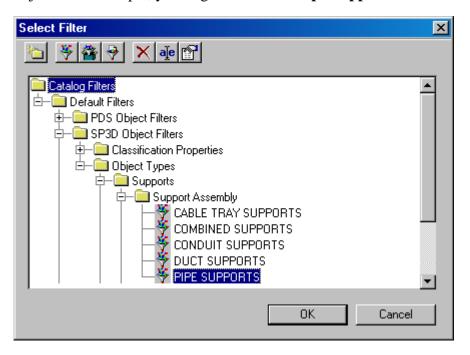
Exporting 3D Drawings to MicroStati	ion: An Overview	

Drawings by Query Filters: An Overview

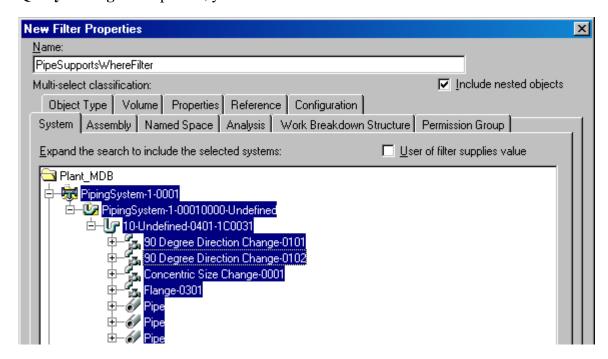
When you create the Drawings by Query components (Orthographic, Piping Isometric, and Drawings by Query Manager), you use filters to determine the content of the documents produced. You define Orthographic and Piping Isometric Drawings by Query components using a **Setup** command to specify a *what* filter that accesses the model database and determines which objects are included in the drawings, as well as which hierarchy is traversed to create a tree in the **Management Console**. The exact position in the hierarchy is determined by the *where* filter. You define a Drawings by Query Manager component using a **Setup** command to specify a *where* filter that identifies the location of the objects included in the drawing.

For more information on the tasks involved in creating Drawings by Query, see *Orthographic Drawings by Query Common Tasks*, page 183 and *Piping Isometric Drawings by Query Common Tasks*, page 195.

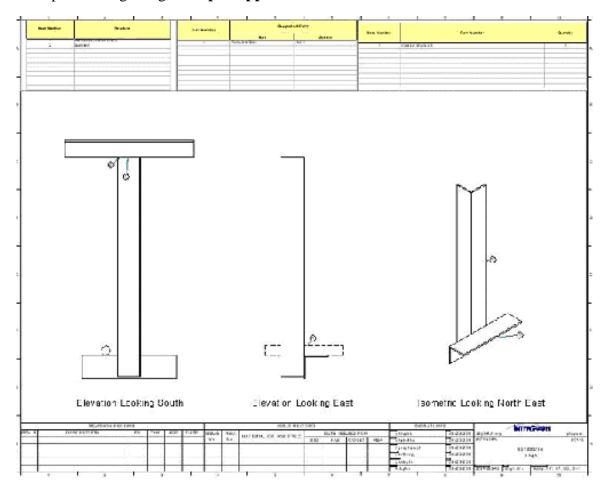
For example, if you are creating Orthographic drawings that include pipe supports, you create a new **Orthographic Drawings by Query** component. When you run **Setup** on the component, you select a normal filter that returns only pipe support objects. For example, you might choose the **Pipe Supports** filter as shown below:



After saving your **Orthographic Drawings by Query** component setup as a package, you define a Drawings by Query Manager to determine *where* in the model to collect the objects you specified in the *what* filter. When you run **Setup** on the **Drawings by Query Manager** component, you select a normal filter.



When you run the query, the software compounds the *where* filter with the *what* filter to return the desired objects. Each object is documented in a drawing using the template and rules you setup for the package. The following graphic shows an example drawing using the **Pipe Supports** filter:



For more information on defining filters, see the *Common User's Guide* available from **Help > Printable Guides**.

- Orthographic Drawings by Query: An Overview, page 181
- Piping Isometric Drawings by Query: An Overview, page 193

Drawings by Query Filters: An Overview						

Orthographic Drawings by Query: An Overview

You can generate orthographic (volume) drawings using a filter-based query. The Orthographic Drawing by Query component allows you to create drawings for many objects in the model all in the same manner. This component does not require physical volumes in the model. The software uses a filter-based query to collect the objects and document them automatically.

For example, Hanger drawings use parts that can be queried from the database. There are many of them, and in most cases, they are documented the same way. Using the Orthographic Drawing by Query component means you do not have to place hundreds of drawing volumes to document each type of part or assembly.

The basic workflow for creating Orthographic Drawings by Query is as follows:

- Create an Orthographic Drawing by Query component that specifies what you want to document
- Create a Query Manager that says where to look for the data
- Run the filter-based query
- Create the drawings
- Update the drawings, if necessary
- Publish the drawings to a viewable graphic file; no physical data is published

Administrator Setup

Your administrator sets up appropriate templates to use with the Orthographic Drawings by Query component. The administrator is also responsible for creating filters that define "what" and filters that specify "where" in the model to look for the objects. The template and the "what" filter information within the component are saved as a package.

Drawings by Query Manager

The Drawings by Query Manager uses the normal filter to specify "where" to look for the objects included in the drawing. You do not use an asking filter with the Drawings by Query Manager. For more information on the filters necessary for setting up a Drawings by Query component, see *Drawings by Query Filters: An Overview*, page 177.

- Orthographic Drawings by Query Common Tasks, page 183
- Setup an Orthographic Drawing by Query Component, page 185
- Setup Command (Drawings by Query Manager Component), page 189
- Setup Command (Orthographic Drawing by Query Component), page 185

Orthographic Drawings by Query Common Tasks

The following tasks are used when you create orthographic drawings by query.

For information on filters required for creating Orthographic Drawings by Query, see *Drawings by Query Filters: An Overview*, page 177.

Setup an Orthographic Drawing by Query Component

Create an Orthographic Drawing by Query component and perform **Setup** on it to specify the appropriate filter. For more information, see *Setup an Orthographic Drawing by Query Component*, page 185. When you select a filter in **Setup**, you are specifying the "what" portion of the query. In other words, you are specifying the objects to be included in the drawing. You can also specify how much extra space you want to include around the objects.

Edit the Template

You edit the drawing border template to meet your output requirements. For more information, see *Edit Border Template Command (Tools Menu)*, page 96.

Save the Orthographic Drawing Component as a Package

Save the Orthographic Drawing by Query component as a package so you can join it with a **Drawings by Query Manager** to generate the query drawings. For more information, see *Save a Package*, page 39.

Add a Drawings by Query Manager

You create the **Drawings by Query Manager**, then set it up to specify the "where" portion of the query for the Orthographic and Piping Isometric Drawing components. For more information, see *Create a Drawings by Query Manager*, page 190.

Run the Query

You use the **Run Query** command to execute the query specified by the Orthographic Drawing by Query component and the Query Manager. For more information, see *Run Query Command*, page 192.

Create or Update the Drawings

To create or update the drawings, right-click the component and select the appropriate command. For more information, see *Updating Documents: An Overview*, page 337.

Viewing the Drawing Log

You can view the drawing log to see any messages associated with the drawing. For more information, see *View Log Command*, page 52.

Set Drawing Properties

You can specify the properties for the drawing component or drawing documents by right-clicking and selecting **Properties**. For more information, see *Edit Document Properties*, page 334.

Viewing in 3D

You can view the drawing in 3D by right-clicking an Orthographic Drawing by Query component and selecting **View in 3D** on the shortcut menu. For more information, see *View Items in 3D*, page 104.

Publish the Orthographic Drawing

Publish the Orthographic Drawing by Query documents to The Engineering Framework (TEF). You can only publish if you are registered with TEF. For more information, see *Publishing Documents: An Overview*, page 373.

Note

• The viewable files created when you publish drawings and reports provide relationship links to the 3D Model Data. You must also publish the 3D Model Data to provide the navigation between the viewable files and the 3D Model Data. For more information, see 3D Model Data: An Overview, page 317.

Setup Command (Orthographic Drawing by Query Component)

Sets component options for creating Orthographic Drawings by Query. This command is available on the right-click popup menu for the Orthographic Drawing by Query component.

Related Topics

- Orthographic Drawings by Query: An Overview, page 181
- Setup an Orthographic Drawing by Query Component, page 185

Setup Dialog Box (Orthographic Drawing by Query Component)

Sets options on Orthographic Drawing by Query components.

Filter - Identifies the filter that defines the "what" portion of the query. The software uses the filter to determine the objects included in the drawings when they are generated. Select **More** in the **Filter** dropdown list to display the **Select Filter** dialog box. Click **Properties** to display the **Filter Properties** dialog box. For more information on filters for Orthographic Drawings by Query, see *Drawings by Query Filters: An Overview*, page 177.

Volume Growth - Specifies the *growth range*, or extra space, you want to include around the objects identified by the query. The larger the setting, the more space is included around the object.

Related Topics

• Setup Command (Orthographic Drawing by Query Component), page 185

Setup an Orthographic Drawing by Query Component

- 1. Right-click the folder where you want to store the new Orthographic Drawing by Query component, then select **New**. The **Add Component** dialog box appears.
- 2. Select the **Orthographic Drawing by Query** component, and then click **OK**. The Orthographic Drawing by Query component is created in the folder.
- 3. Right-click the **Orthographic Drawing by Query** component and select **Setup** to specify the properties for the component.

- 4. On the **Setup** dialog box, specify a filter. Click **More** to display the **Select Filter** dialog box. This is the filter that specifies "what" objects to include in the drawing. Click **Properties** to display the current filter properties. For more information on filters for Drawings by Query, see *Drawings by Query Filters: An Overview*, page 177.
- 5. If you want extra space included around the objects, specify a **Volume Growth** value. The larger the value, the more space is included around each object.
- 6. Click OK to save the changes to the Orthographic Drawings by Query component.

To use the component to generate orthographic drawings you need to save it as a package and associate it to a **Drawings by Query Manager**. For more information, see *Create a Drawings by Query Manager*, page 190.

Notes

- To delete a component, right-click it and select **Delete**.
- To rename a component, right-click it and select **Rename**.

- Orthographic Drawings by Query Common Tasks, page 183
- Orthographic Drawings by Query: An Overview, page 181

Select Filter Command

Specifies a filter for the orthographic or piping isometric drawings. The filter narrows the objects returned for the drawings.

For more information on setting up filters for Drawings by Query components, see *Drawings by Query Filters: An Overview*, page 177.

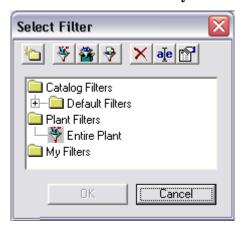
Related Topics

- Create an Isometric Drawing, page 205
- Piping Isometric Drawings by Query: An Overview, page 193
- Setup an Orthographic Drawing by Query Component, page 185
- Volume Drawings: An Overview, page 161

Select Filter Dialog Box

Creates, edits, deletes, and selects filters for use with the **Define Workspace**, **Surface Style Rules**, and **Select by Filter** commands. You can access this dialog box in several ways:

- Click File > Define Workspace and select the More option in the Filter box
- Click **Format** > **Surface Style Rules**, click **New** or **Modify**, and then select the **More** option in the **Filter** box.
- Click Tools > Select by Filter.



The tree view displays the following types of filters:

• Catalog Filters - These filters are used like reference data in the Catalog. For example, a catalog filter could apply to company-wide operations. Your administrator could define Company_Filter_1, Company_Filter_2, and so forth.

- **Plant Filters** These filters are available to all users assigned to a specific database model. You must have the appropriate privileges to create, edit, or delete these filters.
- My Filters These are personal filters that you create and place in the My Filters folder. They are visible only to you, the owner. You cannot see the personal filters of other users, and they cannot see your personal filters. Select a filter from one of those listed, or create a new filter to meet your specific requirements.
- New Folder Creates a new folder.
- New Filter (Simple or Asking) Displays the New Filter Properties dialog box where you can create a new filter. Asking filters allows you to specify the parameters of the search. An asking filter has built-in functionality to ask for values (with boxes that you are required to record). The values apply to properties you have already designated you will supply when the filter is run. Asking filters are portable between models.
- New Compound Filter Displays the New Compound Filter Properties dialog box where you can create a new compound filter containing the Or, And, or Not operators.
- **→ New SQL Filter** Displays the **New SQL Filter Properties** dialog box where you can type the text of an SQL query.
- ➤ Delete Removes a filter or folder from the Select Filter list and delete it. If you delete a folder, the software also deletes its contents.
- **Rename** Changes the name of an existing filter or folder from the **Select Filter** list.
- **Properties** Displays the **Filter Properties** dialog box on which you can select the properties that determine your filter search criteria.

Notes

- If this dialog box is activated from the **Select by Filter** command, you can select multiple filters on this dialog box. Hold CTRL or SHIFT and click each filter. On **OK**, all objects that fit the selected filters are selected.
- If this dialog box is activated from the **Select by Filter** command, it clears the select set before adding objects to the select set.

Setup Command (Drawings by Query Manager Component)

Sets options for creating a Drawings by Query Manager. This command is available on the right-click popup menu for Drawings by Query Manager components.

The Drawings by Query Manager is used in conjunction with other components, such as the Orthographic Drawing by Query and Piping Isometric Drawing by Query components, to complete the query for objects in the model. The Drawings by Query Manager provides the filter that specifies the "where" side of the query. It tells the query "where" to look for the objects specified by the component "what" filter.

Related Topics

- Create a Drawings by Query Manager, page 190
- Orthographic Drawings by Query: An Overview, page 181
- Piping Isometric Drawings by Query: An Overview, page 193

Setup Dialog Box (Drawings by Query Manager Component)

Sets options for creating Drawings by Query Manager components.

Filter - Identifies the filter to use to define the "where" portion of the query. The software uses the filter to determine where to look for the objects requested in Orthographic Drawing by Query and Piping Isometric Drawing by Query components when they are generated. Select More in the dropdown list to display the Select Filter dialog box. Click Properties to display the current filter properties. For more information on filters for the Drawings by Query Manager, see *Drawings by Query Filters: An Overview*, page 177.

Package - Specifies the package to use in completing the query.

Note

• You must create an Orthographic Drawing by Query or Piping Isometric Drawing by Query package before setting up the Drawings by Query Manager component. For more information, see *Save Package Command*, page 38.

- Create a Drawings by Query Manager, page 190
- Setup Command (Drawings by Query Manager Component), page 189

Create a Drawings by Query Manager

The Drawings by Query Manager component works in conjunction with the Orthographic Drawing by Query and Piping Isometric Drawing by Query packages. Before using this command, you must create packages for your Orthographic Drawing by Query and Piping Isometric Drawing by Query components. For more information, see *Save Package Command*, page 38.

1. Right-click the folder where you want to create your **Drawings by Query Manager**. The **Add Component** dialog box appears.



- You can store the **Drawings by Query Manager** anywhere in the **Management Console**, but it is best to store it in the same location as the components with which it works.
- 2. Select the **Drawings by Query Manager** component, then click **OK**. The Drawings by Query Manager component is created in the folder.
- 3. Right-click the **Drawings by Query Manager** component and select **Setup** to specify the properties for the component. The **Setup** dialog box appears.
- 4. Specify a filter in the **Filter** field. The dropdown shows the most recently selected filters. Select **More** in the dropdown list to display the **Select Filter** dialog box and specify a filter. Click **Properties** to display the current filter properties.
 - The filter you select is the "where" portion of the query, as opposed to the "what" portion specified when you setup the Orthographic Drawing by Query or Piping Isometric Drawing by Query component. The filter you specify here tells where in the model you want to look for the objects.
 - For more information on filters for Drawings by Query, see *Drawings by Query Filters: An Overview*, page 177.
- 5. In the **Package** field, specify the Orthographic Drawing by Query or Piping Isometric Drawing by Query package you created. The dropdown contains the most recently selected packages. Select **More** to display the **Select Package** dialog box. For example, if you are defining a Drawings by Query Manager for an Orthographic Drawing by Query, select an Orthographic Drawing by Query package.
- 6. Click **OK** to save the settings.

To create the drawings, you need to run the query. For more information, see *Run Query Command*, page 192.

Notes

- To delete a component, right-click it and select **Delete**.
- To rename a component, right-click it and select **Rename**.

- Orthographic Drawings by Query: An Overview, page 181
- Piping Isometric Drawings by Query: An Overview, page 193
- Setup Command (Drawings by Query Manager Component), page 189

Run Query Command

Runs the query associated with the selected Drawings by Query Manager component. For example, if you have a Piping Isometric Drawing by Query package associated with a Drawings by Query Manager and you execute **Run Query**, the software looks for piping in the model. The query results display beneath the style in the **Management Console** and also in the **Detail View**. Then you can create isometric drawings from the query results by right-clicking on the component and selecting **Create Drawings**.

If you have an Orthographic Drawing by Query package associated to a Drawings by Query Manager, the **Run Query** command runs the query associate with the components, collects the objects from the database, and builds the information that will be included in the drawing. Then you can create orthographic drawings from the query results by running the **Create Drawings** command on the Orthographic Drawing component.

The **Run Query** command uses the filters specified when you performed **Setup** components.

- Orthographic Drawings by Query: An Overview, page 181
- Piping Isometric Drawings by Query: An Overview, page 193

Piping Isometric Drawings by Query: An Overview

Isometric drawings communicate several important types of information to a pipe fabrication workshop. This information includes pipe cut lengths, bend angles, and welds. You create isometric drawing by associating a Piping Isometric Drawing by Query component to a Query Manager. The Piping Isometric Drawing by Query component specifies the "what" portion of the query, while the Query Manager specifies the "where".

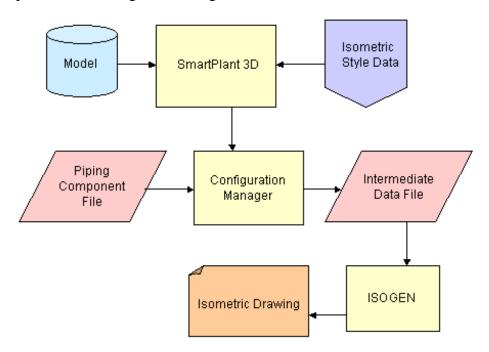
The basic workflow for creating Piping Isometric Drawings by Query is as follows:

- Create a Piping Isometric Drawing by Query component that specifies what you want to document
- Create a Drawings by Query Manager that specifies where to look for the data
- Run the filter-based query
- Create the drawings
- Update the drawings, if necessary
- Publish the drawings to a viewable graphic file; no physical data is published

Each Piping Isometric Drawing by Query component has an associated isometric style. The delivered styles are Iso_Pipeline, Iso_Piperun, Iso_PenSpool, Iso_Spool, Iso_WBS, and Iso_Stress. You can open the Isometric Style Options Browser from any of the isometric drawing styles by right-clicking the drawing style in the Management Console and selecting Edit Options on the shortcut menu. The browser allows you to set options for isometric drawing format and content.

After creating the Piping Isometric Drawings, you can open them for viewing, editing, printing, and publishing (if you are registered with The Engineering Framework). To troubleshoot the drawings, you can use the **View Data** command to access part and reports information and log files.

The software uses a third-party engine made by Alias, ISOGEN, to generate isometric drawings from the three-dimensional model. The following illustration shows the process of creating the drawings.



The SmartPlant 3D software creates a piping component file (PCF), which is used along with the isometric style information to create an intermediate data file (IDF). The ISOGEN engine then reads the IDF and creates the isometric drawings.

Administrator Setup

Your administrator should setup appropriate isometric styles to use with the Piping Isometric Drawings by Query component. The administrator is also responsible for creating asking filters that define "what" and filters that specify "where" in the model to look for the objects. The template and the "what" filter information within the component are saved as a package.

Drawings by Query Manager

The Drawings by Query Manager uses the filter to specify "where" to look for the objects included in the drawing.

For more information on the filters necessary for setting up a Drawings by Query component, see *Drawings by Query Filters: An Overview*, page 177.

- Customizing Isometric Drawing Styles: An Overview, page 207
- Piping Isometric Drawings by Query Common Tasks, page 195

Piping Isometric Drawings by Query Common Tasks

The following tasks are used when you create Piping Isometric Drawings by Query.

For information on filters required for creating Piping Isometric Drawings by Query, see *Drawings by Query Filters: An Overview*, page 177.

Setup a Piping Isometric Drawing by Query Component

Create and perform set up for a Piping Isometric Drawing by Query component. For more information, see *Setup a Piping Isometric Drawing by Query Component*, page 198. When you select a filter in **Setup**, you are specifying the "what" portion of the query. In other words, you are specifying the objects to be included in the drawing.

Save the Piping Isometric Drawings by Query Component as a Package

Save the Piping Isometric Drawing by Query component as a package so you can join it with a **Query Manager** to generate the query drawings. For more information, see *Save a Package*, page 39.

Add a Drawings by Query Manager

You create a **Drawings by Query Manager**, then set it up to specify the "where" portion of the query for the Piping Isometric Drawing by Query component. For more information, see *Create a Drawings by Query Manager*, page 190.

Run the Query

You use the **Run Query** command to execute the query specified by the Piping Isometric Drawing by Query component and the Drawings by Query Manager. For more information, see *Run Query Command*, page 192.

Create or Update the Drawings

To create or update the drawings, right-click the component and select the appropriate command. For more information, see *Updating Documents: An Overview*, page 337.

Viewing the Drawing Log

You can view the drawing log to see any messages associated with the drawing. For more information, see *View Log Command*, page 52.

Set Drawing Properties

You can specify the properties for the drawing component or drawing documents by right-clicking and selecting **Properties**. For more information, see *Edit Document Properties*, page 334.

Viewing the Pipeline in 3D

You can view the pipeline in 3D by opening a piping isometric drawing and selecting **View in 3D** on the toolbar. For more information, see *View Items in 3D*, page 104.

Viewing Piping Data

You can view the piping extraction data by right-clicking a piping isometric drawing and selecting **View Extraction Data** on the shortcut menu. For more information, see *View Piping Isometric Extraction Data*, page 203.

Publish the Piping Isometric Documents

Publish the Piping Isometric Drawing by Query documents to The Engineering Framework (TEF). You can only publish if you are registered with TEF. For more information, see *Publishing Documents: An Overview*, page 373.

Note

• The viewable files created when you publish drawings and reports provide relationship links to the 3D Model Data. You must also publish the 3D Model Data to provide the navigation between the viewable files and the 3D Model Data. For more information, see 3D Model Data: An Overview, page 317.

Setup Command (Piping Isometric Drawing by Query Component)

Sets component options for creating Piping Isometric Drawings by Query. This command is available on the right-click popup menu for the Piping Isometric Drawing by Query component.

Related Topics

- Piping Isometric Drawings by Query: An Overview, page 193
- Setup a Piping Isometric Drawing by Query Component, page 198

Setup Dialog Box (Piping Isometric Drawing by Query Component)

Sets options on Piping Isometric Drawing components.

Filter - Identifies the asking filter to use to define the "what" portion of the query. The software uses the filter to determine the objects included in the drawings when they are generated. Select **More** in the **Filter** dropdown list to display the **Select Filter** dialog box. Click to display the **Filter Properties** dialog box. For more information on filters for Piping Isometric Drawings by Query, see *Drawings by Query Filters: An Overview*, page 177.

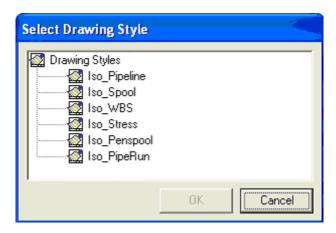
Style - Specifies the isometric style to use when generating the output for the piping isometric drawing. The delivered styles include **Iso_Pipeline**, **Iso_Piperun**, **Iso_Spool**, **Iso_PenSpool**, **Iso_WBS**, and **Iso_Stress**. Your administrator may provide more isometric styles. Click **More** in the dropdown to display the **Select Drawing Style** dialog. For more information, see *Select Drawing Style Dialog Box*, page 198.

You can click the **Properties** to access the **Isometric Style Options Browser** and edit the isometric style as needed. For more information, see *Use the Isometric Style Options Browser*, page 214.

- Setup a Piping Isometric Drawing by Query Component, page 198
- Setup Command (Piping Isometric Drawing by Query Component), page 197

Select Drawing Style Dialog Box

Displays a list of all available Isometric Styles. This dialog displays when you select **More** in the **Style** dropdown on the **Setup** dialog box for a Piping Isometric Drawing component or package.



Related Topics

- Setup a Piping Isometric Drawing by Query Component, page 198
- Setup Command (Piping Isometric Drawing by Query Component), page 197
- Setup Dialog Box (Piping Isometric Drawing by Query Component), page 197

Setup a Piping Isometric Drawing by Query Component

- 1. Right-click the folder where you want to store the new Piping Isometric Drawing by Query component, then select **New**. The **Add Component** dialog box appears.
- 2. Select the **Piping Isometric Drawing by Query** component, and then click **OK**. The Piping Isometric Drawing by Query component is created in the folder.
- 3. Right-click the Piping Isometric Drawing by Query component, and select **Setup** on the shortcut menu.
- 4. On the **Setup** dialog box, specify a filter in the **Filter** field. The dropdown shows the most recently selected filters. Click **More** to display the **Select Filter** dialog box and specify a filter. The filter you select is the "what" portion of the query; for example, Pipe Runs. Click **Properties** to display the current filter properties. For more information on filters for Drawings by Query, see *Drawings* by Query Filters: An Overview, page 177.

Note

• Select a filter that is appropriate for the isometric styles you want to use for this Piping Isometric Drawing by Query component. For example, for the **Iso_Pipeline** style, the filter must contain piping and

- pipeline systems. For the **Iso_Spool** and **Iso_PenSpool** styles, the filter must contain spool assemblies.
- 5. Specify a **Style** to use for the piping isometric drawings. The delivered styles include **Iso_Pipeline**, **Iso_Piperun**, **Iso_Spool**, **Iso_PenSpool**, **Iso_WBS**, and **Iso_Stress**. Your administrator may provide more isometric styles. Click **More** in the dropdown to display a list of all available styles. For more information, see *Select Drawing Style Dialog Box*, page 198.
- 6. Click **Properties** to display the **Isometric Style Options Browser**. Modify options for the isometric styles you added as needed. For more information, see *Use the Isometric Style Options Browser*, page 214.
- 7. Click **OK** to create the Piping Isometric Drawing component as specified.

To use the component to generate piping isometric drawings you need to save it as a package and associate it to a **Drawings by Query Manager**. For more information, see *Create a Drawings by Query Manager*, page 190.

Notes

- To delete a component, right-click it and select **Delete**.
- To rename a component, right-click it and select **Rename**.

- Piping Isometric Drawings by Query: An Overview, page 193
- Setup Command (Piping Isometric Drawing by Query Component), page 197

View Pipeline in 3D Command

Displays the pipeline system in a 3D graphical view. This command is similar to the **View in 3D** command available for orthographic drawings.

You can access this command by right-clicking a pipeline system in the **Management Console**.

The **View in 3D** command enables the selection capability in the **Workspace Explorer**. For more information, see *Workspace Explorer Command (View Menu)*, page 26.

- Create an Isometric Drawing, page 205
- Piping Isometric Drawings by Query: An Overview, page 193

View Extraction Data Command

Displays part and report information for a line that has been processed. You can view the log file and Piping Component File (PCF) data for the extraction. If you have customized the style to produce report files such as the neutral file and cut pipe list, you can also view those files directly in the **Extraction Data** dialog box.

An important advantage of this command is that it allows you to view the files for an extraction without navigating to the output folder in Windows Explorer.

To access this command, right-click a line or isometric drawing in the tree view or list view. This command helps you to troubleshoot extraction errors.

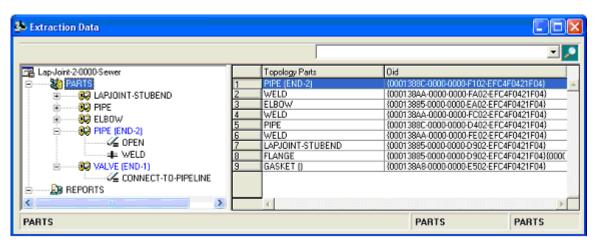
Related Topics

• View Piping Isometric Extraction Data, page 203

Extraction Data Dialog Box

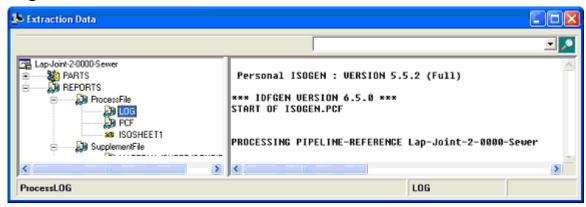
Provides troubleshooting information for the isometric drawing extraction process by displaying parts and reports. The search box at the top of this dialog box allows you to find keywords in the data. You can access this dialog box by right-clicking a piping isometric drawing and selecting **View Extraction Data** on the shortcut menu.

You can view the extracted parts by expanding the **Parts** node on the left side of the dialog box.

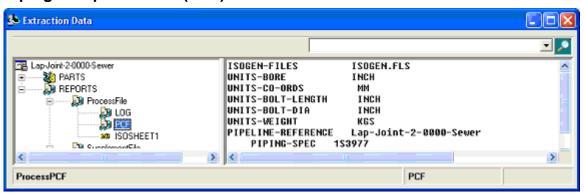


The **Reports** node includes:

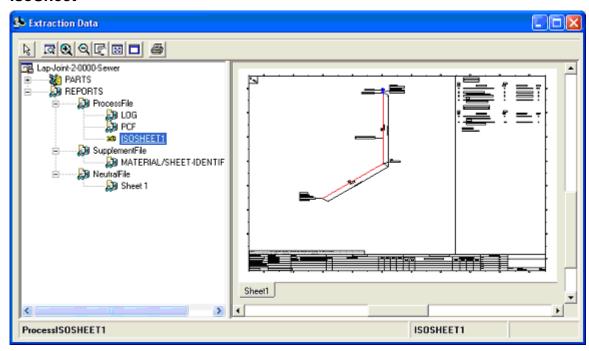
Log File



Piping Component File (PCF)

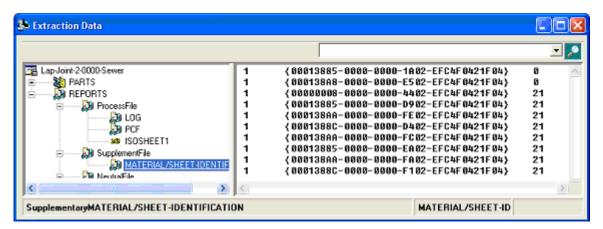


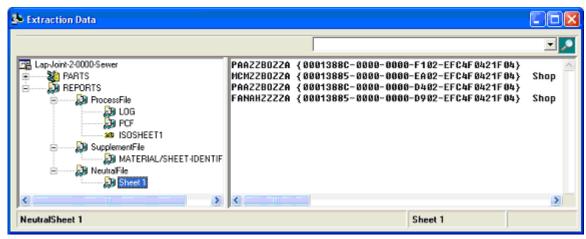
IsoSheet



Other Report Files

Examples include Supplement file and Neutral file.





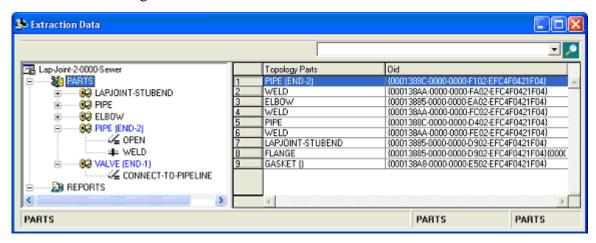
Related Topics

- View Extraction Data Command, page 201
- View Piping Isometric Extraction Data, page 203

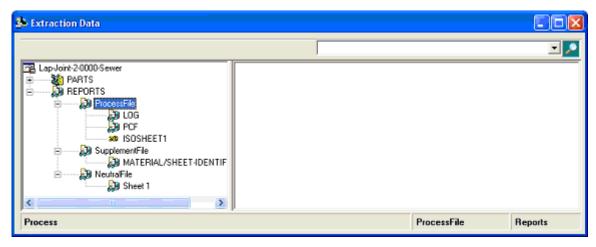
View Piping Isometric Extraction Data

- 1. In the **Management Console**, right-click an isometric drawing.
- 2. Select **View Extraction Data** on the shortcut menu. The dialog box displays the part and report information for the line, including the log file and Piping Component File (PCF) data.

3. To view the extracted parts information, select and expand the **Parts** node on the left side of the dialog box.



4. To view the report information, select and expand the **Reports** node on the left side of the dialog box.



5. Type a word into the **Search** field at the top of the dialog box and press **Enter** or click to locate the keyword in the extraction data.

Notes

- If you have customized the style to produce report files, such as the neutral file and cut pipe list, you can also view those files directly in the dialog box.
- An important advantage of the View Extraction Data command is that it
 allows you to view the files for an extraction without navigating to the
 output folder in Windows Explorer.

- Create a Piping Component File (PCF), page 221
- View Extraction Data Command, page 201

Create an Isometric Drawing

1. In the **Management Console**, verify that at least one isometric piping drawing exists. If none exists, add a component for isometric drawings.

Setup a Piping Isometric Drawing by Query Component, page 198

2. Modify options for the isometric styles you added as needed.

Use the Isometric Style Options Browser, page 214

3. Right-click the isometric drawing style, and click **Run Query**. The software populates the **Detail View** with the available items for that style.



- To display available items for all isometric styles in a component, right-click the component in the Management Console, then click Run Query.
- 4. Right-click a folder, component, pipeline, or spool and click **Update Document(s)** to create isometric drawings.



- You can extract multiple objects (for example, pipelines) if you hold
 Ctrl or Shift while you select the objects in the Management Console or Detail View.
- 5. To view a drawing, double-click the drawing name in the **Detail View**.

Note

• You can remove items from an isometric piping drawing component by modifying the filter and running the query again.

Related Topics

• Piping Isometric Drawings by Query: An Overview, page 193

Piping Isometric Drawings by Query: An Overview						

Customizing Isometric Drawing Styles: An Overview

Isometric drawing styles control several aspects of the isometric drawing output, including the output location and the type of object used for drawing creation. Each drawing style is also associated with a set of options and a backing sheet.

To customize the isometric drawing styles for your company, you can copy and then modify the delivered isometric drawing styles.

Isometric Drawing Styles

The delivered isometric drawing styles are **Iso_Pipeline**, **Iso_Piperun**, **Iso_Spool**, **Iso_PenSpool**, **Iso_WBS**, and **Iso_Stress**. Each delivered style has an associated XML file and IGR file. The XML file contains the isometric options, and the IGR file is the backing sheet for the isometric drawing.

- **Iso_Pipeline** Creates a final isometric drawing used to construct the plant. This style creates one drawing per pipeline system. This drawing style is an example of a potential configuration for a fabrication isometric. It includes a material list.
- **Iso_Piperun** Creates a draft of an isometric drawing for checking against project guidelines. This style creates one drawing per pipe run. This drawing style is an example of a drawing configuration used for checking pipeline designs prior to extracting the fabrication isometric.
- **Iso_Spool** Creates an isometric drawing used in the fabrication shop to manufacture the pipe. This style uses piping spools. You can create spools using the **Generate Spools** command in the Piping task. Like the final isometric style, it includes a material list.
- **Iso_PenSpool** Creates an isometric drawing that documents penetration spools that consist of a penetration plate and several piping spools. This style creates one drawing per penetration spools. You can create penetration spools using the **Create Penetration Spools** command in the Piping task.
- Iso_WBS Creates an isometric drawing that documents a collection of
 parts that are assigned to one Work Breakdown Structure (WBS) item of
 the type Group Iso Drawing. This style creates one drawing per WBS
 item.
- **Iso_Stress** Creates a Piping Component File (PCF) that can be output to the CAESAR II pipe stress analysis software. No drawing is created. To save the PCF file, use the **Save As** command. For more information, see *Create a Piping Component File* (*PCF*), page 221.

You can create other isometric drawing styles, such as a Bid style for construction contractors to bid on a project.

Isometric Options

The isometric options define the isometric drawing output, which includes symbols, dimensions, layers, drawing frame attributes, material lists, weld lists, detail sketches, and many other settings. The options are stored in XML file in the [Product Directory]\CatalogData\Symbols\PmfgIsoStyleData folder on the server.

You should modify the option control data using the **Edit Options** command. The command displays the **Isometric Style Options Browser**, which is the tool used to control centrally all the options related to the appearance and information content of the various styles of isometric drawings. If you directly manipulate the XML file, your changes may not take effect in the software.

You can also use the **Edit Options** command to import, export, and save symbol maps.

The isometric option control data is integrated within the catalog reference data. You must have access rights to the catalog reference data to edit and save the option control data.

Backing Sheets

The backing sheet allows you to customize the drawing style with your company's logo, watermark, and drawing borders. The delivered backing sheets are stored as IGR files in the [Product Directory]\CatalogData\Symbols\PmfgIsoStyleData folder on the server.

A document can include two different types of sheets: working sheets and background sheets. Working sheets contain design data, and background sheets contain title block graphics, borders, company logo, and watermarks. Each working sheet can contain a reference to a background sheet. If this reference is set, the size of the background sheet determines the size of the working sheet. Also, graphics on the background sheet become visible in the working sheet.

When you create a backing sheet for isometric drawings, the backing sheet must have the following characteristics:

- It must contain a single, empty working sheet.
- It must contain a single background sheet containing the required graphics. The background sheet should be set to the appropriate size and scale.
- The working sheet must contain a reference to the background sheet.
- The working sheet must be active when you save the template.

• The working sheet should have layers set up with the required colors, unless the color is specified explicitly in the isometric options. If the working sheet does not have the required layers, the ISOGEN software creates the layers automatically and assigns them to the colors specified in the options.

Related Topics

• Use the Isometric Style Options Browser, page 214

Isometric Style Common Tasks

The following tasks are used when you create new piping isometric styles.

Modify Drawing Border Files

You can create or modify border files two ways. You can modify existing delivered border files. For more information, see *Modify an Existing Border File*, page 218.

If you have existing MicroStation DGN files you want to use as a drawing border, you can import them. For more information, see *Import an Existing MicroStation DGN Border*, page 219.

Create New Isometric Styles

You can add new isometric styles to the delivered *BulkLoadIsoKeys.xls* file and bulkload the changes into the model data. For more information, see *Create a New Isometric Style*, page 220.

Use the Isometric Style Options Browser

You can modify, save, and import style options through the **Isometric Style Options Browser**. For more information, see *Use the Isometric Style Options Browser*, page 214.

Importing Data from the Style XML File

You can import style data from the style XML file directly through the **Isometric Style Options Browser**. When you import styles, they are shown immediately within the Browser. For more information, see *Import Data from the Style XML File*, page 216.

Develop the Look and Feel of Drawings

You can change the look and feel of an isometric drawing by changing options within the **Isometric Style Options Browser**. Options can specify everything from drawing content to the system controls for output definition. For more information, see *Developing the Look and Feel of the Drawing: An Overview*, page 225.

Use Alternative Text

AText is an abbreviation for alternative text, an ISOGEN feature that allows you to change or remove any text on the isometric drawing. You can substitute your own text in the place of standard ISOGEN words. For more information, see *Alternative Text Options: An Overview*, page 548.

Map Isometric Data to Drawing Layers

You can map layers within a previously created drawing border file to isometric data. The options used to define the mapping are found in the **Isometric Style Options Browser** in the **Drawing.Layers** and **Drawing.Definitions** options. For more information, see *Mapping Isometric Data to Drawing Layers*, page 223.

Configure the Material List

You can specify three different styles for material lists on isometric drawings. You use the **Isometric Style Options Browser** to set the options for the MTO Neutral File. For more information, see *Configuring the Material List: An Overview*, page 234.

Assign Labels

You assign labels to attributes within a drawing through the Isometric Style Options Browser. For more information, see *Assign Labels*, page 215.

Populate the Title Block

You can use labels to customize the title block of a drawing. Labels are often used for single pieces of data, such as the approval date or your company name. You can use the **Isometric Style Options Browser** to specify options for the appearance and content of the title block. For more information, see *Populate the Title Block*, page 216.

Setting the Symbol Mapping

You can set symbol mapping in the **Isometric Style Options Browser**. For more information, see *Symbol Mapping (SymbolMAP): An Overview*, page 543.

Edit Options Command

Sets various options that affect isometric drawing output, such as dimension styles, layers, drawing frame text, material lists, and weld lists.

You can also use the **Edit Options** command to import, export, and save symbol maps. You can import isometric keys from either a bulkload spreadsheet or from a previously created and saved XML file containing isometric keys.

You can access this command by right-clicking a Piping Isometric Drawings component or package in the **Management Console**. You must have access rights to the catalog reference data to save isometric options to the Catalog database.

Related Topics

• Customizing Isometric Drawing Styles: An Overview, page 207

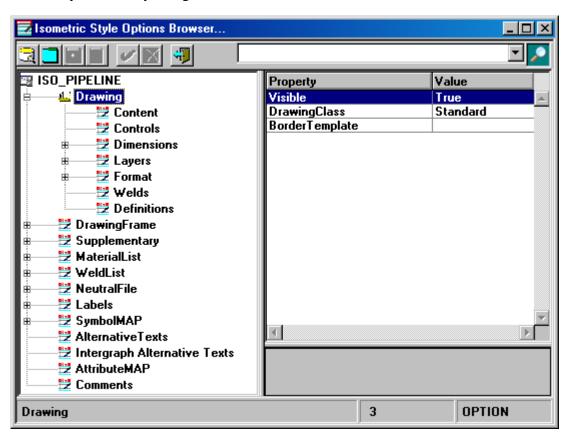
Isometric Style Options Browser Dialog Box

Sets options for an isometric drawing style. You can use this browser to control all the options related to the appearance and information content of the various styles of isometric drawings. You can save changes permanently to the Catalog database.

The left pane of the browser window groups the options for a style. Click a node to view its corresponding table. For descriptions of the options, see *Appendix B: Isometric Drawing Options*, page 393. The **SymbolMAP** node shows the current symbol mapping options available. For more information on symbols and symbol keys, see *Appendix D: Symbols and Symbol Keys*, page 555.

The right pane of the browser window shows the values set for the options. The grid presents two overall types of options. One type lists the options in two columns: **Property** and **Value**. An example is the **DrawingFrame** options. The other type lists the options in a multi-column format, where each row is part of an associated collection of options. An example is **Material List.Cut List.Summary File.Column**.

You can sort the columns by clicking the column headings. For the collection-based option categories, you can insert or delete rows by pressing the **Insert** or **Delete** keys on the keyboard or by using the **Insert Row** and **Delete Row** buttons on the toolbar.



- Save to Catalog Stores the isometric style options in the reference data. You must have write permission to the reference data for the software to store your changes in the Catalog database.
- Import Data From File Imports either a bulkload spreadsheet containing the isometric keys or a previously saved XML file containing the isometric keys. This button is enabled when you select the root, **SymbolMAP**, **AlternativeTexts**, or **AttributeMAP** node in the **Options Browser**. For more information, see *Import Data from the Style XML File*, page 216.
- Save Style XML File Saves the current style option data to an XML file. You can later import it. This button is enabled when you select the root node in the **Options Browser**.
- **Catalog Style Rule Properties** Displays information about the current style, such as name, type, description, and path to the isometric drawings. This information is stored in the Catalog database. This button is enabled when you select the root node in the **Options Browser**.

- ✓ Insert Row Adds a new row for data to the table currently displayed in the Table section of the dialog box. The row is inserted above the selected row.
- ➤ Delete Row Deletes the selected row from the table displayed in the Table section of the dialog box.
- **Exit** Closes the dialog box and prompts you to save changes if necessary.

Search - Finds an option based on text that you type. This command searches in the **Property** column. The software returns the results in the table portion of the browser. To find an option, type text in the **Search** box in the upper right of the **Isometric Style Options Browser** and either press **Enter** or click **Search** . You can also type specific option numbers in the **Search** box. For example, you can type **?OPT:79** to find option 79.

Related Topics

- Customizing Isometric Drawing Styles: An Overview, page 207
- Edit Options Command, page 212

Use the Isometric Style Options Browser

Isometric drawing styles control several aspects of the isometric drawing output, including the output location and the type of object used for drawing creation. Each drawing style is also associated with a set of options. You modify the isometric style options using the **Edit Options** command on the right-click menu for an Isometric Piping Drawing. This command displays the **Isometric Style Options Browser**.

- Right-click a Piping Isometric component or package in the Management Console, then click Edit Options to display the Isometric Style Options Browser.
- 2. Select a style option category in the hierarchy. Expand a branch on the tree to see the options.
- 3. Edit the options as necessary in the data grid on the right-hand side of the dialog box.

💡 Tips

- To find an option, type text in the Search box and either press Enter or click Find to locate the option in the hierarchy.
- You can find the option and switch number for the currently selected item displayed at the lower right of the dialog box.
- For the collection-based option categories (such as **Drawing.Layers.Column**), you can insert or delete rows in the grid by press the **Insert** or **Delete** keys or by clicking **Insert Row** ✓ or **Delete Row** X.

- 4. Click **Save To Catalog** to save the changes to the Catalog database. You must have write permission to reference data to save changes to isometric styles in the catalog.
- 5. Save the option style data by selecting the root node then click Save Style XML File. To import style data, click Import Data From File.
- 6. You can view the catalog style rule properties when you click **Latalog Style** Rule Properties.

Related Topics

Customizing Isometric Drawing Styles: An Overview, page 207

Assign Labels

You assign labels to attributes within a drawing through the **Isometric Style Options Browser**. There are different types of labels that can be applied. The Component Note, Continuation Note, and the Nozzle Note are used as examples below.

Isometric Component Note

- Right-click a Piping Isometric Drawing component or package in the Management Console, then click Edit Options to display the Isometric Style Options Browser.
- 2. Expand **Labels.ComponentNote**. The right side of the dialog box shows the options specified as Component Notes within the current style.
- 3. To create a new Component Note, click **Insert Row** . Set the attributes for the new Component Note as needed. For example: Select **PipeSupport** from the dropdown list in the **LabelAttribute2** field.
- 4. In the **LabelName** field, click the ellipsis button to open the **Catalog Labels** dialog box. Select a label to use for the Component Note.
- 5. If you want the Component Note to be enclosed by a border or bubble, specify it in the **MessageEnclosure** field.
- 6. Click **Save to Catalog** to save the changes to the Catalog database. You must have write permission to reference data to save changes to isometric styles.
- 7. Update your document to see the new applied style.

Continuation and Nozzle Note

- 1. Right-click an isometric style in the **Management Console**, then click **Edit Options** to display the **Isometric Style Options Browser**.
- 2. Expand **Labels.END-CONNECTION-EQUIPMENT** or **END-CONNECTION-PIPELINE**. These are the labels that give you the nozzle notes and continuation notes.
- 3. Make changes as necessary.
- 4. Save to the catalog as shown in the previous example.

5. Update the document to see the new applied style.

Related Topics

• Labels Options: An Overview, page 535

Import Data from the Style XML File

You can import style data from the style XML file directly through the **Isometric Style Options Browser**. When you import styles, they are shown immediately within the Browser. The level of the hierarchy selected when you invoke **Import** and all levels below it are replaced with the new values from the imported file. The following procedure uses the **Labels.DrawingFrame** level as an example.

- Right-click a Piping Isometric Drawing component or package in the Management Console, then click Edit Options to display the Isometric Style Options Browser.
- 2. Expand Labels.DrawingFrame and select DrawingFrame.
- 3. Click **Import Data From File** . The **Select File To Import** dialog appears.
- 4. Browse to the directory where the style XML file is located. For example, you may have edited the style content of the *Iso_Pipeline.xml* file located in *Program Files\SmartPlant\3D\Drawings\Templates\Styles*.
- 5. Select the file and click **OK** to retrieve the style information. A dialog box displays to confirm the file you selected. Click **Yes** to continue. The style option information is updated in the browser. Bulkloading is not necessary.
- 6. Click **Save to catalog** to save the changes to the Catalog database. You must have write permission to reference data to save changes to isometric styles.
- 7. Update your document to see the new applied style.

Related Topics

- Customizing Isometric Drawing Styles: An Overview, page 207
- Isometric Style Common Tasks, page 210

Populate the Title Block

You can use labels to customize the title block of a drawing. Labels are often used for single pieces of data, such as the approval date or your company name. You can use the **Isometric Style Options Browser** to specify options for the appearance and content of the title block.

The workflow for setting up attributes to populate the title block involves several steps. They are broken down as follows:

• Mapping HostAttributes to IsogenAttributes using the AttributeMAP options within the Isometric Style Options Browser.

- Assigning labels as needed using **Labels.DrawingFrame** options.
- Positioning text on the border with the **DrawingFrame.Attributes** options.

Mapping Attributes

- Right-click a Piping Isometric Drawing component or package in the Management Console, then click Edit Options to display the Isometric Style Options Browser.
- 2. Select **AttributeMAP**. Notice the mapping of **HostAttributes** to **IsogenAttributes**. For example, the following graphic shows HostAttribute **Piping Specification** is mapped to IsogenAttribute **PIPING-SPEC**.

	HostAttribute	IsogenAttribute
23	Piping Specification	PIPING-SPEC



- You can customize the mapping of ISOGEN Attributes
 ATTRIBUTE11 through ATTRIBUTE99. ATTRIBUTE1 through
 ATTRIBUTE10 are reserved by Intergraph. We recommend you start
 adding your own attributes from ATTRIBUTE21.
- 3. Expand **Labels.DrawingFrame**.
- 4. Notice the mapping of **IsogenAttributes** to labels. For example, IsogenAttribute **PIPELINE-REFERENCE** is mapped to the delivered catalog label **IsoPipelineReference**.

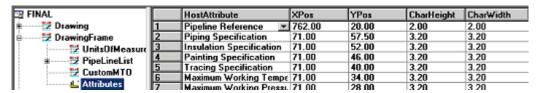
Some of the IsogenAttributes are *hardcoded*. For example, **PIPING-SPEC** always returns to the specification of the pipeline. It does not need or accept a label.

Assigning Labels within the Title Block

- 1. Expand Labels.DrawingFrame.
- 2. Specify the mapping of **LabelAttributes** to **LabelNames**. In the **LabelName** column, click the ellipsis button to display the **Catalog Labels** dialog box.
- 3. Expand branches of the catalog label hierarchy and select the label you want to assign to the **Label Attribute**.

Positioning Labels on the Drawing Border

- 1. Expand **DrawingFrame.Attributes**.
- 2. Notice the mapping of **HostAttributes** to location on the drawing sheet. The following graphic shows a sample mapping.



- 3. All values are listed in **mm**, measured from the origin of the sheet. For example, **Pipeline Reference** is shown in **2 mm font size at X=762mm**.
- 4. Change values as needed to position labels on the drawing border.

Related Topics

- Customizing Isometric Drawing Styles: An Overview, page 207
- Edit Options Command, page 212
- Isometric Style Options Browser Dialog Box, page 212

Modify an Existing Border File

You can create a drawing border from scratch using the **2D Drawing Editor**. You use the commands available within the **2D Drawing Editor** to place graphics and create appropriate layers.

When using layers in a template, keep in mind that the software preserves the **Default** layer and any layer that begins with *User* (for example, a layer named **UserAnnotationLayer**) when you update drawings. Manual markups on other layers are not preserved.

You should name new border files with the name of the desired isometric style, such as Iso_Pipeline or Iso_Piperun.

- 1. Navigate to [Program Files]\SmartPlant\3D\CatalogData\Symbols\PmfgIsoStyleData and copy the appropriate existing border igr file to the Drawings\Catalog\Templates directory. You may want to create a subdirectory for the new file you are creating.
- 2. Rename the copied file with the .sha extension.
- 3. Open the copied .sha files with the **2D Drawing Editor**.
- 4. Fit the view.
- 5. Select **Tools > Layers**. The **Layer** ribbon appears.



- 6. Select **Tools > Display Manager**. On the **Layers** tab, scroll down to see the values currently set in the *.sha* file. These are the color, line type, and width values for the named layers.
- 7. Make changes as needed, save the file, and exit the **2D Drawing Editor**. The next time you use this .*sha* file as your drawing border template, the graphics will show the changes you made.

Related Topics

- Customizing Isometric Drawing Styles: An Overview, page 207
- Isometric Style Common Tasks, page 210
- Working with Layers: An Overview, page 223

Import an Existing MicroStation DGN Border

If you have an existing MicroStation DGN file which is being used as a border (seed file), you can re-use it for your isometric drawings.

You should name new border files with the name of the desired isometric style, such as Iso_Piperun or Iso_Pipeline.

- 1. Using MicroStation tools, merge the seed and border file into a single DGN file.
- 2. Find the size for the required .*sha* file. For example, you may require a C-size to match your DGN file.
- 3. Navigate to the [Workstation Install Directory]\Common2D\Template directory and locate the Normal.sha file.
- 4. Right-click the file and select **Properties**. Remove the read-only flag from the file.
- 5. Open the *Normal.sha* file in the **2D Drawing Editor**.
- 6. Go to **File > Sheet Setup** and change the size of the file to match the size of the DGN file.
- 7. Save and close the file.
- 8. Open the **2D Drawing Editor** by double-clicking the *shape2dserver.exe* file in the *Common2D\Shape2D\Bin* directory.
- 9. Select **File > Open** and open the DGN file.
- 10. Select **Edit > Select All** to select all the DGN graphics.
- 11. Select **Edit** > **Cut** to remove the DGN graphics from the working sheet.
- 12. Select **View > Background Sheets**.
- 13. Select **Edit** > **Paste** to paste the DGN graphics on the background sheet.
- 14. Remove all extra layers using *cmddeletelayer.dll*, which is located in the \Symbol2D\Bin and can be added as a custom command in the **2D Drawing Editor**.
- 15. Change the background color if necessary using **Tools > Options**. Make other changes as needed.
- 16. Select **View > Working Sheets**.
- 17. Create a layer in the working sheet with "User" as the prefix of the name (for example, a layer named **UserAnnotationLayer**).

Note

• When using layers in a template, keep in mind that the software preserves the **Default** layer and any layer that begins with *User* when you update drawings. Manual markups on other layers are not preserved.

18. Save the file as a border template .*sha* file. The MicroStation graphics are saved as part of the new border template file to be used in the Drawings and Reports task.

Related Topics

- Customizing Isometric Drawing Styles: An Overview, page 207
- Isometric Style Common Tasks, page 210
- Working with Layers: An Overview, page 223

Create a New Isometric Style

You may require different isometric styles for different types of isometric drawings. You can add new isometric styles to the delivered *BulkLoadIsoKeys.xls* and bulkload the changes into the model data for use in the Drawings and Reports task. For more information on the *BulkLoadIsoKeys.xls* file, see the *Drawings and Reports Reference Data Guide*.

- 1. Open the *BulkLoadIsoKeys.xls* file located in *CatalogData\Bulkload\DataFiles*.
- 2. On the **PipeMfgRules** sheet, select a style row to use as a starting point for your new style. Right-click the row and select **Copy**. For example, you may want to create a new Iso_Pipeline style called **Iso_PipelineID**. You would select the row containing the delivered Iso_Pipeline style as the one to copy.
- 3. Select the End row. Right-click the row and select **Insert Copied Cells**.
- 4. Add the letter **A** to column A for this row. This tells the software to add the row when the file is bulkloaded.
- 5. Change the **Name** to your new style name. For example, you might name it **Iso_PipelineID**.
- 6. Change the value in the **OutPutIsoDrawingsLocation** column to the output location for your new style. For example, your location might be **IsoDrawings\Iso_PipelineID**.
- 7. Change the value in the **IsoBackingSheet** column to the location of the backing sheet. For example, your backing sheet may be located in **PmfgIsoStyleData\Iso_PipelineID.sha**.
- 8. Change the **IngrOption** value to the location of the style XML file. For example, your new style XML file may be located in **pmfgIsoStyleData\Iso_PipelineID.xml**.
- 9. Save the *BulkLoadIsoKeys.xls* file and close Excel.
- 10. Browse to the folder containing your new style files in the *Symbols* share on the server and copy the XML file. Rename it to match the information you provided in the *BulkLoadIsoKeys.xls* file.
- 11. Right-click the XML file and select **Properties**. Remove the read-only flag from the file.

- 12. Open the XML file with a text editor and edit it as needed to match the new naming convention for the file. In the example used above, you would edit to use Iso_PipelineID.
- 13. Save and close the file.
- 14. Bulkload the *BulkLoadIsoKeys.xls* file to update the model data. Make sure you bulkload using **Add/Modify/Delete** mode.
- 15. When in SP3D, switch to the Drawings and Reports task. The new style should be available when you add a style to an Isometric Piping drawing type.

Notes

• For more information on bulkloading files, see the *SmartPlant 3D Reference Data Guide*.

Related Topics

- Customizing Isometric Drawing Styles: An Overview, page 207
- Isometric Style Common Tasks, page 210

Create a Piping Component File (PCF)

When you create a Piping Isometric document that uses the Iso_Stress style, the software does not create drawings. Instead it creates the data necessary for a Piping Component File (PCF). You can then output the Piping Component File (PCF) to the CAESAR II pipe stress analysis software.

Note

• You can also view the PCF data with the **View Extraction Data** command. For more information, see *View Piping Isometric Extraction Data*, page 203.

To retrieve the PCF data, you actually use the **Save As** command.

Save As × Output Folder Rules As displayed C Append Parent Folder C Append Model Hierarchy Output Folder: Browse... File Types: Component Type Target File Type MicroStation 3D DGN MicroStation (*.dgn) ✓ Isometric Drawings Shape2dServer (*.sha) Spreadsheet Reports Worksheets (*.xls) Snapshot Drawings Shape2dServer (*.sha) ✓ Volume Drawings: Shape2dServer (*.sha)

1. Right-click the Piping Isometric document that uses the Iso_Stress style and select **Save As**. The **Save As** dialog box displays.

2. Specify the **Output Folder Rule** to be used. You can save the item as it appears in the **Management Console**, with its parent folder appended or with the entire model hierarchy appended.

Cancel

- 3. Specify the **Output Folder** location. Click **Browse** to display a dialog box to locate the appropriate folder location.
- 4. Check the **Isometric Drawings** component type. You can select multiple component types. For more information, see *Save As Dialog Box*, page 105.
- 5. In the **Target File Type** dropdown for the Isometric Drawing component type, select **PCR File (.pcf)**.
- 6. Click **OK** to save the files as specified.

The PCF file is saved to the location you specified, ready for use in stress analysis.

Note

• The saved drawings retain the same names they had in this task.

Related Topics

- Customizing Isometric Drawing Styles: An Overview, page 207
- Save As Command, page 105

Working with Layers: An Overview

Layers are used to distinguish between graphics within a template or drawing. You can create layers in the **2D Drawing Editor** with the **Tools > Layer** command.

If you intend to create manual markups within an Isometric Piping drawing, you should have a layer in the drawing template with a name prefix of *User* (for example, a layer named **UserAnnotationLayer**). You could alternatively place your markups on the **Default** layer. The software preserves these layers when you update drawings. Other layers are not preserved.

If named layers **do not exist** in the template, the software creates them using the symbology specified in the style XML file. In the **Isometric Style Options Browser**, expand **Drawing.Layers.Columns** to create new layers within the style XML file. Map definitions to the layers under **Drawing.Definitions**. To map isometric data to drawing layers using isometric options, see *Mapping Isometric Data to Drawing Layers*, page 223.

If the named layers **do exist** in the template, use **Tools > Display Manager** in the **2D Drawing Editor** to change the symbology used within the template.

Related Topics

• Isometric Style Common Tasks, page 210

Mapping Isometric Data to Drawing Layers

The **Isometric Style Options Browser** maps layers within a previously created drawing border file to isometric data. The options used to define the mapping are the **Drawing.Layers** and **Drawing.Definitions** options.

When using layers in a template, keep in mind that the software preserves the **Default** layer and any layer that begins with "User" (for example, a layer named **UserAnnotationLayer**) when you update drawings. Manual markups on other layers are not preserved.

A Warning

- Layer definitions created with **Tools > Display Manager** in the 2D Drawing Editor override these settings.
- 1. Right-click a Piping Isometric Drawing component or package, and click **Edit Options**. The **Isometric Style Options Browser** appears.
- 2. Expand **Drawing.Layers**. You can specify a default color to use for all layers if a color is not explicitly set.
- 3. Expand **Drawing.Layers.Columns**. The columns represent the mapping of the layers of the isometric drawing to the content for each layer.

Tip

- With the isometric drawing open in the 2D Drawing Editor, you can click **Tools > Display Manager** to see a list of the layers for the isometric drawing.
- 4. Expand **Drawing.Definitions**. The definitions specify the layers on the isometric drawing. You can specify scale and line thickness.
- 5. Click **Save To Catalog** to save the changes to the Catalog database. You must have write permission to reference data to save changes to the isometric style.

When you update your drawing document, the new styles are applied.

Note

• For more information on the drawing layer and drawing definition options, see *Layers (Drawing)*, page 421 and *Definitions (Drawing)*, page 443.

Related Topics

- Isometric Style Common Tasks, page 210
- Working with Layers: An Overview, page 223

Developing the Look and Feel of the Drawing: An Overview

You can change the look and feel of an isometric drawing by changing options within the **Isometric Style Options Browser**. Options can specify everything from drawing content to the system controls for output definition.

To set options for the drawing frame, see Set Drawing Frame Options, page 225.

To set options for drawing dimensions, see *Set Drawing Dimension Options*, page 227.

To control drawing content, see *Control Drawing Content*, page 228.

To set options for drawing controls, see Set Drawing Control Options, page 228.

To specify the drawing format, see *Specify Drawing Format*, page 229.

Notes

• For more information on isometric style options, see *Appendix B: Isometric Drawing Options*, page 393.

Related Topics

• Customizing Isometric Drawing Styles: An Overview, page 207

Set Drawing Frame Options

Using the **Drawing Frame** and **Attribute Map** options, you can specify the content and placement of drawing frame attributes on isometric drawings. Drawing frame text can include revision control information, process conditions, and miscellaneous design or specification notes, placed in the isometric drawing border or title block area. For more information on drawing frame options, see *Drawing Frame Options: An Overview*, page 446.

To specify this text, you first map ISOGEN attributes with user-defined text strings. You will use the text strings during the remainder of the attribute definition process. Then, you specify the size and position of the attribute text in the **Drawing Frame.Attributes** category.

For example, you can map the ISOGEN attributes DRAWING-NUMBER and DESCRIPTION with the strings Drawing Number and Description, respectively. When ISOGEN processes the Piping Component File (PCF) content given below, the values "CW-PipeRun1" and "Chillwater Drawing" are inserted into the title block accordingly.

DRAWING-NUMBER CW-PipeRun1

DESCRIPTION Chillwater Drawing

You also can use labels to specify drawing frame text. In the **Labels.Drawing Frame** category, you map ISOGEN attributes to label definitions in the catalog.

- 1. Right-click a Piping Isometric Drawing component or package, and click **Edit Options**. The **Isometric Style Options Browser** appears.
- 2. In the left frame of the dialog box, open the **Attribute Map** category.
- 3. Add rows or modify the rows in the grid.

→ Tips

- Type a meaningful text string in the **Host Attribute** column. You will use this text string in the **Drawing Frame** category.
- Select corresponding strings in the **ISOGEN** Attribute column.
- 4. Under **Drawing Frame**, click the **Attributes** category.
- 5. List all the attributes along with their corresponding locations and text sizes. Use the **Host Attribute** strings you typed in the **Attribute Map** category.
- 6. To use a label for drawing frame text, add an ISOGEN attribute to the **Labels.Drawing Frame** category, and map it to a label name and message enclosure.

Notes

- Sample labels for drawing frame text on isometric drawings include a pipeline reference label and current date label. The templates for these labels are located in the [Product Directory]CatalogData\Symbols\Labels\Base Templates folder.
- Another example of drawing frame customization is removing the north arrow. For more information, see *Edit the North Arrow on Isometric Drawings*, page 227.
- You can customize the mapping of ISOGEN Attributes ATTRIBUTE11
 through ATTRIBUTE99. ATTRIBUTE1 through ATTRIBUTE10 are
 reserved by Intergraph. We recommend that you start adding your own
 attributes from ATTRIBUTE21.

Related Topics

• Developing the Look and Feel of the Drawing: An Overview, page 225

Edit the North Arrow on Isometric Drawings

- 1. Right-click a Piping Isometric Drawing component or package, and click **Edit Options**.
- 2. Open the **Drawing Frame.Attributes** category.
- 3. Click in the grid, and then press **Insert** to add a new row, or use the **Insert Row** command on the toolbar.
- 4. In the **Host Attribute** column, select **NORTH ARROW SYMBOL**.
- 5. In the **XPos** and **YPos** boxes, specify the X- and Y-coordinates of the north arrow.



- To turn the north arrow off, type **0** in the **XPos** and **YPos** boxes.
- 6. In the **CharHeight** and **CharWidth** boxes, specify the height and width.
- 7. Save the options, and extract some drawings to test.

Notes

• It is not necessary to map the North Arrow Symbol attribute in the **Attribute Map** category.

Related Topics

• Set Drawing Frame Options, page 225

Set Drawing Dimension Options

Using the **Drawing.Dimensions** options in the **Isometric Style Options Browser**, you can specify how dimensions are used and displayed within the drawing. The options include such things as how dimensions are rounded, limits at which to suppress dimension display, and how coordinates are displayed. For example, to turn on the dimensioning for tapped branches on piping, you would do the following:

- 1. Right-click a Piping Isometric Drawing component or package, and click **Edit Options**. The **Isometric Style Options Browser** appears.
- 2. Expand **Drawing.Dimensions**.
- 3. Change the **TapOnPipe** option as needed. You have three setting from which to select: None, Full, or Pipe Only.
- 4. Click **Save To Catalog** to save the changes to the Catalog database. You must have write permission to reference data to save changes to the isometric style.
- 5. Update the document to see the changes made to the drawing dimensions.

Notes

• For more information on drawing dimension options, see *Dimensions* (*Drawing*), page 414.

Related Topics

• Developing the Look and Feel of the Drawing: An Overview, page 225

Control Drawing Content

Using the **Drawing.Content** options, you can specify the content of the isometric drawing. The options allow you to set such things as turning on and off specific coordinates or excluding certain items from the drawing. For descriptions of these options, see *Content (Drawing)*, page 399.

For example, if you want to exclude continuation parts from the drawing, you would set the following:

- 1. Right-click a Piping Isometric Drawing component or package, and click **Edit Options**. The **Isometric Style Options Browser** appears.
- 2. Expand **Drawing.Content**.
- 3. Change the **ExcludeContinuationParts** option to **True**.
- 4. Click **Save To Catalog** to save the changes to the Catalog database. You must have write permission to reference data to save changes to the isometric style.

When you update your drawing document, the software excludes the continuation parts from the drawing content.

Related Topics

• Developing the Look and Feel of the Drawing: An Overview, page 225

Set Drawing Control Options

You use the **Drawing.Control** options to set system controls for the isometric drawings. The options included within the **Isometric Style Options Browser** include such things as:

- **DrawingSize** Specifies a paper size for the drawing, including European and ANSI sizes.
- IsoScale Controls the finished isometric drawing size. You can increase
 or decrease the final plotted size from the size defined by DrawingSize, or
 CustomHeight and CustomWidth.
- **NoSymbolMapOK** Continues to extract the isometric pipeline if the software cannot find the SKEY of a component.

For more information on drawing control options, see *Controls (Drawing)*, page 403.

Related Topics

• Developing the Look and Feel of the Drawing: An Overview, page 225

Specify Drawing Format

The **Drawing.Format** options include such things as text size and skew dimensions. You use the **Isometric Style Options Browser** to specify drawing format. For example, you could do the following:

- Specify the style of the enclosure box for flange rotation angles is Round Ends with the **FlangeRotationStyle** option.
- Set the **NorthArrowBox** option to **True** to display the north arrow inside its own box.

For more information on drawing format options, see Format (Drawing), page 423.

Related Topics

• Developing the Look and Feel of the Drawing: An Overview, page 225

Creating Custom Symbols for Isometric Drawings: An Overview

You can easily create customized symbols for your isometric drawings. Alias, the company that creates the ISOGEN software, provides a free symbol editor utility that you can use to create and modify symbols.

After creating the symbol file, you associate the file with an isometric style and test it by extracting isometric drawings.

Related Topics

• Create a Custom Symbol for Isometric Drawings, page 230

Create a Custom Symbol for Isometric Drawings

- 1. Visit the Alias <u>web site</u> (http://www.alias.ltd.uk) to download and install the free ISOGEN Editor utilities.
- 2. Click **Start > Programs > Alias Isometrics > Iso Utils > Symbols Editor** to launch the symbol editor software.
- 3. In the combo box on the ribbon, select the type of item you want to redefine. For example, select **Redefining Valves**.
- 4. Click **Symbol > New**.
- 5. On the **Add New Symbol** dialog box, specify options for the new symbol.

→ Tips

- Click the ellipsis button beside the Old Symbol Key box to browse through a list of symbols to use as a basis for the symbol. For example, you can select Gate Valve.
- Select the **Copy Symbol from** box and keep the **Library** option selected.
- Click the ellipsis button beside the **Spindle Key** box to browse through the available spindles. For example, select the **01SP** spindle.
- 6. Click Create Symbol.
- 7. Modify the symbol as necessary.

? Tips

• Click the commands on the **Move** menu to change the location of the symbol, the start point (green circle), and the end point (red circle).

- Click in the grid to draw additional lines for the symbol. Click **Undo** on the vertical toolbar at the left to correct any mistakes. Right-click to stop drawing lines.
- When finished, click **Done** on the vertical toolbar at the left.
- 8. Click **File > Save As** and save the symbol file in .*asc* format.
- 9. Click **File > Exit** to close the editor.
- 10. Open the SmartPlant 3D software and switch to the Drawings and Reports task.
- 11. Right-click a Piping Isometric Drawing component or package, and select **Edit Options**.
- 12. Under **Supplementary**, click **DataFiles**.
- 13. In the **DataFileType** column, select **BINARY-SYMBOLS**.
- 14. In the **FilePath** column, click the ellipsis button and browse to the symbols library file (.bin file) you created. If necessary, select **All Files** (*.*) in the **Files of type** field.
- 15. Save the options to the catalog and extract some isometric drawings to test.

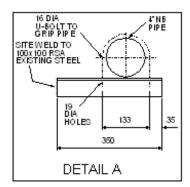
Related Topics

• Creating Custom Symbols for Isometric Drawings: An Overview, page 230

Adding Detail Sketches to Drawings: An Overview

Detail sketches are small inserted drawings that provide additional details for components. The sketches are typically used to show more information about hangers and supports, branch connections, support lugs, and special welds.

To include detail sketches on your drawing, you must prepare symbols in a graphics package such as SmartSketch, AutoCAD, or MicroStation. Then, you can specify options that control various characteristics of the detail sketch, such as the text formatting and placement. You also must map the sketches to part class names of components. If a component belonging to a specified part class is in the drawing, the software prints a callout next to the component, and the sketch is included on the drawing. An example detail sketch follows.



The software delivers example detail sketch templates to each client in the [Product Directory]\Drawings\Templates\DetailSketches folder. These templates provide an easy way to set up detail sketches for testing and review. The sketches are the proper size with correct positioning of labels, so you can save time byre-using these templates when creating your own.

Informational notes are similar to detail sketches. These note refer to pipelines, spools, or components in the drawing. An example informational note follows.



Related Topics

• *Add a Detail Sketch*, page 232

Add a Detail Sketch

- 1. Open the **2D Drawing Editor** by double-clicking the **shape2dserver.exe** file in the \Common2D\Shape2D\Bin folder in the product directory.
- 2. Create a symbol file.

💡 Tip

- Alternatively, open one of the delivered symbol files (.sym format) and modify it. The delivered symbol files are located on each client in the [Product Directory]\Drawings\Templates\DetailSketches folder. Double-clicking these symbol files opens them in the 2D Drawing Editor. You can click the commands on the Help menu to find more information about creating and modifying symbols.
- 3. Save the symbol file.
- 4. Open the SmartPlant 3D software and switch to the Drawings and Reports task.
- 5. Right-click a Piping Isometric component or package, and click **Edit Options**.
- 6. Under **Supplementary**, set the **DetailSketches.ShowDetailSketch** option to **True**.
- 7. Set the **Path** option to the location of the symbol file. For example, set the path to **C:\Program Files\SmartPlant\3D\Drawings\Templates\DetailSketches**. You must add the \character on the end manually.
- 8. Set other options such as the file format, label type, and sketch position.
- 9. Under **Supplementary.DetailSketches.SketchMapping**, map the part classes and symbols.



- You can find a list of part classes in the catalog by switching to the Catalog task and browsing the hierarchy to Drawings > Symbol Map.
- 10. Save the options to the catalog and extract some isometric drawings to test the detail sketches.

Notes

- The software delivers example detail sketch templates in .sym format to each client in the [Product Directory]\Drawings\Templates\DetailSketches folder. These templates provide an easy way to set up detail sketches for testing and review. The sketches are the proper size with correct positioning of labels, so you can save time by re-using these templates when creating your own.
- You also can create the sketch symbol in a graphics package with the capability of exporting to .dxf. Then, open the symbol in the **2D Drawing Editor** to create the symbol file.

Related Topics

• Adding Detail Sketches to Drawings: An Overview, page 232

Configuring the Material List: An Overview

The software contains three different styles for the material lists on isometric drawings. **Fixed Layout** is the default material list for which customization is limited. **Variable Layout** allows you to specify the attributes in the columns of the material list. **User Defined** is the most customizable, allowing full control of the attributes in the columns, the number of sections in the material list, and remarks.

Set Up a Material List, page 234

A material list can contain cut list information. The cut list includes the pieces of cut pipe and their required lengths. You can specify summary files for the material list and cut list.

Set Up a Cut List, page 235

You can also specify labels that correspond to items in the material list.

Specify a Label for the Material List, page 236

Related Topics

• Customizing Isometric Drawing Styles: An Overview, page 207

Set Up a Material List

- 1. Right-click a Piping Isometric Drawing component or package in the **Management Console** and click **Edit Options**.
- 2. Open the **Material List** category.
- 3. Select a style: **Fixed Layout**, **Variable Layout**, or **User Defined**.

? Tips

- **Fixed Layout** is the default material list and is not customizable.
- Variable Layout is a customizable version of the Fixed Layout style
 and allows you to specify the attributes in the columns of the material
 list.
- **User Defined** is the most customizable of the three styles, allowing full control of the attributes in the columns, the number of sections in the material list, and remarks.
- If the **Visible** option under **DrawingFrame** is **False**, you must use the Fixed Layout material list style, because only a Fixed Layout material list can be used with an ISOGEN drawing frame.
- 4. Specify the options for the selected style.

For a summary file of the material list, open the **Material List.Summary File** category and specify the options. The following steps show and example of how to specify the material list summary file:

- 1. Set Material List.Summary File.Enabled to True.
- 2. Set Material List.Summary File.User Defined to True.
- 3. Under **Material List.Summary File.Path**, define a valid path.
- 4. Under **Material List.Summary File.Column**, define values such as the name of the attribute, a width, and the justification. To insert a new row, click in the grid and press **Insert** on the keyboard.

Notes

- The amount of area on the drawing that is reserved for the material list must be set properly, or overlap can occur between the piping symbology and the text of the material list. You can set the reserved area for the material list by setting the **DrawingFrame.ReservedAreaMatList** option.
- By default, bolts and gaskets are accumulated automatically in the material list by diameter. You can deactivate this feature by setting MaterialList.BoltAccumulation or MaterialList.GasketAccumulation to Suppress.
- To display a cut list with the material list on the drawing, open the **MaterialList.CutList** category and specify the necessary options.
- You can specify several different files with material list information, such as the MTO neutral file. For more information, see *Understanding Material List Files*, page 245.
- You can move components from one sort group to another in the materials list by using the **MaterialList.Transfers** category.
- You can also specify labels that correspond to items in the material list using the **Labels.Material List** category.

Related Topics

• Configuring the Material List: An Overview, page 234

Set Up a Cut List

- 1. Right-click a Piping Isometric Drawing component or package in the **Management Console** and click **Edit Options**.
- 2. Open the **Material List.Cut List** category.
- 3. Select a style: **Fixed Layout** or **User Defined**.

? Tips

• **Fixed Lavout** is the default cut list and is not customizable.

- User Defined is a customizable version of the cut list.
- 4. Specify the options for the selected style.

For a summary file of the cut list, open the **Material List.Cut List.Summary File** category and specify the options. The following procedure is an example of how to specify the cut list summary file.

- 1. Set Material List.Cut List.Summary File.Enabled to True.
- 2. Set Material List.Cut List.Summary File.User Defined to True.
- 3. Under **Material List.Cut List.Summary File.Path**, define a valid path.
- 4. Under **Material List.Cut List.Summary File.Column**, define values such as length, size, and cut piece number. To insert a new row, click in the grid and press **Insert** on the keyboard or use the **Insert Row** command on the toolbar.

Related Topics

• Configuring the Material List: An Overview, page 234

Specify a Label for the Material List

- 1. Right-click a Piping Isometric Drawing component or package in the **Management Console** and click **Edit Options**.
- 2. Open the Labels.Material List category.
- 3. Under **LabelAttribute**, select an attribute. You can insert a row by clicking in the grid and pressing **Insert** on your keyboard or use the **Insert Row** command on the toolbar.
- 4. Under **LabelName**, click the browse button, and select an isometric label from the catalog.
- 5. Open the **Material List** category.
- 6. Set the **ActiveList** option to **UserDefined**.
- 7. Open the **MaterialList.UserDefined.Column** category.
- 8. Insert a row for the attribute you added.
- 9. Define the location for the attribute using the **xPosition** and **MaxChars** columns.

Related Topics

• Configuring the Material List: An Overview, page 234

Displaying a Pipeline List: An Overview

You can specify an option to display process or design information from each pipe run on the isometric drawing. This pipeline list is an embedded Excel workbook.

Note

• When defining embedded report layout (sizing of columns and rows), consider the report usage first. Because of a Microsoft limitation concerning the size of Windows metafile objects within other applications, the data displayed may be incomplete. Therefore, no column should be out of screen when using 100 percent zoom for the report. Otherwise some columns are ignored when the report is embedded within the drawing. The same limitation exists for rows. To preserve the maximum number of rows displayed, the total header row(s) height should be a minimum of the overall report. Using Microsoft Excel default settings, the maximum number of columns is approximately 20 and the maximum number of rows is approximately 75 (including header rows). For more information on setting the defaults in Microsoft Excel, see your Microsoft Excel documentation.

Related Topics

• Display a Pipeline List, page 237

Display a Pipeline List

- 1. Right-click a Piping Isometric Drawing component or package in the **Management Console**, and click **Edit Options**.
- 2. Open the **Drawing Frame** category.
- 3. Set the **PipelineList.ShowPipeLineListBox** option to **True**.

Note

• When defining embedded report layout (sizing of columns and rows), consider the report usage first. Because of a Microsoft limitation concerning the size of Windows metafile objects within other applications, the data displayed may be incomplete. Therefore, no column should be out of screen when using 100 percent zoom for the report. Otherwise some columns are ignored when the report is embedded within the drawing. The same limitation exists for rows. To preserve the maximum number of rows displayed, the total header row(s) height should be a minimum of the overall report. Using Microsoft Excel default settings, the maximum number of columns is approximately 20 and the maximum number of rows is approximately 75 (including header rows). For more information on setting the defaults in Microsoft Excel, see your Microsoft Excel documentation.

Related Topics

• Displaying a Pipeline List: An Overview, page 237

Printing Welds on Isometric Drawings: An Overview

To enable weld output on an isometric drawing, you must configure a few items in the software.

First, you must map each type of weld to a symbol key (SKEY). You can complete this mapping by opening the **Isometric Styles Options Browser** for an isometric style and using the **Symbol Map** group of options.

Second, you must set options to show the weld symbols as well as the weld numbers on the isometric drawing. The **ShowWelds** option indicates whether the weld symbol appears on the drawing, and the **ShowWeldNumbers** option indicates if the weld number appears.

You also can specify that a weld list appear on the drawing. The columns shown in the weld list can be customized. The weld list does not appear if the option setting to display weld numbers is not also enabled. If needed, you can specify that the weld list information is also saved in a summary file.

Related Topics

• Print Welds, page 239

Print Welds

- 1. Right-click a Piping Isometric Drawing component or package in the **Management Console**, and select **Edit Options**.
- 2. Open the **Drawing.Welds** category.
- 3. Set the **ShowWelds** option to **True**.
- 4. Set the **ShowWeldNumbers** option to **True**.
- 5. To display a weld list on the drawing, set the **WeldList.Visible** option to **True**.

Note

• You can save the weld list information to a file by specifying the **WeldList.SummaryFile** options.

Related Topics

• Printing Welds on Isometric Drawings: An Overview, page 239

Specify a Label for the Weld List

- 1. Right-click a Piping Isometric Drawing component or package in the **Management Console**, and select **Edit Options**.
- 2. Open the **Labels.Weld List** category.

- 3. Under **LabelAttribute**, select an attribute. You can insert a row by clicking in the grid and pressing **Insert** on your keyboard or by using the **Insert Row** command on the toolbar.
- 4. Under **LabelName**, click the browse button, and select an isometric label from the catalog.
- 5. Open the **Weld List** category.
- 6. Set the **ActiveList** option to **UserDefined**.
- 7. Open the **WeldList.UserDefined.Column** category.
- 8. Insert a row for the attribute you added.
- 9. Define the location for the attribute using the **xPosition** and **MaxChars** columns.

Related Topics

- *Print Welds*, page 239
- Printing Welds on Isometric Drawings: An Overview, page 239

Specifying Fonts on Isometric Drawings: An Overview

You can change the font that appears on isometric drawings. A font information file (.fif) is delivered in the workstation setup in the [Product

Directory]\Drawings\3rdParty\Alias\ProjectManager\Data folder. You specify this file in the isometric options, and then you can select fonts for the material list and the drawing.

Related Topics

• Select Fonts, page 241

Select Fonts

- 1. Right-click a Piping Isometric Drawing component or package, and click **Edit Options**.
- 2. Open the **Supplementary.Data Files** folder.
- 3. In the **Data File Type** column, type **FONT-INFORMATION-FILE**.
- 4. In the **File Path** column, click the browse button and select the font information file (*.fif*) in the [*Product Directory*]\Drawings\3rdParty\Alias\ProjectManager\Data\folder.
- 5. Under **Drawing.Format**, set the **Text Font** option to the font of your choosing. This option controls the font of the text on the isometric drawing.
- 6. Under **Material List**, set the **Text Font** option to the font of your choosing. This option controls the font of the text in the material list.

Related Topics

• Specifying Fonts on Isometric Drawings: An Overview, page 241

Specifying Flow Arrows: An Overview

Isometric drawings commonly have annotations to denote the direction of fluid flow through pipe as well as through pipe components. To specify flow direction in the model, switch to the Piping task. Select a pipe run and change the flow direction by clicking the circular icons on the run. Then, use the **Isometric Style Options Browser** to define when and where flow arrows are placed on isometric drawings.

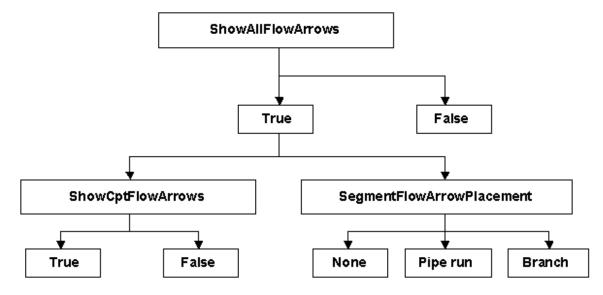
Two overall types of flow arrows exist: segment and component. Segment flow arrows appear on pipe runs, while component flow arrows appear beside components such as valves.

The options for flow arrow placement are located in the **Drawing.Content** and **Format** categories. The **ShowAllFlowArrows** option in the **Format** category toggles all flow arrows on and off.

If the **ShowAllFlowArrows** option is **True**, you can further specify component and segment arrow placement. The **SegmentFlowArrowPlacement** option is set to **None**, **Pipe run**, or **Branch** to denote how arrows are placed on the pipe. The **Pipe run** selection means a flow arrow is placed on each pipe run. The **Branch** selection means a flow arrow is placed on the first leg of each branch, including the header. If the pipeline does not branch, then no additional (there should always be at least one) flow arrow is placed even if one pipe run ends and another begins.

When you configure your isometric styles, you can set the **ShowCptFlowArrows** and **SegmentFlowArrowPlacement** options to match your company specifications. Thereafter, you can control all flow arrows on the isometric drawing by toggling the **ShowAllFlowArrows** option on or off.

The following diagram shows the relationships among the flow arrow options.



Related Topics

• Set Flow Arrow Options, page 243

Set Flow Arrow Options

- 1. Right-click a Piping Isometric Drawing component or package, and click **Edit Options**.
- 2. Under **Drawing.Content**, set the **ShowCptFlowArrows** option.

💡 Tip

- Set the option to **True** if you want component flow arrows to appear for this style. Set the option to **False** if you do not want component flow arrows to appear for this style.
- 3. Under **Drawing.Format**, set the **SegmentFlowArrowPlacement** option.

💡 Tip

- Select **Pipe Run** if you want a flow arrow placed on each pipe run. Select **Branch** if you want a flow arrow placed on the first leg of each branch, including the header. If the pipeline does not branch, then no additional (there should always be at least one) flow arrow is placed even if one pipe run ends and another begins. Select **None** if you do not want any segment flow arrows on the drawing.
- 4. Under **Drawing.Format**, set the **ShowAllFlowArrows** option to either **True** or **False**. This option is a master switch that toggles all flow arrows on or off.

Notes

- When you configure an isometric style, set the **ShowCptFlowArrows** and **SegmentFlowArrowPlacement** options to match your company specifications. Thereafter, you can toggle all flow arrows on or off with the **ShowAllFlowArrows** option.
- When you set the **Drawing.Format.ShowAllFlowArrows** option to **True**, you also must set the **Drawing > Format > SegmentFlowArrowPlacement** option to a value other than **None**.

Related Topics

• Specifying Flow Arrows: An Overview, page 242

Creating Additional Isometric Output: An Overview

You can specify that the software create output files in addition to the isometric drawings. A list of these additional files follows.

- Bending file
- Material take-off (MTO) neutral file
- Material control file
- Printed material list file
- Cut list
- Weld file

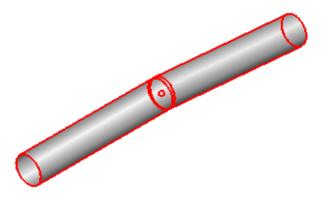
Related Topics

- Understanding Bending Files, page 244
- Understanding Material List Files, page 245
- *Understanding Weld Files*, page 246

Understanding Bending Files

The main purpose of the bending file is to provide an input to a bending machine, which bends the pipe.

Here is an example of a bent pipe and its corresponding bending file information. The pipe configuration is a 2 ft pipe/5 deg bend/2 ft pipe.



	X	Y	Z	Radius	Angle
START	0.0	0.0	0.0	0.0	0.0
BEND1	-607.3	0.0	53.1	0.0	0.0
FINISH	-1216.9	0.0	53.1	0.0	0.0

The START row indicates the start point at 0, 0, 0. The BEND1 row indicates the relative coordinates of the bend location from the start point in millimeters. The distance of 607.3 mm is almost 2 ft (23.909 in). For the FINISH point, the distance of 1216.9 mm is almost 4 ft (47.909 in). The 53.1 mm distance is the absolute elevation between the start and finish points.

You can specify that a bending file is created by using the **Supplementary.Report Files** option for an isometric style. The keyword BENDING_FILE_APPEND places
the bending file information for each extraction consecutively in one file, and the
keyword BENDING_FILE_OVERWRITE erases the file and writes to it with each
extraction.

Related Topics

• Creating Additional Isometric Output: An Overview, page 244

Understanding Material List Files

The software allows you to specify several different types of material list files for isometric drawings. In addition, you can include cut list information.

Printed Material List

The printed material list is a text file that contains the same information as the material list on the face of the drawing. The information is continuously appended to the file. You can specify that a printed material list file is created by using the **Supplementary.ReportFiles** category.

MTO Neutral File

The MTO neutral file can have either a plain text format or an Excel workbook format. The purpose of this file is to provide the extracted information to a material control system. You can control the objects included in the neutral file as well as the columns of information. This file is also called the Intergraph MTO neutral file.

Optionally, weld and bolt data can appear in the MTO neutral file. The data from these components can appear in columns already existing in the MTO neutral file, or you can specify new columns.

The MTO neutral file is set with the **Report.MTONeutralFile** options. For more information, see *Set Styles for the MTO Neutral File*, page 246.

Material Control File

The purpose of this file is the same as the MTO neutral file: to provide information to a material control system. This material control file is an Alias file. You can specify this file by using the **Material List.Summary File** options.

Cut Pipe Report

The cut pipe report provides a list of the pieces of cut pipe and their required lengths. To specify options for a cut list, you use the **Material List.CutList** options.

Related Topics

• Creating Additional Isometric Output: An Overview, page 244

Set Styles for the MTO Neutral File

The following procedure shows how to set styles for an MTO Neutral File using the **Isometric Style Options Browser**.

- Right-click a Piping Isometric Drawing component or package in the Management Console, then click Edit Options to display the Isometric Style Options Browser.
- 2. Expand **NeutralFile** and change option settings as necessary.
- 3. Click **Save to catalog** to save the changes to the Catalog database. You must have write permission to reference data to save changes to isometric styles.
- 4. Update your document to see the new applied label style.

Neutral File Example

- 1. For the **NeutralFile.Enabled** option, select **True** to turn the Neutral File display on.
- 2. For the **NeutralFile.WeldData.Enabled** option, select **True** to display weld data. You can also display **BoltData** and **GasketDAta**.
- 3. To add columns for weld data, expand to **NeutralFile.WeldData.Column** and add column data as needed for your neutral file contents.
- 4. Add columns for bolts and gaskets if needed.
- 5. Save the changes to the catalog and update your document to see the new Neutral File data.

Related Topics

• *Understanding Material List Files*, page 245

Understanding Weld Files

The weld file contains the same information as the weld list on the isometric drawing. You can specify a weld file by enabling the **Weld List.SummaryFile** options.

Related Topics

• Creating Additional Isometric Output: An Overview, page 244

Generic Module Folder: An Overview

The Generic Module Folder component provides a way for you to run your custom VB modules to create custom drawings. You set up the Generic Module Folder component to use your custom VB module. For example, you might create a VB module to create MicroStation 3D drawings, AutoCAD 3D drawings, or Electrical Isometric drawings. The Generic Module Folder is simply a container for your VB modules. You create the component just like other components within the Drawings and Reports task, but when you perform **Setup** on the component, you specify a custom VB module.

The software builds the list of available VB modules from the Symbol share, specifically from \Symbols\Drawings\Catalog\DwgTypeModules. You must store your VB module in this location to have access to it in the Drawings and Reports task.

Custom VB Module

The behavior of the component depends entirely on how the VB module is developed. Everything is controlled by the VB module. When you right click the component, the VB module determines which menu items are available. If the module has a command that creates documents, you see the **Create Documents** command. If, for whatever reason, the module does not need documents, the document-related commands do not appear on the shortcut menu. Other commands the VB module might include are **Rename**, **Delete**, **Refresh**, and **Print**. For more information on general commands available, see *Managing Documents: An Overview*, page 29.

If the module supports publishing, the right-click menu includes the **Publish** command. You need to be setup to work with The Engineering Framework (TEF) if this is the case. For more information, see *Publishing Documents: An Overview*, page 373.

For information on the interfaces used to create a custom VB drawing module, see the *Programmer's Guide*. Contact your administrator or Intergraph Support if you need the *Programmer's Guide*.

- Setup Command (Generic Module Folder Component), page 248
- Use a Generic Module Folder Component, page 249

Setup Command (Generic Module Folder Component)

Sets the custom VB module associated with the Generic Module Folder component. This command is available on the right-click popup menu for the Generic Module Folder component.

Related Topics

- Generic Module Folder: An Overview, page 247
- Use a Generic Module Folder Component, page 249

Setup Dialog Box (Generic Module Folder Component)

Sets the custom VB module for the Generic Module Folder component.

Modules - Specifies a custom VB module. The dropdown list contains a list of the most recently used VB modules. Click **More** to display the **Select Module** dialog box. For more information, see *Select Module Dialog Box (Generic Module Folder Component)*, page 248.

Related Topics

- Generic Module Folder: An Overview, page 247
- Setup Command (Generic Module Folder Component), page 248
- Use a Generic Module Folder Component, page 249

Select Module Dialog Box (Generic Module Folder Component)

Provides a hierarchical list of available VB modules. The hierarchy includes all VB modules located in \Symbols\Drawings\Catalog\DwgTypeModules. This dialog box displays when you select **More** in the **Modules** dropdown on the **Setup** dialog for a Generic Module Folder component.

- Generic Module Folder: An Overview, page 247
- Setup Command (Generic Module Folder Component), page 248
- Setup Dialog Box (Generic Module Folder Component), page 248
- Use a Generic Module Folder Component, page 249

Use a Generic Module Folder Component

The Generic Module Folder is a container component for a custom VB module. This procedure assumes you have created custom VB modules and stored them in \Symbols\Drawings\Catalog\DwgTypeModules.

- Right-click the folder in the Management Console where you want to store the Generic Module Folder component, then select New. The Add Component dialog box appears.
- 2. Select the **Generic Module Folder** component, then click **OK**. The Generic Module Folder component is created in the folder.
- 3. Right-click the **Generic Module Folder** component and select **Setup**.
- 4. On the **Setup** dialog box, specify a custom VB module from the dropdown list. Click **More** to display the **Select Module** dialog box. The **Select Module** dialog box provides a hierarchy of modules available from \Symbols\Drawings\Catalog\DwgTypeModules.
- 5. Click OK to save the changes to the Generic Module Folder component. The new custom component is added to the folder in the **Management Console**.
- 6. Right-click the new custom component and use the available commands to perform operations on the component. The commands available depend entirely on the functionality provided in the VB module.

- Generic Module Folder: An Overview, page 247
- Select Module Dialog Box (Generic Module Folder Component), page 248
- Setup Command (Generic Module Folder Component), page 248
- Setup Dialog Box (Generic Module Folder Component), page 248

Generic Module Folder: An Overview					
250 Drawings and Reports User's Guide					

Spreadsheet Reports: An Overview

The Spreadsheet Report component allows you to define report templates and create report deliverables.

The software features a tabbed editor called the **Report Editor** that allows you to create and modify report templates. Most of the delivered reports use a combination of filters and SQL. The **Report Editor** includes an interface for entering database query syntax and testing the results of a query as part of the report. The editor also features an Excel component in which you can drag and drop attributes onto a worksheet. The default format of all reports is Microsoft Excel format.

After you make changes to a template, you can test the template by using the **Update Document(s)** or **Update Now** command. For more information on updating documents, see *Updating Documents: An Overview*, page 337.

There are several types of spreadsheet reports delivered with the software. They are defined and described in the **Reports.xls** workbook. For more information on the workbook, see the *SmartPlant 3D Drawings and Reports Reference Data Guide*.

- Create and Update a Delivered Report, page 266
- Spreadsheet Reports Common Tasks, page 260
- Understanding Report Templates: An Overview, page 254
- Understanding the Reporting Process: An Overview, page 252

Understanding the Reporting Process: An Overview

The reporting process involves the effort of three people:

- A reports database administrator who creates the reporting databases
- A reports designer who creates report templates that define what is reported and in what format
- An end user who runs the reports

Role of the Reports Database Administrator

The report database is created when you create the model. The reports database administrator is responsible for the databases, including regeneration when necessary. The reports databases work in conjunction with the five databases required by the software. For more information, see the *Project Management User's Guide* and the *Installation Guide* available from **Help > Printable Guides**.

The administrator must re-create the reports databases anytime the metadata changes, such as when new attributes or custom interfaces are added to the reference data or when changes are made to the catalog schema.

Note

• The software requires the Reports Database and permissions be set correctly to make the reporting functionality available.

Role of the Reports Designer

Reports designers can be classified in three categories: Casual, Expert, and Consultant. All three types of reports designers create and modify report templates, but the manner in which they do so may differ. All three types of designers use the **Report Editor** to create report templates. The Casual user creates filter-based reports, but an Expert user may choose to write SQL query-based reports. A Consultant creates and modifies report files (.*rtp*, .*rqe*, .*rfm*) as well as using the **Report Editor** to develop complex queries. The Expert and Consultant designers are more skilled in the use of SQL and the complexities of the data model.

To view the delivered report templates and examples, see the \Symbols\Reports\Types of Reports folder located on the product server computer. The reports are divided further into component folders. For example, the Piping-related reports and all their supporting files are located in the Piping folder. The reports in this location are catalog reports and are bulkloaded into the Catalog database.

The delivered spreadsheet reports are defined and described in the **Reports.xls** workbook. For more information on the workbook, see the *SmartPlant 3D Drawing* and *Reports Reference Data Guide*.

Role of the End User

End users can interactively run reports in most of the tasks in the software using the **Run Report** command on the **Tools** menu. Users can select a report template listed on the **Catalog Reports** tab or the **My Reports** tab on the **Run Report** dialog box and then run a report based on that template.

End users also answer prompts for parameterized reports when they are run. The prompt questions are specified when these types of reports are designed.

End users can access personal report templates, stored in the **Personal Report Templates** location specified on the **Tools > Options** dialog box. Administrators or report designers are usually responsible for creating catalog report templates.

For more information about running reports, see the *Common User's Guide* available from the **Help > Printable Guides** command in the software.

! Important

• The **Run Report** command is intended for quickly generating reports that will not be updated in the future. To manage report generation, you must use the reporting functionality in this task.

In addition to the three different roles involved in reporting, two different time periods exist in reporting: design time and run time. Design time refers to the part of the process when the report templates are specified and edited. Run time refers to the time when the report deliverable is created using the **Tools > Run Report** command or the functionality in this task.

- Create a New Report Template from an Existing Template, page 263
- Create and Update a Delivered Report, page 266
- Edit a Report Template, page 295
- Spreadsheet Reports: An Overview, page 251
- Understanding Report Templates: An Overview, page 254
- Understanding the Report Templates Folder: An Overview, page 256

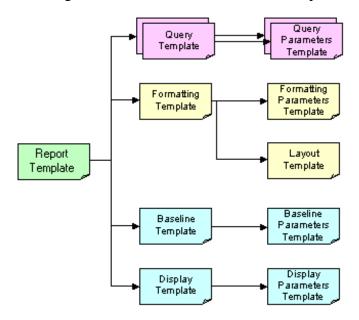
Understanding Report Templates: An Overview

To create report templates, you must be familiar with database query syntax and the data model in the software. You can use the **SP3D Schema Browser** to view the data model. The executable, *MetaDataBrowser.exe*, is delivered in *Program Files\SmartPlant\3D\Programming\tools\Bin*. You need to load the **Programming Resources** to access this executable. For more information, see the *Installation Guide* available from **Help > Printable Guides**.

Typically, a reports designer creates the templates based on the content and format customarily used in the company or project. This task provides a user interface to create or edit report templates.

A report template consists of several sub-template files that together control the data extracted from the databases, the supporting attributes needed, and the output format. The statements in the report templates search the databases and extract the most commonly sought after database information and display it in a useful manner.

The manner in which data is extracted from the database is controlled typically by a query template. The types of templates include report templates, label templates, query templates, query parameters templates, and formatting templates. You can compare the template file configuration to Visual Basic programs; for example, the report template file is analogous to a Visual Basic project file (.vbp), and the other templates are analogous to the files to which the .vbp points: .cls, .bas, and .frm files. The diagram below shows the interrelationships among the types of templates.



Another way to categorize the templates is by the type of access. Templates can be catalog report templates or personal report templates. End users can define the location of personal report templates by clicking **Tools > Options** and selecting the

File Locations tab. The **Tools > Options** command is available in most of the tasks in the software.

To view the delivered report templates and examples, see the \Symbols\Reports\Types of Reports folder located on the SmartPlant 3D server computer. The reports in this location are catalog reports and are bulkloaded into the Catalog database. You can also copy reports directly into the Catalog database. When you copy a report to the Catalog database, the software creates a directory with the report name in the Symbols share. The software also creates all the files associated with the report in that directory. For more information, see Copy Report To Catalog Command (Report Shortcut Menu), page 299.

The delivered spreadsheet reports are defined and described in the **Reports.xls** workbook. For more information on the workbook, see the *SmartPlant 3D Drawing* and *Reports Reference Data Guide*.

- Create a New Report Template from an Existing Template, page 263
- Edit a Report Template, page 295
- Understanding the Report Templates Folder: An Overview, page 256
- Understanding the Reporting Process: An Overview, page 252
- Using Queries to Extract Data for Reports: An Overview, page 258

Understanding the Report Templates Folder: An Overview

The following list provides descriptions of the report templates folders located in [Product Directory]\CatalogData\Symbols\Reports\Types of Reports on the SmartPlant 3D server computer. Report templates contain the necessary SQL statements and links to other templates to extract the required information and produce an output report.

Baseline - Produces a baseline report, which is used with a differential report. A differential report shows changes in comparison to a baseline report. For example, you can view which items have been added, deleted, or modified between the two reports. Baseline information is not currently editable.

Baseline Parameters - Defines the baseline prompts presented to the user when the report runs. Examples include currency or dates for reporting purposes.

Display - Displays or publishes the formatted result of a report. Display information is not currently editable.

Formatting - Formats the results of queries for reports. This type of template can also refer to formatting parameters if the formatting requires parameters.

Formatting Parameters - Defines the formatting prompts presented to the user when the report runs. Examples include units of measure and coordinate systems.

Labels - Creates labels, such as piping component labels that appear on isometric drawings. Labels are mini-reports. You can have only one query for labels. The display template is not used for labels.

Layout - Provides sample report layouts in Microsoft Excel (.xls) format. The templates provide examples of column header names and overall output organization.

Queries - Contains the SQL statements or property-based queries that retrieve data during report creation. This type of template can also refer to query parameters if the query requires parameters. For more information about report queries, see *Using Queries to Extract Data for Reports: An Overview*, page 258.

Query Parameters - Defines the query prompts presented to the user when the report runs. For example, a prompt can ask the user for the system parent name.

Reports - Creates reports. The report templates contain references to other templates, such as query and formatting templates. You can have several queries for report templates.

Schema - Contains XML code that provides a schema for reporting entities and templates.

The following table gives the associated file type for each report template.

Report Template	File Extension
Baseline	.rbl
Baseline Parameters	.rbp
Display	.rdy
Formatting	.rfm
Formatting Parameters	.rfp
Labels	.rtp
Layout	.xls
Queries	.rqe
Query Parameters	.rqp
Reports	.rtp
Schema	.xml

- Create a New Report Template from an Existing Template, page 263
- Edit a Report Template, page 295
- Understanding Report Templates: An Overview, page 254
- Using Queries to Extract Data for Reports: An Overview, page 258

Using Queries to Extract Data for Reports: An Overview

Data for reports can be extracted from the software in a variety of ways:

- With queries to the Reports database. The Reports database contains unions of views on the other databases (Site, Model, Catalog, Site Schema, and Catalog Schema). Although these queries retrieve data from the model and catalog, they actually query the Reports database instead of going directly to the other databases. Most delivered reports query the Reports database when users run the reports from this task or from the Tools > Run Reports command in other tasks. The Reports database is also used to return data when a report is updated using the Update Document(s) or Update Now command and when the report query is executed through the Edit Template command.
- With queries containing filters that use bound objects to retrieve data directly from the model databases rather than the Reports database.
- By using the drawing that contains an embedded report to define the data the report includes.
- With software written by consultants to gather data for a report.

Regardless of how reports extract data from the software, most reports contain one or more query templates that use either SQL statements or property-based queries to select data to include in the report. The sections that follow describe these two types of queries.

SQL Queries

SQL queries rely on SQL statements to retrieve data. You can write your own SQL queries or you can customize those delivered with the software to retrieve the information you want.

SQL queries can use filters in combination with SQL statements to select data from the database. If you want a particular SQL query to use a filter, the SQL statement must include the following:

```
WHERE oid in (select oid from @RPTtemptable)
```

The oid column name may require qualification, depending on the rest of the statement. However, the SQL variable @RPTtemptable must be used exactly as it is here.

SQL queries can require parameters that the user must supply when he runs the report. To specify parameters for a SQL query, you can do one of the following:

- Use an existing report template that includes a query with parameters as the basis for your report
- Add a query that contains parameters to your report using the Tools >
 Add Query command

If you change the parameters for a report template or add a query with parameters to the template, you can define parameters for an existing report based on that template by right-clicking the report in the **Detail View** and then clicking **Parameters** on the shortcut menu.

- Create a New Report Template from an Existing Template, page 263
- Edit a Report Template, page 295
- Understanding Report Templates: An Overview, page 254
- Understanding the Report Templates Folder: An Overview, page 256

Spreadsheet Reports Common Tasks

The following tasks are used frequently when you create spreadsheet reports.

Create a Spreadsheet Reports Component

You create a Spreadsheet Reports component using **New** command. For more information, see *Add a Component*, page 36.

Create the Report or a New Report Template

Right-click on the Spreadsheet Report component and select **Create Report** to specify the type of report and create it. For more information, see *Create a New Report Template from an Existing Template*, page 263 and *Create and Update a Delivered Report*, page 266.

Edit the Report

Right-click a report and select **Edit Template** to activate the **Report Editor** and customize the report. For more information, see *Report Editor*, page 268. To customize the appearance of the report, go to the **Formatting Tab** and click **Design Layout** to open the report in Excel and modify the appearance and content. For more information, see *Designing Report Layout: An Overview*, page 278.

Add a Query to a Report

You can add a query to a report. For more information, see *Add a Filter-Based Query to a Report Template*, page 308.

Remove a Report Template Component

A report component is represented as a tab in the Report Editor when editing a report template. You use the **Tools > Remove**: *Component* command. For more information, see *Remove Report Component Command (Tools Menu)*, page 315.

Update the Report

Update your report or multiple reports using the Update commands. For more information, see *Updating Documents: An Overview*, page 337.

Setting Report Properties

You can set report properties with the **Properties** command. For more information, see *Edit Document Properties*, page 334.

Copying a Report to the Catalog

Copying a new or edited report to the catalog allows other users to have access to the report. The report is added to the **Report** hierarchy in the Catalog task and can be run using the **Run Report** command in other 3D tasks. For more information, see *Copy a Report Template to the Catalog*, page 301.

Save a Report Template

You have two options for saving report templates. You can save a modified report template and its components to the Catalog using the **File > Save Report Template** command. For more information, see *Save Report Template Command (File Menu)*, page 302. You can also save a selected report template and components to a location that you specify and change the names of report templates and their components before saving. For more information, see *Save a Report Template to a Specified Location*, page 305.

Publish the Report

You can publish Spreadsheet Reports if you are registered with The Engineering Framework (TEF). For more information, see *Publishing Documents: An Overview*, page 373.

Note

• The viewable files created when you publish drawings and reports provide relationship links to the 3D Model Data. You must also publish the 3D Model Data to provide the navigation between the viewable files and the 3D Model Data. For more information, see 3D Model Data: An Overview, page 317.

Create Report Command (Report Shortcut Menu)

Specifies a report template and then generates a report. If the template requires parameters or a filter, you must specify these inputs. Right-click a spreadsheet report component and select **Create Report**.

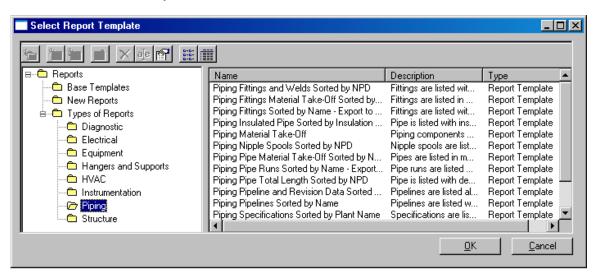
When you create a report, it is added to the **Detail View**. You can edit, print, or publish the report after updating it.

Related Topics

- Create a New Report Template from an Existing Template, page 263
- Create and Update a Delivered Report, page 266
- Spreadsheet Reports: An Overview, page 251

Select Report Template Dialog Box

Selects a report template. This dialog box appears when you click the **Create Report** command for a spreadsheet report component or when you click **Place Report** when a drawing template is open. By browsing through the hierarchy, you can find any report template in the Catalog database. Once you select a template, the software generates the report. You can resize the dialog box and the columns to view the information more clearly.



- **Properties** Displays the properties of the selected item. All properties on the **Properties** dialog box are read-only.
- List View Sets the dialog box to display items in a list view.
- Grid View Sets the dialog box to display items in a spreadsheet-style grid view.



• The **Create Report** and **Place Report** commands create reports based on a selected report template. You can also select a report template and view its properties. The buttons that are grayed out are not available when using these commands.

Related Topics

- Create and Update a Delivered Report, page 266
- Create Report Command (Report Shortcut Menu), page 262
- Spreadsheet Reports: An Overview, page 251

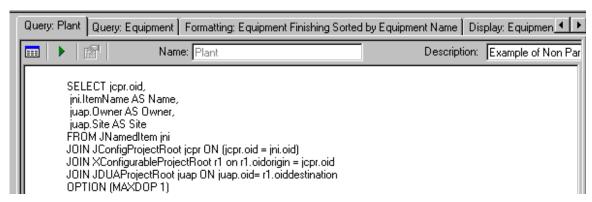
Create a New Report Template from an Existing Template

To create a new report template, you can create a report based on an existing template and then make changes to the template. After your changes are made, you can save the existing template with a new name to create a new template.

- 1. In the **Management Console**, right-click a spreadsheet report component, and click **Create Report** on the shortcut menu.
- 2. On the **Select Report Template** dialog box, click a report template, and click **OK**.

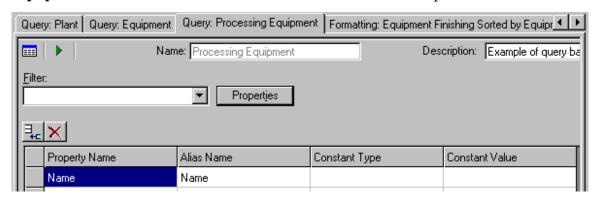


- To create a new filter-based report template that returns properties, select one of the delivered reports that contains a property-based query.
- To create a new report template with a SQL query, select one of the delivered SQL-based reports.
- 3. In the **Detail View**, right-click the report you created and click **Edit Template**. For example, you could create a report using the report template **Equipment Finishing Sorted by Equipment Name**.



4. Use the arrows **1** at the top of the editor to scroll through the Report editor tabs.

5. The **Query** tabs always appear first in the tab order. In the example above, notice that there are two query tabs already available. To add a new query, click **Tools** > **Add Query** and select the existing query you want to use as a starting point for your new query. When you add a query, the software adds it as a tab to the Report Editor. For example, you could add a filter-based query called **Processing Equipment** and a tab of the same name would be added to the Report Editor.



To set the properties on a filter-based query, see *Add a Filter-Based Query to a Report Template*, page 308.

6. To modify SQL queries, modify the SQL statements on the appropriate **Query** tabs to query the databases.

? Tips

- If you are familiar with SQL syntax and the data model, you can delete
 the existing SQL statements, and type a new SQL query or combine
 two SQL queries to create a single query.
- If you want the SQL query to use a filter, your SQL statement must include the following:

```
WHERE oid in (select oid from @RPTtemptable)
```

The oid column name may require qualification, depending on the rest of the statement. However, the SQL variable @RPTtemptable must be used exactly as it is here.

- If the SQL query contains parameters, click **Edit Query Parameters** to make changes to the query prompts presented to the user when the report runs.
- 7. To modify property-based queries, specify a different filter or create a new filter in the **Filter** box on the appropriate **Query** tabs.

→ Tips

- You can add properties to include in the query by clicking **Add**.
- You can remove properties from the query by clicking a row in the table, and then clicking **Remove**.

8. Test the query by clicking **Execute Query** . To see the results of the test, click **Show Results Panel**.

💡 Tip

- If you are creating a differential report, use Tools > Add Baseline to add a Baseline tab for the differential report.
- 9. On the **Formatting** tab, specify the layout of the report by clicking **Design Layout** . The software opens a window in Excel.

♀ Tip

- If you have already opened the Excel layout for the template, the software asks if you want to overwrite it in the temporary location.
- 10. Select an attribute on the left, click **Insert**, and then click in the Excel window to place the attribute. You can also drag and drop the attribute from the left pane to the Excel sheet.
- 11. Right-click the query in the left pane, and click **Properties** to specify layout properties such as grouping, sorting, and comparing.
- 12. Exit Excel, and save the workbook when prompted.
- 13. To modify the layout formatting parameters, click **Edit Formatting Parameters** on the **Formatting** tab.

💡 Tip

- You can also open the **Formatting Parameters** dialog box from the left pane in Excel by right-clicking the **Formatting Parameters** item, and selecting **Properties**.
- 14. On the **Formatting Parameters Designer** dialog box, set attributes for each field on the **Fields** tab.
- 15. Specify information for the units of measure on the **Units of Measure** tab.
- 16. Specify information for the coordinate systems on the **Coordinate Systems** tab.

→ Tip

- In the context of reporting, coordinate systems are also known as matrices. You can see this term in some of the templates if you open them in Notepad.
- 17. Save your changes to the report template.
- 18. To copy the report template to the database, use the **Tools > Copy to Catalog** command.

Copy a Report Template to the Catalog, page 301

Notes

• The software combines the options you set on all the tabs to form the report template. You can add tabs by clicking the commands on the **Tools**

- menu. Each report template can have multiple queries and baselines, but only one format and one display.
- If you want your report templates to be available to other users, you must make changes to the names and locations of report templates on the **Report** sheet of the **Reports.xls** spreadsheet and then bulk load the reference data into the Catalog database. All of the report files (.rtp, .rqe, etc.) must be saved to a directory to which the users have access (like the Symbols share).
- Do not overwrite delivered templates. Save new or modified template files with unique names different from the templates delivered with the software.

Related Topics

- Edit a Report Template, page 295
- Spreadsheet Reports: An Overview, page 251
- Understanding Report Templates: An Overview, page 254
- Using Queries to Extract Data for Reports: An Overview, page 258

Create and Update a Delivered Report

1. In the **Management Console**, verify that at least one spreadsheet report component exists. If none exists, add a component for reports.

Add a Component, page 36

- 2. Right-click the spreadsheet report component, and click **Create Report** on the shortcut menu. For more information, see *Create Report Command (Report Shortcut Menu)*, page 262.
- 3. On the **Select Report Template** dialog box, click a report template, and click **OK**. The software creates the report document in the **Detail View**.



- The software prompts for parameters or a filter if the selected report template requires these inputs.
- 4. Right-click the report, and click **Update Document(s)**. The software generates the report. For more information, see *Updating Documents: An Overview*, page 337.

Note

You can modify the report template if needed. Right-click the report in the Detail View, and click Edit Template. You can add tabs using commands on the Tools menu, and you can save the report template using File > Save Report Template. For more information, see Edit a Report Template, page 295.

- Spreadsheet Reports: An Overview, page 251
- Understanding Report Templates: An Overview, page 254

Edit Template Command (Report Shortcut Menu)

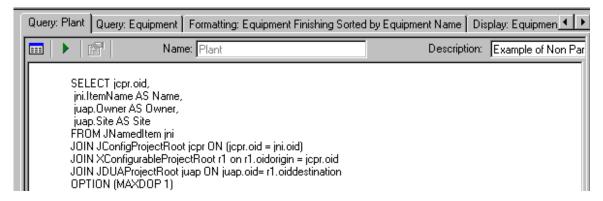
Modifies a template for a report. This command opens the **Report Editor**, which includes several tabs you use to set query, formatting, display, and baseline options.

Related Topics

• Understanding Report Templates: An Overview, page 254

Report Editor

Sets options for report templates. The software combines the options you set on all the tabs to create the report template. The Report Editor is available by clicking **Edit Template** on the shortcut menu that appears when you right-click a report in the Detail View.



You can add tabs by clicking the commands on the **Tools** menu. Each report template can have multiple query tabs and multiple baseline tabs, but only one formatting tab and one display tab.

- Add a Filter-Based Query to a Report Template, page 308
- Create a New Report Template from an Existing Template, page 263
- Edit a Report Template, page 295
- Understanding Report Templates: An Overview, page 254

Query Tab (Report Editor Dialog Box)

Specifies the syntax to retrieve data for the reports. This tab allows you to execute the query and view the results. You can create multiple query tabs for one report template by clicking **Tools** > **Add Query**. For more information on adding queries, see *Add a Filter-Based Query to a Report Template*, page 308.

```
Query: Plant Query: Equipment Formatting: Equipment Finishing Sorted by Equipment Name Display: Equipmen  

Name: Plant Description: Example of Non Par

SELECT jcpr.oid,
jni.ItemName AS Name,
juap. Owner AS Owner,
juap. Site AS Site
FROM JNamedItem jni
JOIN JConfigProjectRoot jcpr ON (jcpr.oid = jni.oid)
JOIN XConfigurableProjectRoot r1 on r1.oidorigin = jcpr.oid
JOIN JDUAProjectRoot juap ON juap.oid= r1.oiddestination
OPTION (MAXDOP 1)
```

Show Results Pane - Displays a pane at the bottom of the tab. When you execute the query, the results appear here.

Execute Query - Runs the query specified on the tab.

Edit Query Parameters - Displays the **Query Parameters Designer** for modifying the query prompts that are presented to the user when he runs the report. For example, a prompt can ask the user for the system parent name. This button is only available if the query is not filter-based.

Name - Displays the name for the query.

Description - Describes the query.

Filter - Displays the name of the filter on which this query is based. This field is only available if the query is filter-based.

Properties - Displays the **Filter Properties** dialog box to change the properties of the filter if necessary. This button is only available if the query is filter-based.

Add - Displays the **Select Properties** dialog box to add a new property to a filter-based query. This button is only available if the query is filter-based.

Remove - Removes a property that was added to a filter-based query. This button is only available if the query is filter-based.

- Create a New Report Template from an Existing Template, page 263
- Edit a Report Template, page 295
- Report Editor, page 268
- Understanding Report Templates: An Overview, page 254

Query Parameters Designer Dialog Box

Sets options for query parameters in labels and reports. This dialog box is accessible in both the Common task and the Drawings and Reports task.

In both tasks, you select a label with query parameters (a SQL label) on the **Label Editor** dialog box. Then, click in the **Properties** area to display the **Query Parameters Designer** dialog box.

You can also view this dialog box if you edit a report template with query parameters in Drawings and Reports.

Name - Provides an internal name for the parameter.

Index - Specifies an index number for the parameter. This number provides a mapping between the prompt that appears at runtime and the query syntax. The reason for this number is the SQL/ADO limitation of positional parameters only.

Caption - Provides a caption for the parameter at runtime.

Prompt Category - Specifies the category of prompt for the query parameter.

Prompt Type - Specifies the type of prompt. This list is based on your selection under **Prompt Category**.

SQL Data Type - Specifies a data type available in the SQL query language.

Default Value - Defines the default value for the parameter.

Displayed - Specifies whether or not the parameter appears on the report.



• You can move the parameters up and down in the list on the dialog box by clicking **Move Up** and **Move Down**.

Select Properties Dialog Box

Browses the data model and selects properties on types. You use the **Select Properties** dialog box when specifying filter properties for the workspace and when defining labels. This dialog box is accessible in both the Common task and the Drawings and Reports task.

In Common, you can access this dialog box when you use the **File > Define Workspace** command or the **Tools > Select by Filter** command to view the properties of a filter. On the **Filter Properties** dialog box, click the **Properties** tab, and in the **Property** column, click **More**. You can also access this dialog box by

clicking **Tools > Options** and selecting the **ToolTips** tab. Click **Edit Tooltip**, and then click **Add** in the **Properties** section.

In Drawings and Reports, you can access this dialog box when you use the **Edit Template** command on a report template to add properties to a filter query.

Object type used as the basis for the property identification - Specifies an object type. Click **More** to access the data model tree. For more information, see *Select Object Type Dialog Box*, page 271.

Relationship - Specifies a direct property or a correlation between object types.

Related object type - Selects another object type. Click **More** to access the data model tree.

Display properties in this category - Specifies a category. You can define categories in the reference data workbooks on the Custom Interfaces sheets.

Select one or more properties - Specifies properties. Press SHIFT to select more than one property.

Related Topics

• Select Object Type Dialog Box, page 271

Select Object Type Dialog Box

Specifies the categories of objects, the feature type, and the component to which you want to add a ToolTip or label. This dialog box is accessible in both the Common task and the Drawings and Report task.

In the Common task, you can access this dialog box when you are specifying an object type for filter properties. You can also access this dialog box when you are editing labels for ToolTips.

When you are working with filter properties or labels, this dialog box opens after you click **More** in the **Object type** box on the **Select Properties** dialog box. When you are working with ToolTips, the **Select Object Type** dialog box opens after you click the browse button on the **ToolTips** tab on the **Options** dialog box.

In the Drawings and Reports task, you can access this dialog box when you use the **Edit Template** command on a report template to add properties to a filter query.

When the **Select Object Type** dialog box opens, a tree view lists categories of objects available in the software. When you double-click an object, the view expands to show the available feature types. After you select a feature type, another level is available for some categories to show the component features you can select. For example, double-click **HVAC**, double-click **HVAC Features**, and then click **HVAC Transition** as the feature, and accept the dialog box.

If objects appear in italics, you cannot select that object on this dialog box. The software uses your previous selections as the basis for this determination. *Italicized* text for objects in the **Workspace Explorer** indicates the objects are hidden with the **Show/Hide** options.

Related Topics

• Select Properties Dialog Box, page 270

Formatting Tab (Report Editor Dialog Box)

Formats the results of queries for reports. This tab allows you to place attributes on an Excel sheet in the layout that you want. You can have only one Formatting tab for each report template.



Design Layout - Opens an Excel window in which you can drag and drop attributes. For more information, see *Designing Report Layout: An Overview*, page 278.

Edit Formatting Parameters - Opens a dialog box to specify fields and units of measure and coordinate systems information. For more information, see *Formatting Parameters Designer Dialog Box*, page 273.

Name - Specifies a name for the formatting template.

Description - Describes the formatting template.

Layout Template - Displays the name of the Excel template used for the layout of the report.

- Create a New Report Template from an Existing Template, page 263
- Edit a Report Template, page 295
- Report Editor, page 268
- Understanding Report Templates: An Overview, page 254

Formatting Parameters Designer Dialog Box

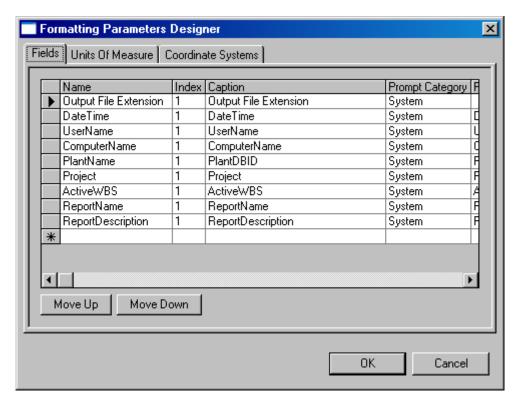
Sets options for formatting parameters. You access this dialog by clicking **Properties** on the **Formatting** tab of the Report Editor.

Related Topics

- Edit a Report Template, page 295
- Understanding Report Templates: An Overview, page 254

Fields Tab (Formatting Parameters Designer Dialog Box)

Specifies various attributes for each parameter. You can move the parameters up and down in the list by selecting them in the table and clicking **Move Up** and **Move Down**.



Name - Provides an internal name for the parameter.

Index - Specifies an index number for the parameter. This number provides a mapping between the prompt that appears at runtime and the query syntax. The reason for this number is the SQL/ADO limitation of positional parameters only.



• By using the parameter index, you can use the same value multiple times in a query while prompting the user only once. For example, in a query

where the same input parameter is used in more than one select statement, use the parameter index (Parameter A index 1, Parameter A index 2, Parameter A, index 3, and so forth).

The index for the first definition of the parameter should be one. For additional definitions, you can increase the parameter index. The software warns you if you use an index greater than one for a unique parameter.

Caption - Provides a caption for the parameter at runtime.

Prompt Category - Specifies the category of prompt for the parameter.

Prompt Type - Specifies the type of prompt. This list is based on your selection under **Prompt Category**.

SQL Data Type - Specifies a data type available in the SQL query language.

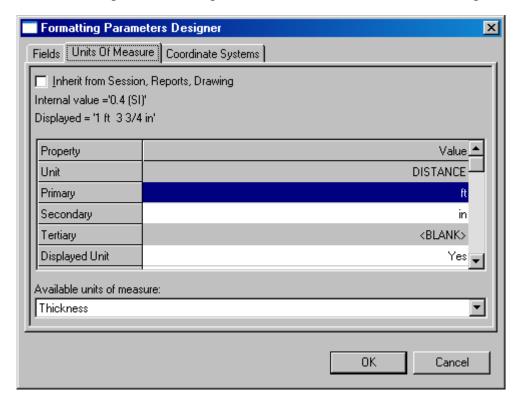
Default Value - Defines the default value for the parameter.

Displayed - Specifies whether or not the parameter appears on the report.

- Edit a Report Template, page 295
- Formatting Parameters Designer Dialog Box, page 273
- Understanding Report Templates: An Overview, page 254

Unit of Measure Tab (Formatting Parameters Designer Dialog Box)

Specifies properties for units of measure. This dialog box is accessible in the Drawings and Report task. You can set the default value for units of measure that are added to the report in a label specified with the **Inherit from Session** option.



Inherit from Session, Reports, Drawing - Indicates that the parameter should use the same units used in the session, the report, or the drawing.

Unit - Displays the type of unit, such as distance.

Primary - Specifies the primary unit of measure, such as yards in yards, feet, inches.

Secondary - Specifies the secondary unit of measure, such as feet in yards, feet, inches.

Tertiary - Specifies the tertiary unit of measure, such as inches in yards, feet, inches.

Displayed Unit - Turns the display of the units on and off.

Precision Type - Specifies **Decimal**, **Fractional**, or **Scientific**. Your selection in this box determines the availability of the remaining boxes on this dialog box.

Decimal Precision - Specifies the number of places after the decimal point. This value can be 0 or greater.

Leading Zero - Places a zero before the decimal point, if applicable.

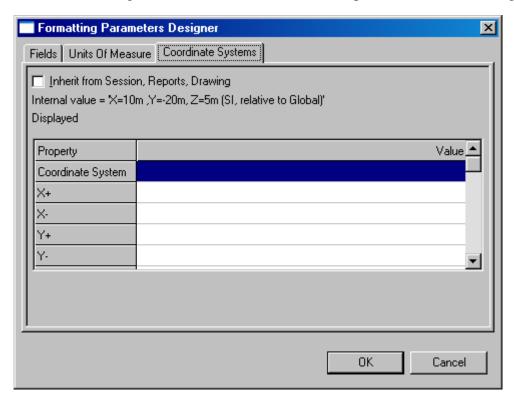
Trailing Zeros - Places zeros after the last significant digit, if applicable.

Fractional Precision - Specifies a fraction for the precision. The highest value that you can specify is 1/2. This box is available only if you select **Fractional** in the **Precision Type** box.

Reduce Fraction - Reduces the fraction. For example, displays 3/4 instead of 6/8. This box is available only if you select **Fractional** in the **Precision Type** box.

Coordinate Systems Tab (Formatting Parameters Designer Dialog Box)

Specifies information about the coordinates. You can select an option to inherit the coordinates settings from the session file, from the report, or from the drawing.



Inherit from Session, Reports, Drawing - Indicates that the parameter should use the same coordinate system used in the session, the report, or the drawing.

Coordinate System - Specifies either the global coordinate system or the plant coordinate system. For more information about coordinate systems, see the *Grids User's Guide* available from the **Help > Printable Guides** command in the software.

X+ - Specifies the positive X-axis. For example, you can type **East**.

- **X-** Specifies the negative X-axis. For example, you can type **West**.
- Y+ Specifies the positive Y-axis. For example, you can type **North**.
- Y- Specifies the negative Y-axis. For example, you can type **South**.
- **Z**+ Specifies the positive Z-axis. For example, you can type **Up**.
- **Z-** Specifies the negative Z-axis. For example, you can type **Down**.

XY Axis Reference - Sets a reference to the X- or Y-axis.

Related Topics

- Edit a Report Template, page 295
- Formatting Parameters Designer Dialog Box, page 273
- Understanding Report Templates: An Overview, page 254

Display Tab (Report Editor Dialog Box)

This tab is not currently available.

Related Topics

- Create a New Report Template from an Existing Template, page 263
- Edit a Report Template, page 295
- Report Editor, page 268
- Understanding Report Templates: An Overview, page 254

Baseline Tab (Report Editor Dialog Box)

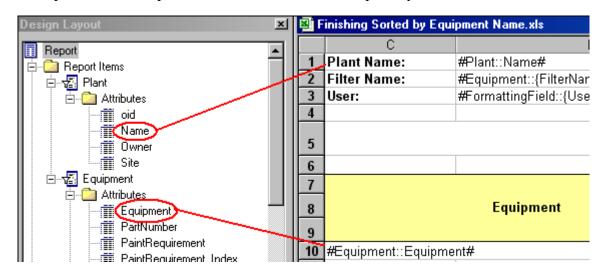
This tab is not currently available.

- Create a New Report Template from an Existing Template, page 263
- Edit a Report Template, page 295
- Report Editor, page 268
- Understanding Report Templates: An Overview, page 254

Designing Report Layout: An Overview

The actual design of a report layout is done within Microsoft Excel. You access this functionality by clicking **Design Layout** on the **Formatting** tab of the **Report Editor**.

The report Design Layout displays within Excel, allowing you to configure properties associated with the layout of the report. You can use Excel commands to modify the layout as necessary, adding color or other graphical definition to make the report meet your corporate standards. You can change the header information, drag and drop additional fields or parameters onto the report, and change the organization of the attributes. The items shown in the **Design Layout** window represent the contents of the report, while the spreadsheet window shows the report layout.



You can edit the properties of the report by right-clicking the Report node at the top of the **Design Layout** hierarchy. You can also edit the properties for report items by right-clicking them. For more information on setting report level properties, see *Properties Command (Shortcut Menu in Microsoft Excel)*, page 280.

Other commands that may prove helpful in layout of a report are the Expand All and Collapse All commands. For more information, see *Expand All Command (Shortcut Menu in Microsoft Excel)*, page 294 and *Collapse All Command (Shortcut Menu in Microsoft Excel)*, page 295.

When you are finished modifying the design layout, save your changes and close Excel.

After you make changes to a template, you can test the template by using the **Update Document(s)** or **Update Now** command.

- Collapse All Command (Shortcut Menu in Microsoft Excel), page 295
- Expand All Command (Shortcut Menu in Microsoft Excel), page 294
- Formatting Tab (Report Editor Dialog Box), page 272
- Properties Command (Shortcut Menu in Microsoft Excel), page 280

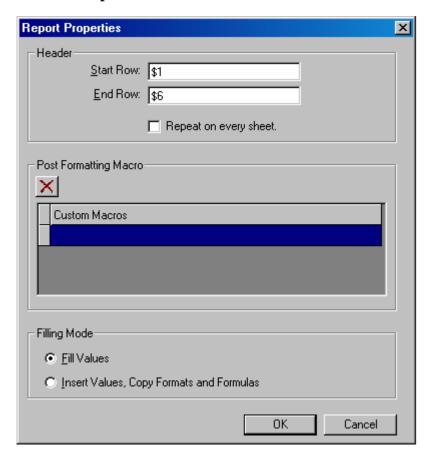
Properties Command (Shortcut Menu in Microsoft Excel)

Sets report and item properties as well as query and formatting parameters, if applicable. You access the **Properties** command by right-clicking on an item in the **Design Layout** window in Excel. The **Design Layout** is displayed in Excel when you click **Design Layout** on the **Formatting** tab of the **Report Editor**.

- Designing Report Layout: An Overview, page 278
- Edit a Report Template, page 295

Report Properties Dialog Box

Sets options for the whole report. You can open this dialog box by right-clicking **Report** in the **Design Layout** window and then selecting **Properties**. The **Design Layout** is displayed in Excel when you click **Design Layout** on the **Formatting** tab of the **Report Editor**.



Header

Start Row - Specifies the starting row for the report header. The default is \$1 for the header.

End Row - Specifies the ending row for the report header. The last row of the header is the yellow-shaded row with the column header text.

Repeat header - Tell the report to use the same header information for each sheet in the design layout.

Post-Formatting Macros

Delete - Removes the selected **Custom Macro** row.

Custom Macros

Specifies macros or add-ins to apply after a report runs. You use macros and add-ins to apply formatting to cells in a report. Three macros are delivered: Hide or Display Rows, Show Difference in Colors, and Simple Automatic Sum. The code for these macros is available in [Product]

Directory]\CatalogData\Symbols\Reports\Components for Reports\Addins\SP3DReportMacros.xla on the server.

HideOrDisplayRows - This macro is used in the context of a grouping report. The reporting software always outputs all rows of data resulting from the query. If grouping is used, the rows are organized into groups according to the design time grouping definition. **HideOrDisplayRows** is used to hide all but one row of each group giving the expected grouped appearance.

ShowDiffInColors - This macro is used in differential reports. If a row in the output shows a row deleted since the first (baseline) report ran, the row appears in red with strike-through lettering. If the row is old (was present in the baseline), the row has the same color as that of the baseline output and any values modified since the baseline are *italicized*. New rows are untouched by this macro.

Note

• The software determines differences in rows (old, new, deleted) by the set of properties named at report design time. If any of these *identity* properties changes, a row is considered new or deleted compared to the baseline report.

SimpleAutoSum - This macro supports the summation of a column of visible numbers. The **SimpleAutoSum** macro provides a summing function that operates only on the visible data, in effect allowing a sum of the sums. It allows you to define a sum function on an output cell and a separate *total* cell elsewhere on the report to which the macro displays the sum of grouped data.

Note

• The reporting software has quantity functionality that adds the values of a particular column of a group. Quantity values output with each row of the group, whether the row is hidden or not. If a typical Microsoft Excel formula is applied, all rows are included in the sum.

Filling Mode

Fill Values - Specifies that the software completes the report data based on the exact number of rows with attributes and blank rows specified in the report template. It is possible that some of the report data for a record could overwrite the next record.

Insert Values, Copy Formats and Formulas - Specifies that the software completes the report data by inserting rows as needed between different attributes.

Note

• By default, all items are associated to the same sheet.

Related Topics

- Edit a Report Template, page 295
- Properties Command (Shortcut Menu in Microsoft Excel), page 280
- Understanding Report Templates: An Overview, page 254

Item Properties Dialog Box

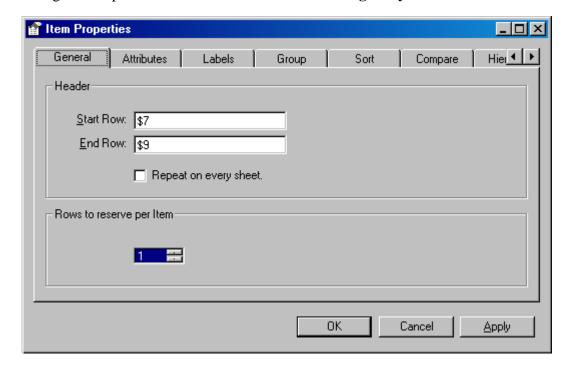
Sets formatting options for the selected item in the **Design Layout** window. You can open this dialog box by right-clicking single item in the **Design Layout** window and then selecting **Properties**. The **Design Layout** is displayed in Excel when you click **Design Layout** on the **Formatting** tab of the **Report Editor**.

Related Topics

- Edit a Report Template, page 295
- Properties Command (Shortcut Menu in Microsoft Excel), page 280
- Understanding Report Templates: An Overview, page 254

General Tab (Item Properties Dialog Box)

Sets general options for the selected item in the **Design Layout** window in Excel.



Header

Start Row - Specifies the starting row for the report header for this particular report item.

End Row - Specifies the ending row for the report header for this particular report item.

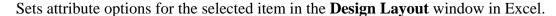
Repeat on every sheet - Tells the report to use the same header information for each additional sheet in the design layout.

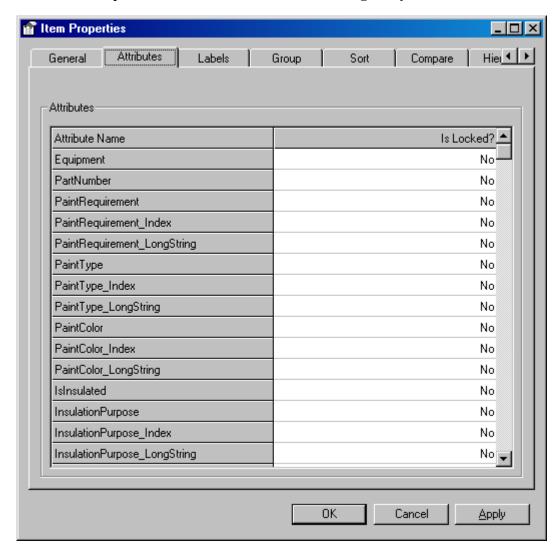
Rows to reserve per item - Specifies the number of rows for the item. For example, you could have one attribute in Row 10, another attribute in Row 11, and another attribute in Row 12. The row spread in this case would be three rows.

Related Topics

• Item Properties Dialog Box, page 283

Attributes Tab (Item Properties Dialog Box)





Attribute Name - Lists the attribute names in the SELECT statement for this particular report item. You can add more attribute names by editing the queries in the Report Editor.

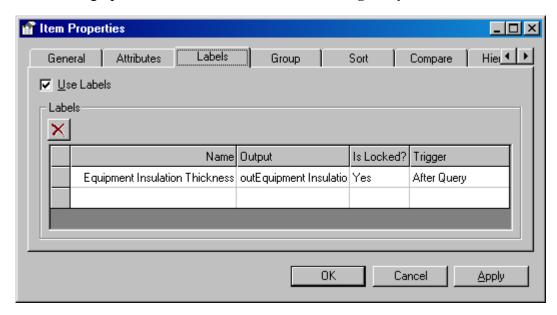
Is Locked? - Specifies whether the attribute is locked or not. You can lock cells to prevent changes. For more information about locking cells and protecting data, see the Microsoft Excel Help.

Related Topics

• Item Properties Dialog Box, page 283

Labels Tab (Item Properties Dialog Box)

Sets labeling options for the selected item in the **Design Layout** window in Excel.



Use Labels - Enables the controls on this tab.

Delete - Removes the selected Label row.

Name - Specifies a label to place on the report. When you click **More** in the dropdown list, the **Select Label** dialog box appears, allowing you to select labels from the catalog. When you click **New Format** in the dropdown list, the **Label Editor** appears, allowing you to specify a new format for a label.

Note

• Any labels you add using the **New Format** functionality are only available for the current report. They are not available for use in other reports.

Output - Identifies the output of the label on the report. In cases where a returned property from a query has the same name as the label name, you must give a different name for the label.

Is Locked? - Specifies whether the label is locked or not. You can lock cells to prevent changes. For more information about locking cells and protecting data, see the Microsoft Excel Help.

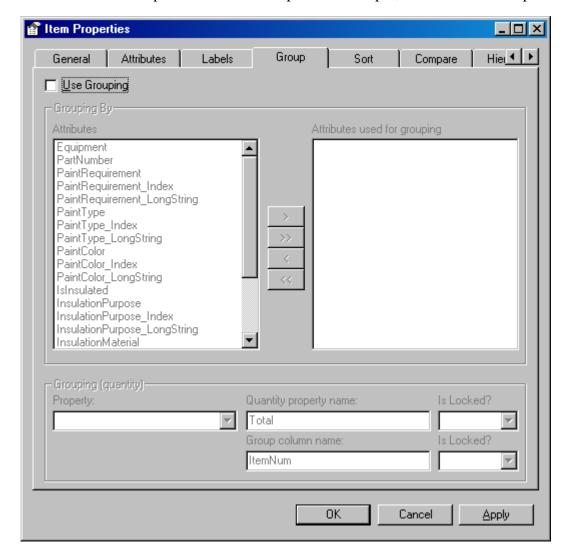
Trigger - Specifies when the software populates the label and places it on the report. The values for the label trigger are After Query, After Grouping, and After Comparison. For example, if you have a material takeoff (MTO) report, you might want the sum of weights for a group to appear on the label. In this case, you will want the label trigger after the grouping of the MTO sections.

Related Topics

• Item Properties Dialog Box, page 283

Group Tab (Item Properties Dialog Box)

Sets grouping options for the selected item in the left pane of the Excel window. The grouping options define the properties that group the input data into composite rows when one row of output is based on multiple rows of input, such as in MTO reports.



Use Grouping- Enables the controls on this tab.

Grouping By

Columns - Lists the columns available for grouping.

Columns used for grouping - Lists the columns you want to use for grouping. You can move items back and forth between the two main boxes on this dialog box by using the arrow keys in the middle of the dialog box.

Grouping (quantity)

Property - Specifies a property to group. The available properties are the same columns available for grouping.

Quantity property name - Specifies a name for the property to group. For example, you can sum the quantity of items to calculate a total number of items for each group.

Is Locked? - Specifies whether the cell is locked or not. You can lock cells to prevent changes. For more information about locking cells and protecting data, see the Microsoft Excel Help.

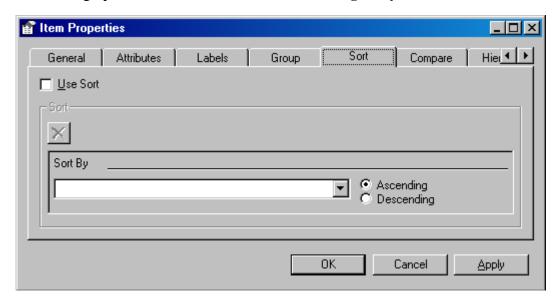
Group column name - Specifies the group column name. For example, the column name might be **ItemNum** to designate that the column contains the total number of items in each group.

Related Topics

• Item Properties Dialog Box, page 283

Sort Tab (Item Properties Dialog Box)

Sets sorting options for the selected item in the **Design Layout** window in Excel.



Use Sort - Enables the controls on this tab.

Delete - Removes the last sorting order from the tab.

Sort By/Then By - Specifies an item by which to sort the report. Once the first sort order is selected, a **Then By** section is added, allowing multiple sorting specifications.

Ascending - Sorts in alphanumeric ascending order.

Descending - Sorts in alphanumeric descending order.

Note

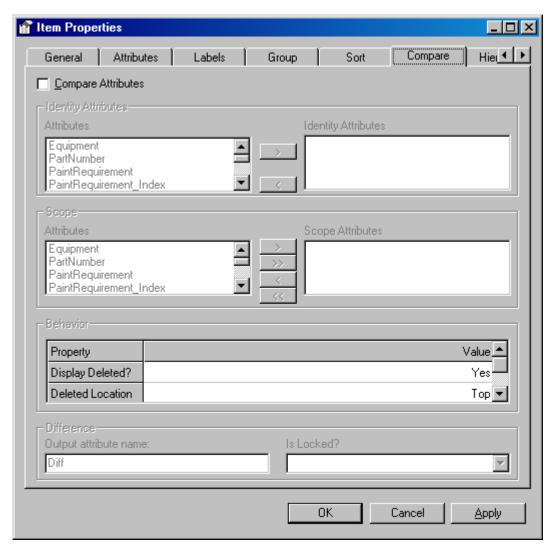
• All attributes specified for sorting must be used for grouping. If you have attributes involved in sorting on the **Sort** tab that do not match the attributes in the **Attributes used for grouping** field of the **Group** tab, an error message appears.

Related Topics

• Item Properties Dialog Box, page 283

Compare Tab (Item Properties Dialog Box)

Sets comparison options for the selected item in the **Design Layout** window in Excel. These options apply to differential reports.



Compare Attributes - Enables the controls on this tab.

Identity Attributes

Attributes - Lists properties used as identifiers of rows. The software uses identifiers to check to see if rows in the baseline report and the current report match. If the rows have different identifiers, then the rows were deleted or added. If the rows have the same identifier, then the row was modified.

You can move properties back and forth between the boxes by using the arrow keys in the middle.

Identity Attributes - Lists properties used as identifiers of rows.

Scope

Attributes - Lists properties compared between the two reports to determine if differences exist between matched rows. You can move properties back and forth between the boxes by using the arrow keys in the middle.

Scope Attributes - Lists properties compared between the two reports.

Behavior

Display Deleted? - Displays deleted rows on the report. Deleted rows are in the baseline report but not in the current report.

Deleted Location? - Specifies the location on the report where the deleted rows appear. The location can be the top of the report, the bottom of the report, or in-line (between other rows).

Display Modified? - Displays modified rows on the report. Modified rows can either be old or new. Old rows are from the baseline report. New rows are from the current report.

Added Location? - Specifies the location on the report where the added rows appear. The location can be the top of the report, the bottom of the report, or in-line (between other rows).

Difference

Output attribute name - Specifies the header text for the attribute that shows the difference flags. The following table shows the flags and their meanings.

Flag	Meaning
A	The row is not in the baseline report but is in the current report.
D	The row is in the baseline report but not in the current report.
О	One or more of the compared attributes in this row are different between the baseline and the current reports. The row is from the baseline report.
N	One or more of the compared attributes in this row are different between the baseline and the current reports. The row is from the current report.
U	The row is the same in the baseline and current reports.

Is Locked? - Specifies whether the attributes are locked or not. You can lock cells to prevent changes. For more information about locking cells and protecting data, see the Microsoft Excel Help.



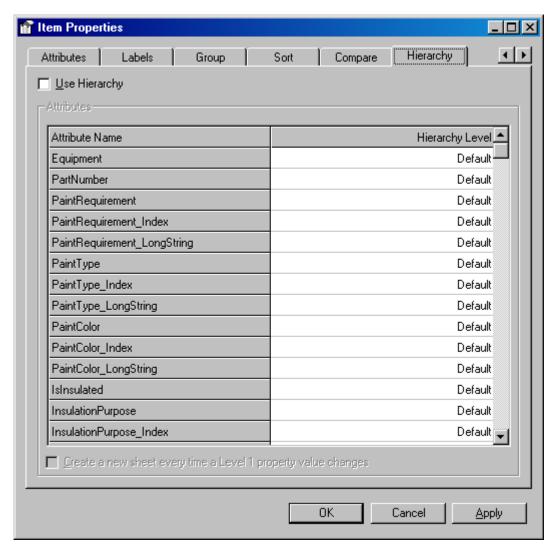
 All attributes specified as comparison identifiers must be used for grouping. If you have differences between the attributes chosen in the Attributes used for grouping field of the Group tab and the attributes chosen in the Identity fields on the Compare tab, an error message appears.

Related Topics

• Item Properties Dialog Box, page 283

Hierarchy Tab (Item Properties Dialog Box)

Sets hierarchy options for the selected item in the **Design Layout** window in Excel.



Note

The list of attributes on this tab is controlled by the values set in the
 Attributes used for grouping grid on the Group tab of the Item
 Properties dialog box. If the Group tab is disabled, the Hierarchy tab is disabled also. You can enable both tabs by checking the Use Grouping box on the Group tab.

Use Hierarchy - Enables the controls on this tab.

Attribute Name - Lists all of the report attributes from the **Group** tab.

Hierarchy Level - Specifies the hierarchy level for the attribute. The hierarchy levels are all set initially to "Default".

Create a new sheet every time a Level 1 property value changes. - Specifies that a new sheet is created in Excel whenever a Level 1 value is changed in the hierarchy.

Related Topics

• Item Properties Dialog Box, page 283

Expand All Command (Shortcut Menu in Microsoft Excel)

Opens all branches of the **Design Layout** in Microsoft Excel. The **Design Layout** is displayed in Excel when you click **Design Layout** on the **Formatting** tab of the **Report Editor**.

- Designing Report Layout: An Overview, page 278
- Edit a Report Template, page 295

Collapse All Command (Shortcut Menu in Microsoft Excel)

Closes all branches of the **Design Layout** in Microsoft Excel. The **Design Layout** is displayed in Excel when you click **Design Layout** on the **Formatting** tab of the **Report Editor**.

Related Topics

- Designing Report Layout: An Overview, page 278
- Edit a Report Template, page 295

Edit a Report Template

- 1. Right-click a report in the **Detail View**, and click **Edit Template**.
- 2. To modify SQL queries, modify the SQL statements on the appropriate **Query** tabs to query the databases. Click **Show Results Pane** to open a lower window that displays the results of the query. Test the query by clicking **Execute Query**



- You can also type a different description for the query in the **Description** box at the top of the tab.
- 3. To modify property-based queries, specify a different filter, or create a new filter in the **Filter** box on the appropriate **Query** tabs.

💡 Tips

- You can add or remove properties in the query using the Add and Remove buttons.
- Filters created with the **Create New Filter** command must be moved to the Catalog before running the report. Select **More** in the dropdown to create filters directly in the database.
- 4. On the **Formatting** tab, specify the layout of the report by clicking **Design Layout** . The software opens a window in Excel.

💡 Tip

- If you have already opened the Excel layout for the template, the software asks if you want to overwrite it in the temporary location.
- 5. Select an attribute on the left, click **Insert**, then click in the Excel window to place the attribute. You can also drag and drop the attribute from the left pane to the Excel sheet.

- 6. Right-click items in the left pane, and click **Properties** to specify layout properties, such as grouping, sorting, and comparing.
- 7. Exit Excel, and save the workbook when prompted.
- 8. To modify the formatting parameters, click **Edit Formatting Parameters** on the **Formatting** tab.

♀ Tip

- You can also open the **Formatting Parameters** dialog box from the left pane in Excel by right-clicking the **Formatting Parameters** item and selecting **Properties**.
- 9. On the **Formatting Parameters Designer** dialog box, set attributes for each field on the **Fields** tab.
- 10. Specify information for the units of measure on the **Units of Measure** tab.
- 11. Specify information for the coordinate systems on the **Coordinate Systems** tab.

→ Tip

- In the context of reporting, coordinate systems are also known as matrices. You can see this term in some of the templates if you open them in Notepad.
- 12. Save the changes to the template by clicking **File > Save Report Template**. The software saves the edited template in the Model database.
- 13. If you added or modified parameters for any query, right-click the report in the **Detail View**, and then click **Parameters** to redefine parameters before you run the report.

Notes

- The software combines the options you set on all the tabs to form the
 report template. You can add tabs by clicking the commands on the **Tools**menu. Each report template can have multiple queries and baselines, but
 only one format and one display.
- You can save the template in another location by clicking File > Save
 Report Template As. By default, templates saved using this command are personal reports that you can run in most of the tasks in the software using the My Reports tab of the Run Report dialog box.

• You can make new report templates available to other users by making changes to the names and locations of report templates on the **Report** sheet of the **Reports.xls** spreadsheet and then bulk loading the reference data into the Catalog database. All of the report files (.*rtp*, .*rqe*, etc.) must be saved to a directory to which the users have access (like the Symbols share).

- Create a New Report Template from an Existing Template, page 263
- Spreadsheet Reports: An Overview, page 251
- Understanding Report Templates: An Overview, page 254
- Using Queries to Extract Data for Reports: An Overview, page 258

Parameters Command (Report Shortcut Menu)

Specifies the parameters for a report, such as units of measure and coordinate systems. You can access this command by right-clicking a report in the **Detail View**. The parameters required by the report are defined in the report template. For more information on the parameter definition, see *Edit Template Command (Report Shortcut Menu)*, page 268.

Related Topics

• Understanding Report Templates: An Overview, page 254

Report Parameters Dialog Box

Specifies parameters for a report deliverable, such as units of measure and coordinate systems.

This dialog box only appears when you run a report that requires user input for the parameters. In addition, the controls on this dialog box may vary, depending on the report definition.

Unit of Measure - Sets the units of measure for the report.

Matrix - Specifies information about the coordinate system.

Notes

- For some reports, several dialog boxes requiring report parameters appear. The dialog boxes take the form of a wizard with **Back**, **Next**, and **Finish** buttons at the bottom.
- The query you set up for running a report can generate dialog boxes that prompt for certain report parameters. In this way, your query can customize the report creation.

Copy Report To Catalog Command (Report Shortcut Menu)

Copies a modified report template to the catalog so it can be used by other users.

The **Copy To Catalog** command is available when you right-click a report document.

Notes

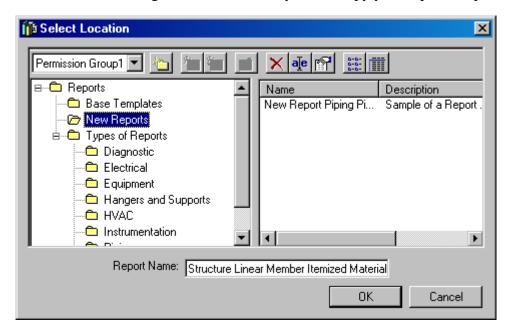
- The **Copy To Catalog** command is not available when multiple report documents are selected. You can only copy one report template to the catalog at a time. If the report template has not been modified from its original state, the command is not available.
- The report does not have to be up-to-date because only the report template is copied. All drawing views are copied as part of the template, but the drawing view content is not included.

Related Topics

Copy a Report Template to the Catalog, page 301

Select Location Dialog Box

Lists available catalog locations to which you can copy your report template.



Note

- If you select the **New** folder in the hierarchy, the **Delete** and **Rename** buttons are enabled. When you select an existing catalog folder, most of the buttons on the dialog box toolbar are disabled.
- New Creates a new folder named New Folder under the selected item in the hierarchy. You can rename the folder or keep the name.
- ➤ **Delete** Deletes the selected report template, if you have the correct permissions. A warning message appears. You cannot undo a delete action once it is accepted.
- **Rename** Renames the selected report template, if you have the correct permissions.
- **Properties** Displays the properties for the selected report template.
- List View Displays the catalog report hierarchy on the left and the report documents on the right.
- **Detail View** Displays the catalog report hierarchy on the left side and a detailed grid view of the report documents on the left.

Permission Group - Shows the catalog permissions for the selected folder or report document.

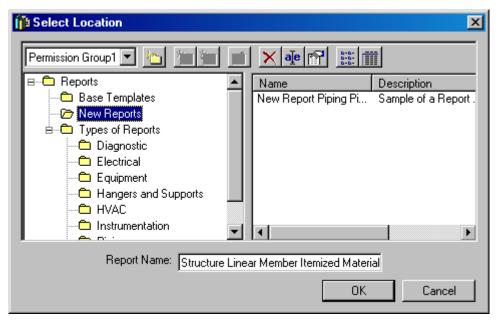
Name - Shows the name of the report being copied to the catalog.

- Copy a Report Template to the Catalog, page 301
- Copy Report To Catalog Command (Report Shortcut Menu), page 299

Copy a Report Template to the Catalog

The **Copy Report To Catalog** command is available on the shortcut menu of a report document with a modified report template.

1. Right-click a report that has a modified template you want to copy to the catalog, and select **Copy Report To Catalog**. The **Select Location** dialog box displays, showing the modified report template in the **Report Name** field.



- 2. Select an existing folder location, or create a new folder by clicking **New** . If you create a new folder, you can change the name of the folder or leave it with the default name. For example, in the graphic above, a new folder was created and renamed **New Reports**.
- 3. Click **OK** to copy the report to the selected folder in the catalog. The report template is now stored in the catalog for all users to use.

Notes

- You can copy only one report template at a time. This command is not available if multiple documents are selected or if the report template has not been modified.
- To delete a report template, select it and click **Delete** X. You are only able to delete a report template if you have appropriate catalog permissions.
- To rename a report template, select it and click **Rename** de. You are only able to rename a report template if you have appropriate catalog permissions.

Related Topics

• Copy Report To Catalog Command (Report Shortcut Menu), page 299

Save Report Template Command (File Menu)

Saves the modified report template and its components to the Catalog. If you want to be able to edit your report in the Detail View, save the report to the Catalog before exiting the Report Editor.

If you want your report templates to be available to other users, you must make changes to the names and locations of report templates on the **Report** sheet of the **Reports.xls** spreadsheet and then bulk load the reference data into the Catalog database.

- Create and Update a Delivered Report, page 266
- Save Report Template As Command (File Menu), page 303

Save Report Template As Command (File Menu)

Saves the selected report template and components to a location that you specify. You can also change the names of report templates and their components before saving. This command is available only on the **File** menu when you are editing a report template in the **Report Editor**.

This command also allows you to change the names of templates and their components before saving.

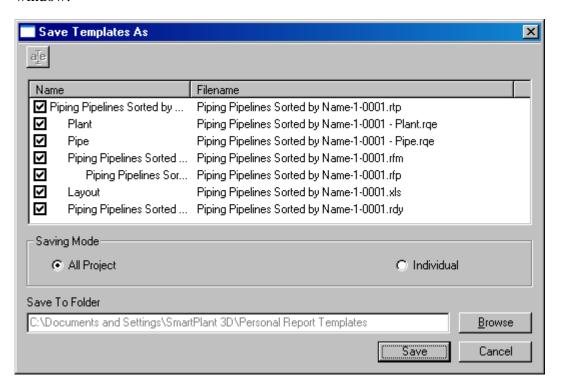
By default, templates saved using this command are personal reports. You can run these reports in most of the tasks in the software using the **Run Report** command on the **Tools** menu. Personal reports are located on the **My Reports** tab of the **Run Report** dialog box. The file location, defined with the **Tools** > **Options** command, must be set to the directory that contains the .rtp file of your report.

You can make new report templates available to other users by making changes to the names and locations of report templates on the **Report** sheet of the **Reports.xls** spreadsheet and then bulk loading the reference data into the Catalog database. Do not overwrite the delivered report templates.

- Create and Update a Delivered Report, page 266
- Save a Report Template to a Specified Location, page 305
- Save Report Template Command (File Menu), page 302

Save Template As Dialog Box

Sets options for saving the components of a report template or label. You can open this dialog box by clicking **File > Save Report Template As** in the **Report Editor** window.



Rename - Renames the components of a report template or label. To modify a component, select it in the table and then click **Rename**. When you change the name of a component, the file name changes automatically.

Name - Displays the name of the report template or label component. You can save parts of the report template or label by clicking the checkboxes to the left of the names.

Filename - Displays the file name of the component.

All Project - Saves all components of the report template or label in a folder structure similar to drawing labels and report templates. You choose a root folder, and the templates are all saved in the root folder.

Individual - Saves each component separately. You can specify a different location for each component.

Save To Folder - Displays the name of the folder to save the report template or label. To change the location, click **Browse**. By default, the files are saved to the personal reports location defined on the **File Locations** tab of the **Options** dialog box in the

Common task. You can change the default file locations. For more information, see the *Common User's Guide*.

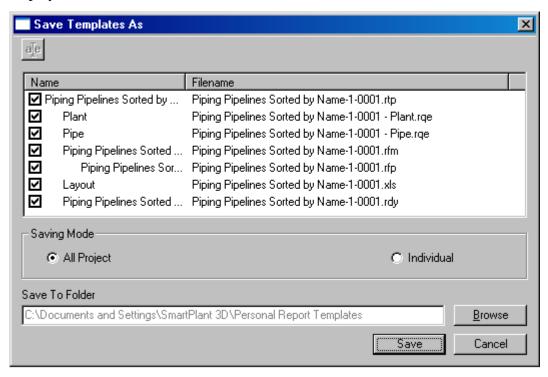
Related Topics

- Create and Update a Delivered Report, page 266
- Save Report Template As Command (File Menu), page 303

Save a Report Template to a Specified Location

The following procedure saves the report template to a location that you specify using the **File > Save Report Template As** command. To save a modified report template and its components to the Catalog, use the **File > Save Report Template As** command. For more information, see *Save Report Template Command (File Menu)*, page 302.

- 1. Right-click a report in the **Detail View** and select **Edit Template**. Modify the report template as needed.
- 2. Click **File > Save Report Template As**. The **Save Template As** dialog box displays.



- 3. To rename a component of the report, select the component in the table, and then click **Rename** After you type the new name of the component, the file name changes automatically.
- 4. To save parts of the report template, click the check boxes to the left of the names to clear the components you do not want to save.

- 5. Specify the saving mode and location of the report template and components, and click **Save**. By default, the files are saved to the personal reports location defined on the **File Locations** tab of the **Options** dialog box in Common. For more information, refer to the *Common User's Guide*.
- 6. Click **Save** to save the report template to the specified location.

Notes

- If you added or modified parameters for any query, right-click the report in the **Detail View**, and then click **Parameters** to redefine parameters.
- If you want your report templates to be available to other users, you must
 make changes to the names and locations of report templates on the
 Report sheet of the Reports.xls spreadsheet, and then bulk load the
 reference data into the Catalog database.
- Do not overwrite delivered templates. Save new or modified template files with unique names different from the templates delivered with the software.

- Edit a Report Template, page 295
- Spreadsheet Reports: An Overview, page 251
- Understanding Report Templates: An Overview, page 254
- Using Queries to Extract Data for Reports: An Overview, page 258

Add Query Command (Tools Menu)

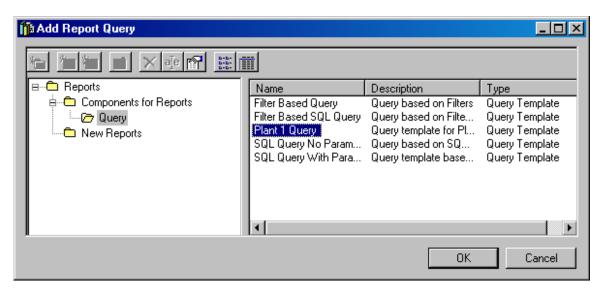
Adds an existing query to the selected report template. Only those .rqe files that are included in the bulk loaded catalog data are available for selection.

Related Topics

- Add a Filter-Based Query to a Report Template, page 308
- Create a New Report Template from an Existing Template, page 263
- Edit a Report Template, page 295
- Report Editor, page 268
- Understanding the Report Templates Folder: An Overview, page 256
- Using Queries to Extract Data for Reports: An Overview, page 258

Add Report Query Dialog Box

Specifies a query template to add to the report. This dialog box appears when you click **Tools > Add Query** from the **Report Editor**. By browsing through the hierarchy, you can find any query template in the Catalog database. This dialog box is resizable.



- **Properties** Displays the properties of the selected item. All properties on the **Properties** dialog box are read-only.
- List View Sets the dialog box to display items in a list view.
- Grid View Sets the dialog box to display items in a spreadsheet-style grid view.

Note

The grayed out tools on this dialog box are not available with the Tools > Add Query command.

Related Topics

- Add a Filter-Based Query to a Report Template, page 308
- Add Display Command (Tools Menu), page 312
- Add Formatting Command (Tools Menu), page 311
- Add Query Command (Tools Menu), page 307
- Remove Report Component Command (Tools Menu), page 315
- Understanding the Report Templates Folder: An Overview, page 256
- Using Queries to Extract Data for Reports: An Overview, page 258

Add a Filter-Based Query to a Report Template

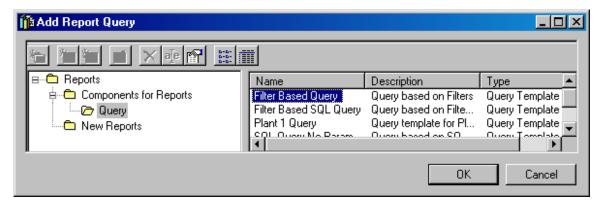
1. In the **Management Console**, verify that at least one spreadsheet report component exists. If none exists, add a component for reports.

Add a Component, page 36

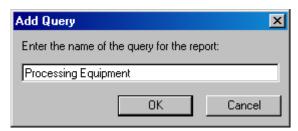
- 2. Right-click the spreadsheet report component, and click **Create Report** on the shortcut menu. For more information, see *Create Report Command (Report Shortcut Menu)*, page 262.
- 3. On the **Select Report Template** dialog box, click a report template, and click **OK**. The software creates the report document in the **Detail View**.

💡 Tip

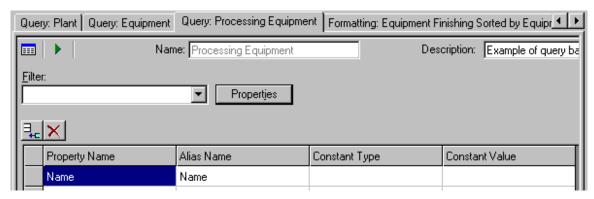
- The software prompts for parameters or a filter if the selected report template requires these inputs.
- 4. Right-click the report, and click **Update Document(s)**. The software generates the report. For more information, see *Updating Documents: An Overview*, page 337.
- 5. Select **Tools > Add Query**. The **Add Report Query** dialog box displays.
- 6. Select a filter-based query to add to the product and click **OK**. For example, you could add a basic **Filter Based Query** to the report template.



7. Name the query on the **Add Query** dialog box.



8. When you click **OK**, the software adds the new query to the Report Editor.



- 9. Select a filter to use with the query. Select **More** to display the **Select Filter** dialog box and specify an existing filter. You can also create a new filter by selecting **Create new filter** in the dropdown.
- 10. Add properties to the query as needed using the table provided. Click **Add** ∃ to display the **Select Properties** dialog box and define the property. When you click **OK** the property is added as a new row to the **Query** tab. For example, you could add a property for the **Insulation Requirement** on certain equipment.
- 11. To remove any properties you do not need in the query definition, select a property row and click **Delete** ×.

12. To test run the query, click **Execute Query** ▶. The **Query** tab updates to show the results in a panel at the bottom of the tab. You can turn the results panel on and off by clicking **Show Results Panel** ■. The graphic below shows an example query results pane for a query that uses a user-defined filter called **ProcessingEquipmentFilter** and the **Insulation Requirement** property.



13. Save your changes to the report template using either **File > Save Report Template** or **File > Save Report Template** As. For more information, see *Save Report Template Command (File Menu)*, page 302 and *Save Report Template As Command (File Menu)*, page 303.

- Select Properties Dialog Box, page 270
- Spreadsheet Reports: An Overview, page 251
- Understanding Report Templates: An Overview, page 254

Add Formatting Command (Tools Menu)

Adds a **Formatting** tab to the selected report template. Each report template can contain only one Formatting tab. If a Formatting tab is already defined for the selected report template, this command is not available.

- Create a New Report Template from an Existing Template, page 263
- Edit a Report Template, page 295
- Report Editor, page 268
- Understanding the Report Templates Folder: An Overview, page 256

Add Display Command (Tools Menu)

Adds a **Display** tab to the selected report template. Each report template can contain only one Display tab.

This command is not currently available.

- Create a New Report Template from an Existing Template, page 263
- Edit a Report Template, page 295
- Report Editor, page 268
- Understanding the Report Templates Folder: An Overview, page 256

Add Baseline Command (Tools Menu)

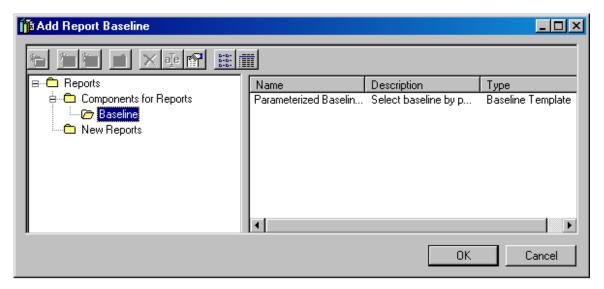
Adds a **Baseline** tab to the selected report template. Each report template can contain only one Baseline tab. If a Baseline tab is already defined for the selected report template, this command is unavailable to you.

Related Topics

- Create a New Report Template from an Existing Template, page 263
- Edit a Report Template, page 295
- Report Editor, page 268
- Understanding the Report Templates Folder: An Overview, page 256

Add Report Baseline Dialog Box

Specifies a baseline template to add to the report. This dialog box appears when you select **Tools > Add Baseline** from the **Report Editor**. Each template can contain only one baseline. By browsing through the hierarchy, you can find any baseline template in the Catalog database.



- **Properties** Displays the properties for the selected report template.
- **List View** Displays the catalog report hierarchy on the left and the report documents on the right.
- **Detail View** Displays the catalog report hierarchy on the left side and a detailed grid view of the report documents on the left.

Note

• The grayed out tools on this dialog box are not available with the **Tools** > **Add Baseline** command.

- Add Baseline Command (Tools Menu), page 313
- Understanding the Report Templates Folder: An Overview, page 256

Remove Report Component Command (Tools Menu)

Removes the selected report component from the report template. To select a component to remove, click the tab for that component in the **Report Editor**. The name of this command changes to reflect the name of the component tab you select. For example, if you select a tab called **Query: Plant**, the command name changes to **Remove Query: Plant**. For example, if you wanted to delete a tab called **Query: Processing Equipment**, click the tab, then select **Tools > Remove Query: Processing Equipment**. The software displays a confirmation message before removing the query definition and tab from the report template.

You cannot use this command to remove display, formatting, or baseline components from a report.

- Create a New Report Template from an Existing Template, page 263
- Edit a Report Template, page 295
- Report Editor, page 268
- Understanding the Report Templates Folder: An Overview, page 256

Spreadsheet Reports: An Overview			

3D Model Data: An Overview

You can create a 3D Model Data component in the Drawings and Reports task. Once created, you set up the component to be based on a specific filter. The filter defines the contents of the component documents when they are created.

You can set the output purpose on the component to save to a SmartPlant Review file or publish to The Engineering Framework (TEF). The publish capability is only available if you are registered with TEF. For more information, see the *SmartPlant 3D Installation Guide*, available from the **Help > Printable Guides** command.

The basic workflow for creating a 3D Model Data component:

- Define your workspace with all objects you will be including in the 3D Model Data documents you are publishing or saving as SmartPlant Review files
- Create a 3D Model Data component and set it up with a filter and purpose
- Create the 3D Model Data documents
- Update the documents
- Save the documents to a predefined location for viewing in SmartPlant Review or publish the documents to a SmartPlant Review file

Project Supervisor Setup

Your project supervisor should set up appropriate filters that define the objects to include in the component documents when they are created.

You must install the **Intergraph Schema Component** and the **Framework Client** to use this component. For more information, see the *Installation Guide* available from the **Help > Printable Guides** command.

- 3D Model Data Component Common Tasks, page 318
- Publishing Documents: An Overview, page 373
- Setup a 3D Model Data Component, page 320

3D Model Data Component Common Tasks

The following tasks are used to setup a 3D Model Data component and configure it for viewing in SmartPlant Review or publishing to The Engineering Framework.

Define Your Workspace

Before creating a 3D Model Data component to publish or to save as a SmartPlant Review file, make sure the current workspace includes all the objects you require. For more information on defining the workspace, see the *Common User's Guide* available from **Help > Printable Guides**.

Setup a 3D Model Data Component

You can create and setup a 3D Model Data component. For more information, see *Setup a 3D Model Data Component*, page 320.

Create and Update the 3D Model Data Documents

To create the 3D Model Data documents, right-click the component and select the **Create Documents** command. When this command is complete, the documents are listed in the **Detail View**. After creating the 3D Model Data documents, you need to update them. For more information, see *Updating Documents: An Overview*, page 337.

Set Document Properties

You can specify the properties for the 3D Model Data component and documents by right-clicking and selecting **Properties**. For more information, see *Edit Document Properties*, page 334.

Save Data as a SmartPlant Review File

If you are not configured with The Engineering Framework (TEF), you can output your 3D Model Data to a SmartPlant Review (.vue) file. For more information, see *Save 3D Model Data for SmartPlant Review*, page 365.

Publish Data to The Engineering Framework

If you are configured with TEF, you can publish your 3D Model data to SmartPlant Foundation for retrieval in other tools. For more information, see *Publishing Documents: An Overview*, page 373. For more information on retrieving, see the *Common User's Guide* located in **Help > Printable Guides**.

Setup Command (3D Model Data Component)

Sets options for a 3D Model Data component. This command is available on the right-click menu on a 3D Model Data component.

Notes

- You need to define the workspace or verify that the workspace contains all the objects required by the 3D Model Data filter subset.
- You can publish .zvf files to SmartPlant Foundation and launch SmartPlant Review through SmartPlant Foundation to view the .zvf files. You can also use **File > View and Markup** to use SmartPlant Markup to navigate the model. You do not have to use the **Save as SmartPlant Review** command to use the SmartPlant Review features.
- After you create a component, you must update the documents to extract the data from the database and create the .vue, .zvf, and .xml files.

Related Topics

- 3D Model Data Component Common Tasks, page 318
- 3D Model Data: An Overview, page 317
- Setup a 3D Model Data Component, page 320

Setup Dialog Box (3D Model Data Component)

Sets options for the 3D Model Data component.

Filter - Identifies the filter to use to define the objects to include in the component definition. The filter needs to be specific to the objects that you want to publish to The Engineering Framework or to save to a SmartPlant Review file.



 You need to define the workspace to contain all the objects required by the 3D Model Data filter subset.

Purpose - Provides options for publishing the 3D Model Data.

- Save as SmartPlant Review (.vue file) Specifies that the file be saved as a .vue file for use in SmartPlant Review.
- **Publish to SmartPlant Foundation (.zvf file)** Specifies that the file be saved as a .zvf file for publishing to The Engineering Framework and SmartPlant Foundation. This option is available if you are configured to use The Engineering Framework. For more information about TEF configuration, see the section titled *The Engineering Framework* in the *SmartPlant 3D Installation Guide*, available from the **Help > Printable Guides** command.

Note

• You can publish .zvf files to SmartPlant Foundation and launch SmartPlant Review through SmartPlant Foundation to view the .zvf files. You do not have to use the **Save as SmartPlant Review** command to use the SmartPlant Review features.

Related Topics

- Setup a 3D Model Data Component, page 320
- Setup Command (3D Model Data Component), page 319

Setup a 3D Model Data Component

The 3D Model Data component is used with the **Save as SmartPlant Review** and **Publish** commands. Before creating a 3D Model Data component, you should create filters that specify the objects required for your output. Also, you must define the workspace to contain all the objects required by the 3D Model Data filter subset.

1. Right-click a folder in the **Management Console** or the **Detail View**.

♀ Tip

- To add a folder, right-click the root or another folder.
- 2. On the shortcut menu, select **New**.
- 3. Select the **3D Model Data** component on the **Add Component** dialog box, and click **OK**.
- 4. Right-click the new 3D Model Data component, then select **Setup**.
- 5. Specify a filter to identify the model data to include when the data is saved or published. Select **More** to display the **Select Filter** dialog box. Click **Properties** to display the current filter properties.
- 6. Select an output purpose for the 3D Model Data. You can save the file to a SmartPlant Review (.vue file) or publish to The Engineering Framework (.zvf file). The option to publish to The Engineering Framework (TEF) is not available if you are not registered to work with TEF.
- 7. Click **OK** to save the component settings.

Note

• After you create a component, you must update the documents to extract the data from the database and create the .vue, .zvf, and .xml files.

- 3D Model Data Component Common Tasks, page 318
- 3D Model Data: An Overview, page 317
- Publish Command, page 378
- Publishing Documents: An Overview, page 373
- Setup Command (3D Model Data Component), page 319

Setting Properties: An Overview

The software updates properties from parent nodes to child nodes and drawings in the **Management Console**.

For example, you can display the **Properties** dialog box for a folder named **Isometric Drawings**. If you set the **Division Location** property to **Huntsville**, **Alabama**, the software pushes this value to the items contained within the **Isometric Drawings** folder.

You can specify inheritance for each item on its **Properties** dialog box. If you set the override flag for a property, the property is not inherited from the parent. You can provide a new, overriding value for the property. This new value then propagates to other items deeper in the hierarchy.

Properties and Publishing

Several document properties impact the publish of the document to The Engineering Framework. Before you can publish documents in the software, you must configure your computer for TEF. The configuration includes installing SmartPlant Foundation Client and Intergraph Schema Component and registering the model with TEF. For more information about TEF configuration, see the *SmartPlant 3D Installation Guide*, available from **Help > Printable Guides**.

Even if you are registered with TEF, you must set certain properties to enable the publishing capability. Properties that control publishing are found on the **WBS Tab**. For more information, see *Set Properties for Publishing Documents*, page 335.



• The software considers blanks or cleared values as override flags.

- Edit Document Properties, page 334
- Properties Command, page 322

Properties Command

Views and edits properties about the selected item.

You can place drawing property labels when you edit a drawing template. For more information, see *Place a Drawing Property Label on a Template*, page 124.

The properties of child items are inherited from the parent item unless you set the **Override** column for the properties.

Related Topics

- Edit Document Properties, page 334
- Setting Properties: An Overview, page 321

Properties Dialog Box

Sets options for items in the **Management Console**. All items in the **Management Console** have a **Properties** command on their right-click shortcut menus. Using the **Properties** dialog box, you can control how you want properties to propagate through the hierarchy.

You can specify labels for some of the properties on the tabs. Click the browse button at the right of the table cell to display the **Choose Label** dialog box.

Notes

- The **WBS** tab is only available if you are registered with The Engineering Framework (TEF).
- If you are not registered with TEF, the **Issue** tab is always available. If you are registered with TEF, the **Issue** tab is only available if you have issued documents to a contract and the information is read-only. For more information, see *Issue Request Documents*, page 383.

Related Topics

- Edit Document Properties, page 334
- Properties Command, page 322
- Setting Properties: An Overview, page 321

General Tab (Properties Dialog Box)

Shows general properties for a drawing item.

If you access the **Properties** dialog box after selecting multiple drawings, this tab will not be available.

Name - Displays the name of the property.

Value - Sets the current value of the property.

Behavior - Specifies whether to inherit or override a property in the hierarchy of items in the **Management Console**. If the property is **Inherited**, the setting comes from items higher in the hierarchy. If the property is not read-only, you can provide a new, overriding value for the property by changing the behavior setting to **Override**. This new value then propagates to other items deeper in the hierarchy. If you set the behavior to **Force Override**, the property setting is forced to items deeper in the hierarchy, even if they are set to **Inherit**.

Properties

Location- Displays the location of the file on the server.

Published - Indicates whether the drawing has been published to The Engineering Framework.

Size - Displays the size of the file in KB (kilobytes).

Size On Disk - Displays the size of the file on disk.

Related Topics

• Properties Dialog Box, page 322

Title Area Tab (Properties Dialog Box)

Sets options for the title area of drawings.

If you access this tab after selecting multiple drawings, these fields will appear empty, regardless of what information was defined for any of the drawings individually. Any information you add to this tab will overwrite the previously defined information in corresponding rows in the selected drawings.

Name - Displays the name of the property.

Value - Displays the current value of the property.

Behavior - Specifies whether to inherit or override a property in the hierarchy of items in the **Management Console**. If the property is **Inherited**, the setting comes from items higher in the hierarchy. If the property is not read-only, you can provide a new, overriding value for the property by changing the behavior setting to **Override**. This new value then propagates to other items deeper in the hierarchy. If you set the behavior to **Force Override**, the property setting is forced to items deeper in the hierarchy, even if they are set to **Inherit**.

Properties

Border - Specifies the border attribute that stores the name of border for the title block. This attribute also stores the dimensions of the border.

Charge Number - Defines the charge number for the drawing.

Charge Title - Describes the **Charge Number** box. The charge title text is placed to the left of the charge number.

Company Name - Specifies the name of the company for which the project is designed.

Desc1 - Describes the drawing. This description is the first of four lines of text used to describe the drawing.

Desc2 - Describes the drawing. This description is the second of four lines of text used to describe the drawing.

Desc3 - Describes the drawing. This description is the third of four lines of text used to describe the drawing.

Desc4 - Describes the drawing. This description is the fourth of four lines of text used to describe the drawing.

Division Location - Specifies the location of the division for which the project is designed.

Division Name - Specifies the name of the division for which this project is designed.

Drawing Naming Rule - Specifies a default or custom naming rule to the drawing. The default name rules provided include:

- **Default Drawing Name Rule** This is the default rule for Snapshot and Volume drawing types. It uses the drawing type name as the base for the drawing name. For example, a drawing type named "Equipment Plan" generates drawings with names "Equipment Plan-01-0001", "Equipment Plan-01-0002", "Equipment Plan-01-0003", when the location for the model is "01".
- **Default By Query Name Rule** This is the default rule for all Drawings by Query drawing types. It uses the name of the object for which the drawing is being created or the filter used to generate the drawing. For example, for object PUMP001A-01-0001, the drawing has the same name, PUMP001A-01-0001. For a drawing created from "PipingPartFilter#3", the drawing name is the same as that of the filter, "PipingPartFilter#3". The rule does not apply unique numbering because there is only one drawing for each object or filter.

• **Default Report Name Rule** - This is the default rule for all Spreadsheet Reports. It uses the name of the report as the base for the document name. For example, for report EquipLoc1, the software creates "EquipLoc1-01-0001", "EquipLoc1-01-0002", "EquipLoc1-01-0003", when the location for the model is "01".

Note

• User-defined naming rules appear in the list if you bulkload against the **CDrawingSheet** class, which is the class for the drawing object. The rules are defined on the **NamingRules** sheet in the *GenericNamingRules.xls* workbook. For more information, see the *SmartPlant 3D Reference Data Guide* available from **Help > Printable Guides**.

Drawing Number - Displays the unique identifier for the drawing.

Drawing Size - Defines a standard note value for the border size. For more information about these values, see the *PDS Drawing Manager User's Guide*.

Drawing Type - Defines the three-letter code to identify the type of drawing. For example, the type can be DGN.

Job Number - Defines the unique identifier assigned to a capital project or job.

Note Line - Specifies text for a miscellaneous note line.

Plant Name - Specifies the name of the plant or project for which the drawing is designed.

Project Version - Defines the number and letter sequence that identifies a particular generation of a document that was created since the last approved revision.

Revision Numbers - Defines the number of the current revision for this drawing.

Sheet - Defines the number of the page and the total number of pages that are associated with this one. For example, the value might be **3 of 5**.

Site Location - Specifies the site location for which the drawing is designed.

Site Name - Specifies the name of the site where the plant is being constructed.

Title1 - Specifies text for the first miscellaneous title. This title is usually a description of the area shown on the drawing.

Title2 - Specifies text for the second miscellaneous title. This title is usually a description of the type of drawing.

Title3 - Specifies text for the third miscellaneous title.

Related Topics

• Properties Dialog Box, page 322

Signature Area Tab (Properties Dialog Box)

Sets options for the signature area of drawings.

If you access this tab after selecting multiple drawings, these fields will appear empty, regardless of the information defined for any of the drawings individually. Any information you add to this tab overrides the previously defined information in corresponding rows in the selected drawings.

Name - Displays the name of the property.

Value - Sets the current value of the property.

Behavior - Specifies whether to inherit or override a property in the hierarchy of items in the **Management Console**. If the property is **Inherited**, the setting comes from items higher in the hierarchy. If the property is not read-only, you can provide a new, overriding value for the property by changing the behavior setting to **Override**. This new value then propagates to other items deeper in the hierarchy. If you set the behavior to **Force Override**, the property setting is forced to items deeper in the hierarchy, even if they are set to **Inherit**.

Properties

Approved By - Specifies the name of the person responsible for approving the drawing.

Approved Date - Specifies the date the drawing was approved.

Checked By - Specifies the name of the person responsible for checking the drawing.

Checked Date - Specifies the date the drawing was checked.

Designed By - Specifies the name of the person who specified or designed the information on the drawing.

Designed Date - Specifies the date the drawing was designed.

Drawing Status - Defines the status code for the drawing.

Drawn By - Specifies the name of the person who drew the drawing, or created it.

Drawn Date - Specifies the date the drawing was drawn or created.

Extra Sign By1 - Specifies the name of an extra person who is signing the drawing.

Extra Sign By2 - Specifies the name of an extra person who is signing the drawing.

Extra Sign Date1 - Specifies the date the drawing was signed by the person whose name appears on this line.

Extra Sign Date2 - Specifies the date the drawing was signed by the person whose name appears on this line.

Extra Sign Title1 - Defines the title of the person whose name appears on this line.

Extra Sign Title2 - Defines the title of the person whose name appears on this line.

Mfg Rep Date - Specifies the date that the manufacturing representative initials the drawing.

Mfg Rep Name - Specifies the name of the manufacturing representative who signed the drawing.

Plant Number - Defines the plant number.

Proj Engineer Date - Specifies the date the project engineer initials the drawing.

Proj Engineer Name - Specifies the name of the person who is the project engineer for the project using the drawing.

Spec By - Specifies the name of the person who specified or designed the information on this drawing.

Spec Date - Specifies the date this drawing was specified or designed.

Related Topics

Properties Dialog Box, page 322

Style Tab (Properties Dialog Box)

Sets options for the style of drawings and reports.

If you access this tab after selecting multiple drawings, these fields will appear empty, regardless of the information defined for any of the drawings individually. Any information you add to this tab overrides the previously defined information in corresponding rows in the selected drawings.

Name - Displays the name of the property.

Value - Sets the current value of the property.

Behavior - Specifies whether to inherit or override a property in the hierarchy of items in the **Management Console**. If the property is **Inherited**, the setting comes from items higher in the hierarchy. If the property is not read-only, you can provide a new, overriding value for the property by changing the behavior setting to **Override**. This new value then propagates to other items deeper in the hierarchy. If you set the behavior to **Force Override**, the property setting is forced to items deeper in the hierarchy, even if they are set to **Inherit**.

Properties

Coordinate System - Specifies the global or an active coordinate system.

Important

If you want to output large coordinates on your drawings, define a coordinate system using large negative coordinates. For example, if you want coordinates of 400,000 ft output on drawings, define a coordinate system origin of -400,000 ft and place your model elements close to global 0. Select the new coordinate system in the Coordinate System field on the Style tab. For more information on defining coordinate systems, see the *Grids User's Guide* available from Help > Printable Guides.

Volume Naming Rule - Specifies the naming rule applied to the content of the drawing.

Baseline Date - Identifies a date in time when a *snapshot* of the drawing document was taken. It is a way of date-marking the document so you can more easily identify when objects have changed.

WBS Project - Specifies the Work Breakdown Structure (WBS) project style to be used with the drawing. This property serves as the answer to an asking filter when specified in a **View Style**. For more information, see *View Style Properties Dialog Box*, page 57.

Related Topics

• *Properties Dialog Box*, page 322

Issue Tab (Properties Dialog Box)

Sets options for internal issues.

Issue Number - Type the issue number for the drawing. When you have selected multiple drawings or a node in the **Management Console**, this field is not available to edit, as an issue number is created for each drawing, depending on its current revision history.

Const Date - Specifies the date the drawing was issued for construction.

Fab Date - Specifies the date the drawing was issued for fabrication.

Ref Date - Specifies the reference date for the drawing.

Job Spec - Identifies the job or material specification for the drawing.

Bid Date - Specifies the date the drawing was issued for bid.

Revision Number - Defines the revision number for this issue of the drawing.

Description - Describes briefly the scope of the issue. When you have selected multiple drawings or a node in the **Management Console**, this field is not available to edit, as each document displays a revision number based on revision history.

When Not Using The Engineering Framework

If you are not using the Engineering Framework and access **Properties** on a single document, the **Issue** tab is blank with editable value fields for new records. You cannot delete a field once it has been added.

If you are not using the Engineering Framework and access **Properties** on a folder, the **Issue** tab is blank with editable value fields that propagate values to the documents within the folder.

When Using The Engineering Framework

If you are using the Engineering Framework and have issued requests for the document, the **Issue** tab is read-only. The Issue information is retrieved from TEF for informational purposes only.

Notes

- You can create only one issue per instance of the **Properties** dialog box. To create another issue, close the dialog box and open it again.
- For information on issuing requests for contracts when working with The Engineering Framework, see *Issue Request Documents*, page 383.

Related Topics

Properties Dialog Box, page 322

Revision Tab (Properties Dialog Box)

Sets properties for handling revisions. The **Revision** tab is always read-write, regardless of whether or not The Engineering Framework is used (subject to user permissions).

Revision Mark - Specifies the letter of the current revision.

Revised By - Identifies the initials of the person who made the revisions.

Revision Date - Specifies the date of the revision.

Check - Identifies the initials of the person who checked the revisions.

Check Date - Specifies the date the revisions were checked.

Approved By - Identifies the initials of the person who approved the revisions.

Approval Date - Specifies the date the revisions were approved.

Description - Describes briefly of the scope of the revisions.

When Not Using The Engineering Framework

If you are not using the Engineering Framework and access **Properties** on a single document, the **Revision** tab is blank with editable value fields for new records. You cannot delete a field once it has been added.

If you are not using the Engineering Framework and access **Properties** on a folder, the **Revision** tab is blank with editable value fields that propagate values to the documents within the folder.

When Using The Engineering Framework

If you are using the Engineering Framework, you use the **Revise** command to create revision numbers. It reserves a revision number by adding it to the document Revision properties. The revision number is added in the form of a blank row on the **Revision** tab of the **Properties** dialog box.

After reserving the revision number, you right-click the document and select **Properties**. Go to the **Revision** tab and edit the **Revision** fields. For more information, see *Revising: An Overview*, page 367.



• You can create only one revision per instance of the **Properties** dialog box. To create another revision, close the dialog box and open it again.

Related Topics

Properties Dialog Box, page 322

WBS Tab (Properties Dialog Box)

Sets options for the Work Breakdown Structure (WBS) of drawings and reports. This tab is only available when you are registered with The Engineering Framework (TEF).

If you access this tab after selecting multiple drawings, these fields appear empty, regardless of the information defined for any of the drawings individually. Any information you add to this tab overrides the previously defined information in corresponding rows in the selected drawings.

Name - Displays the name of the property.

Value - Sets the current value of the property.

Behavior - Specifies whether to inherit or override a property in the hierarchy of items in the **Management Console**. If the property is **Inherited**, the setting comes from items higher in the hierarchy. If the property is not read-only, you can provide a new, overriding value for the property by changing the behavior setting to **Override**.

This new value then propagates to other items deeper in the hierarchy. If you set the behavior to **Force Override**, the property setting is forced to items deeper in the hierarchy, even if they are set to **Inherit**.

Properties

Project Name - Displays the project to which the item belongs. In TEF, a project is the scope of work approved for capital expenditure (that is, a job).

Document Type - Specifies the type of document, such as Civil Plan.

Document Style - Specifies the style of document, such as Ortho for orthographic drawing.

Discipline - Specifies the discipline for the document. If this is a 3D Model Data document, set the property to **SmartPlant Review Document**. If it is a drawing or report document, set the discipline to match the type of document.

Allow Publish - Sets the document as a publishable document.

Working with The Engineering Framework

You can only publish documents to The Engineering Framework after the appropriate properties are set on the **WBS** tab. The **WBS** tab is not available if the local machine login is not authenticated as a valid SmartPlant Foundation user. The properties that must be defined for publishing are: **Document Type**, **Document Style**, **Discipline**, and **Allow Publish**. For more information, see *Set Properties for Publishing Documents*, page 335.

Related Topics

• Properties Dialog Box, page 322

Notes Tab (Properties Dialog Box)

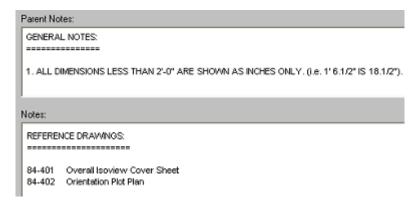
Sets notes for the item.

If you access this tab after selecting multiple drawings, these fields appear empty, regardless of the information defined for any of the drawings individually. Any information you add to this tab overrides the previously defined information in corresponding rows in the selected drawings.

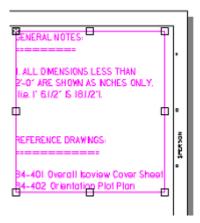
Parent Notes - Concatenates the notes from any parents of the currently selected item and displays the notes. This box is read-only.

Notes - Specifies notes for the currently selected item.

The following graphics demonstrate how parent notes and notes work. The first graphic shows how notes can be added at different levels of the hierarchy. The **Notes** tab shows Parent Notes from a higher-level folder or component. The Notes section shows additional information for a particular drawing document.



When the document is updated and displayed, the Note Region of the template contains the specified information.



Related Topics

• Properties Dialog Box, page 322

Batch Tab (Properties Dialog Box)

Sets options for the batch functionality, which allows you to update documents on another computer. This tab is available only when batch processing is configured. For more information about batch processing, see *Updating Documents: An Overview*, page 337.

If you access the **Properties** dialog box after selecting multiple drawings, this tab will not be available.

Name - Displays the name of the property.

Value - Sets the current value of the property.

Behavior - Specifies whether to inherit or override a property in the hierarchy of items in the **Management Console**. If the property is **Inherited**, the setting comes from items higher in the hierarchy. If the property is not read-only, you can provide a new, overriding value for the property by changing the behavior setting to **Override**. This new value then propagates to other items deeper in the hierarchy. If you set the behavior to **Force Override**, the property setting is forced to items deeper in the hierarchy, even if they are set to **Inherit**.

Properties

Job - Specifies the name of the batch job.

State - Defines the status of the batch job, such as **Working**.

Related Topics

• Properties Dialog Box, page 322

Custom Tab (Properties Dialog Box)

Sets options for user-defined properties.

If you access this tab after selecting multiple drawings, these fields appear empty, regardless of the information defined for any of the drawings individually. Any information you add to this tab override the previously defined information in corresponding rows in the selected drawings.

You can add custom drawing property labels to a template. For more information, see *Place a Custom Drawing Property Label on a Template*, page 125.

Name - Displays the name of the property.

Value - Sets the current value of the property.

Behavior - Specifies whether to inherit or override a property in the hierarchy of items in the **Management Console**. If the property is **Inherited**, the setting comes from items higher in the hierarchy. If the property is not read-only, you can provide a new, overriding value for the property by changing the behavior setting to **Override**. This new value then propagates to other items deeper in the hierarchy. If you set the behavior to **Force Override**, the property setting is forced to items deeper in the hierarchy, even if they are set to **Inherit**.

Related Topics

Properties Dialog Box, page 322

Configuration Tab

Displays the creation, modification, and status information about an object.

Plant - Displays the name of the plant. You cannot change this value.

Permission Group - Specifies the permission group to which the object belongs. You can select another permission group, if needed. Permission groups are created in the Project Management task.

Status - Specifies the current status of the selected object or filter. Depending on your access level, you may not be able to change the status of the object.

Created - Displays the date and time that the object was created.

Created by - Displays the user name of the person who created the object.

Modified - Displays the date and time when the object was modified.

Modified by - Displays the user name of the person who modified the object.

Choose Label Dialog Box

Specifies a label for a document property. This dialog box displays the labels available on the SmartPlant 3D server in the \CatalogData\Symbols\Labels\Base Templates folders.

Related Topics

- Edit Document Properties, page 334
- Properties Command, page 322
- Setting Properties: An Overview, page 321

Edit Document Properties

- 1. In the **Management Console**, right-click an item, and click **Properties** on the shortcut menu. The **Properties** dialog box displays.
- 2. Change the properties as needed. For example, you can set the coordinate system for the drawing on the **Style** tab or view the current approval information on the **Signature** tab.

Notes

- If you do not want an item to acquire a property from its parent, select the **Override** column on the **Properties** dialog box. Type a new value. This value propagates to other items deeper in the hierarchy.
- The software treats blank or cleared property values as overrides.

Related Topics

• Setting Properties: An Overview, page 321

Set Properties for Publishing Documents

- 1. In the **Management Console**, right-click an item, then select **Properties** on the shortcut menu.
- 2. Go to the **WBS Tab**.
- 3. Set the **Document Type** property as needed, such as Civil Plan.
- 4. Set the **Document Style** property as needed, such as Ortho for an orthographic drawing.
- 5. Set the **Discipline** property. If you are registered with The Engineering Framework, this property adds the **Publish** command to the right-click menu for the selected document or documents. If this is a 3D Model Data document, set the property to **SmartPlant Review Document**. If it is a drawing or report document, set the discipline to match the type of document.
- 6. Set the **Allow Publish** property to **Yes**.

Notes

- If you do not want an item to acquire a property from its parent, select the **Override** column on the **Properties** dialog box. Type a new value. This value propagates to other items deeper in the hierarchy.
- The software treats blank or cleared property values as overrides.
- Before you can publish documents in the software, you must configure your computer for TEF. The configuration includes installing SmartPlant Foundation Client and the Intergraph Schema Component and registering the model with TEF. For more information about TEF configuration, see the section titled *The Engineering Framework* in the *SmartPlant 3D Installation Guide*, available from the **Help > Printable Guides** command.

- Publishing Documents: An Overview, page 373
- Setting Properties: An Overview, page 321



Updating Documents: An Overview

Updating documents increases productivity because you can easily keep deliverables current. It is important to understand the different update capabilities.

Refreshing Document Contents

The **Refresh** command allows you to see which documents are out-of-date. For more information, see *Refresh Command (View Menu)*, page 339.

Updating Documents Using Batch Processing

The **Update Document(s)** and **Update** commands query the model to regenerate a single document or multiple documents. For volume and snapshot drawings, if you have edited the previous copy of the drawing (for example, by moving a label), the software remembers those changes and re-creates them. If batch processing is configured for the selected drawings, the update is performed on the Batch Server.

The **Update Document(s)** command is available when you right-click on a parent node in the **Management Console**. For more information, see *Update Document(s) Command*, page 340. The **Update** command is available when you right-click on a single drawing in the **Detail View**. For more information, see *Update Command*, page 341. Both commands run the **Schedule Wizard**, which gives you the ability to either run the batch job now or schedule it to run later. For more information, see *Schedule Wizard Common Tasks*, page 344.

Updating Locally

The **Update Now** command always performs a complete regeneration on the local machine for a single selected drawing. For more information, see *Update Now Command*, page 342.

Batch Processing

With batch processing, you can make sure your documents are updated without having to dedicate your workstation to the task of regenerating hundreds of documents. Using the **Update Documents** and **Update** commands, you instruct the software to update documents on a Batch Server while you continue to work on other tasks.

Batch Server - The computer on which the batch process runs is called the Batch Server. The Batch Server must have Windows 2000 or Windows XP Professional, Microsoft Message Queuing, and SmartPlant 3D Workstation loaded. The computer designated as the Batch Server is usually one that is not being used by a user to perform daily tasks, as the process of updating large numbers of documents and drawings can consume a great deal of the computer's resources. You can have one Batch Server per site database.

! Important

 The user who initially configures the Batch Server must be an administrator on that computer and have write permissions or better on the model, the symbols share, and any permission groups that access drawings.

Client - The SmartPlant 3D workstations that send batch processes to the server are called clients. Microsoft Message Queuing must be configured on any clients accessing the server to perform batch operations.

For more information about setting up batch, see *Configuring the Batch Server*, page 354.

Notes

- Most of the scheduling is stored on the Batch Server in the form of scheduled items in the Windows Task Scheduler. Once models have been assigned to the Batch Server, new processes will appear in the Processes tab of the Task Manager dialog box on that computer. The Batch Manager process indicates that at least one model can use this computer as a Batch Server. For each model selected on the Setup 3D Drawings Batch Server dialog box, one Batch Server process appears in the list. If the Batch Manager or Batch Server processes are stopped, the computer will not process batch updates.
- You can also have a Batch Tier process running for each of the selected models. This process is created when the Batch Server process finds a batch job and terminates automatically after the Batch Server has been inactive for a while.
- The default timeout value for updating documents through the Batch Server is 40 minutes. For more information on setting the timeout value, contact Intergraph Support. You can find support information on our web site http://ppm.intergraph.com/services/support.asp.

- Configuring the Batch Server, page 354
- Refresh a Document, page 339
- Removing a Model from the Batch Server, page 357
- *Update a Component*, page 353

Refresh Command (View Menu)

Compares drawings with the model for any new, modified, or deleted items or modified properties. When you click **Refresh**, the software updates the document icons to reflect the current status of the documents compared to the model. If you activate this command from a folder or application component, the software refreshes all the documents within the parent node. You can also multi-select documents within the **Detail View**.

This command is not available until you generate documents. You can access this command by right-clicking an item and selecting the command on the shortcut menu, or you can select **View > Refresh**.

Related Topics

- Refresh a Document, page 339
- Updating Documents: An Overview, page 337

Refresh a Document

- Right-click a folder, drawing component, or reports component in the Management Console. The folder or component must contain existing documents.
- 2. On the shortcut menu, click **Refresh**. The software checks the model for any differences. The drawing icons change to reflect the status of the documents compared to the model.

Note

After refreshing a folder, component, or document, you can synchronize it
with the model by right-clicking the item and then selecting Update
Document(s) on the shortcut menu.

- Managing Documents: An Overview, page 29
- Understanding Components: An Overview, page 33
- *Update a Component*, page 353

Update Document(s) Command

Updates existing drawings or reports if they are out-of-date. This command updates all the documents associated with a drawing or report component. You can also multiselect documents within the **Detail View**.

This command is available when you right-click a component in the **Management Console**.

If batch processing is configured for the selected item, the command displays the **Schedule Wizard** so you can specify whether the update should perform now or at a scheduled date and time. The update is performed on the Batch Server. For more information, see *Schedule Wizard Common Tasks*, page 344.

If batch processing is not configured, the command behaves the same as the **Update Now** command, performing a complete regeneration of an entire component contents on the local machine.

For volume drawings, the **Update Document(s)** command is not available until you place drawing volumes for a volume component in the Space Management task. For snapshot drawings, this command is not available until you create the drawings. For reports, this command is not available until you create the report by choosing a report template.

Note

- The software preserves many of the modifications you make between regenerations of volume drawings. For example, if you annotate a volume drawing and then regenerate it, your annotations still appear on the updated drawing.
- To update a single drawing, right-click a drawing in the **Detail View** and select **Update**.

Related Topics

• Updating Documents: An Overview, page 337

Update Command

Updates a single document in the **Detail View**. This command is available when you right-click on a single document or a component. You can also multi-select documents within the **Detail View**.

If batch processing is configured for the selected item, the command displays the **Schedule Wizard** so you can specify whether the update should perform now or at a scheduled date and time. The update is performed on the Batch Server. For more information, see *Schedule Wizard Common Tasks*, page 344.

If batch processing is not configured, the command behaves the same as the **Update Now** command, performing a complete regeneration of an entire drawing on the local machine.

For volume drawings, the **Update** command is not available until you place drawing volumes for a volume component in the Space Management task. For snapshot drawings, this command is not available until you create the drawings. For reports, this command is not available until you create the report by choosing a report template.

Notes

- The software preserves any modifications you make between regenerations of volume drawings. For example, if you annotate a volume drawing and then regenerate it, your annotations still appear on the updated drawing.
- The **Update** command detects when only border changes have been made and only updates the portion of the drawing that is out-of-date.
- The **Update** command works on a single selected drawing and behaves much the same as the **Update Document(s)** command, which is used for updating all documents for a selected component.

Related Topics

• Updating Documents: An Overview, page 337

Update Now Command

Updates a single document in the **Detail View** or multiple documents in the **Management Console** whether or not they are out-of-date. This command is available when you right-click on a single document or on a component. This command works on your local computer regardless of the batch configuration. You can also multi-select documents within the **Detail View**.

For volume drawings, the **Update Now** command is not available until you place drawing volumes for a volume component in the Space Management task. For snapshot drawings, this command is not available until you create the drawings. For reports, this command is not available until you create the report by choosing a report template.

Note

 The software preserves many of the modifications you make between regenerations of volume drawings. For example, if you annotate a volume drawing and then regenerate it, your annotations still appear on the updated drawing.

Related Topics

• Updating Documents: An Overview, page 337

Schedule Wizard

The **Schedule Wizard** appears when you are configured to use batch scheduling, and you right-click a drawing or report and select **Update** on the shortcut menu. You can also access the wizard by right-clicking a component and select **Update Documents**. You can submit an existing batch job request or schedule a new one. You can also multi-select documents within the **Detail View**.

For instructions on batch server configuration, see *Configuring the Batch Server*, page 354.

If you access the Schedule Wizard for a document that already has a batch job scheduled, you can edit or delete the existing batch job. For more information, see *Edit or Delete Batch Jobs*, page 358.

Note

• The default timeout value for updating documents through the Batch Server is 40 minutes. For more information on setting the timeout value, contact Intergraph Support. You can find support information on our web site http://ppm.intergraph.com/services/support.asp.

- Schedule Wizard Common Tasks, page 344
- Updating Documents: An Overview, page 337

Schedule Wizard Common Tasks

The following Schedule Wizard tasks are used when you schedule batch update jobs for drawings and reports documents.

The Schedule Wizard displays when you are configured to use a batch server and select **Update** from the shortcut menu for a selected document. For instructions on batch server configuration, see *Configuring the Batch Server*, page 354.

Submitting or Scheduling a Batch Job

The initial page of the **Schedule Wizard** allows you to specify whether you want to submit a batch update job now or schedule it for later. For more information, see *Submit or Schedule (Schedule Wizard)*, page 345.

Setting Batch Job Frequency

If you selected the **Schedule the batch job** option on the initial page of the **Schedule Wizard**, the second page specifies the batch job frequency, or how often you want the batch job to update. For more information, see *Set Batch Job Frequency* (*Schedule Wizard*), page 347.

Scheduling a Daily Batch Job

When you select the **Daily** option on the second page of the **Schedule Wizard**, you specify the day and time you want the batch job to start. For more information, see *Schedule Daily Batch Job (Schedule Wizard)*, page 349.

Scheduling a Weekly Batch Job

When you select the **Weekly** option on the second page of the **Schedule Wizard**, you specify the time and day you want the job to start on a per week basis. For more information, see *Schedule Weekly Batch Job (Schedule Wizard)*, page 350.

Scheduling a Monthly Batch Job

When you select the **Monthly** option on the second page of the **Schedule Wizard**, you specify the time and day you want the batch job to start and the months in which you want the job to run. For more information, see *Schedule Monthly Batch Job* (*Schedule Wizard*), page 351.

Scheduling a One-Time-Only Batch Job

When you select the **One time only** option on the second page of the **Schedule Wizard**, you specify the time and day you want the batch job to start. For more information, see *Schedule One-Time-Only Batch Job (Schedule Wizard)*, page 352.

Completing the Scheduling

Once you have specified the frequency, date, and time settings for your batch job schedule, the final wizard page appears. This page also appears if you selected the **One time only** option on the second page of the wizard. For more information, see *Complete Schedule (Schedule Wizard)*, page 353.

For documents that have existing batch jobs, the **Schedule Wizard** initial page is different.

Scheduling a New Batch Job

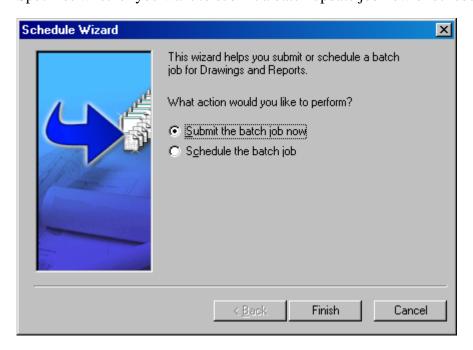
To create a new batch job for the selected document(s), select the **Schedule a new job** option. When you click **Next**, the behavior is the same as the tasks described above, starting with *Set Batch Job Frequency (Schedule Wizard)*, page 347.

Editing or Deleting Existing Batch Jobs

When you select **Update** on the shortcut menu of a document that already has a scheduled batch job, you can edit or delete an existing batch job by selecting the **Edit existing job(s)** option on the second page of the Schedule Wizard. For more information, see *Edit or Delete Batch Jobs*, page 358.

Submit or Schedule (Schedule Wizard)

Specifies whether you want to submit a batch update job now or schedule it for later.



Submit the batch job now - Specifies that the job will be automatically named and submitted to the Batch Server when you click **Finish**. The batch job name defaults using the convention **BatchJob1**, **BatchJob2**, and so forth to keep the batch job name

unique. This option is specified by default when you select **Update** from the shortcut menu for a document has no previously scheduled batch jobs.

Schedule the batch job - Specifies that you want to set a date and time for the batch job to run. Click **Next** to go to the next page of the wizard to continue scheduling the batch job. For more information, see *Set Batch Job Frequency (Schedule Wizard)*, page 347.

Note

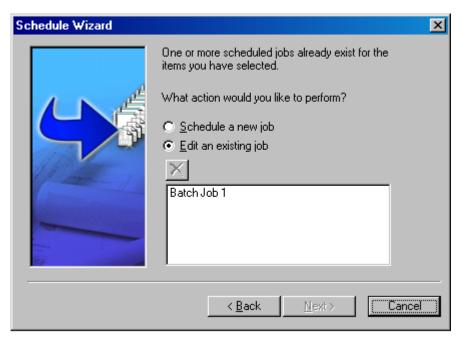
• If the document from which you accessed the **Schedule Wizard** already has batch jobs scheduled, the **Schedule the batch job** option is the default selection. When you click **Next**, you can either create a new batch job for the document or edit an existing one. You can also delete an existing batch job. For more information, see *Update an Existing Batch Job (Schedule Wizard)*, page 346.

Related Topics

- Schedule Wizard Common Tasks, page 344
- Schedule Wizard, page 343
- Updating Documents: An Overview, page 337

Update an Existing Batch Job (Schedule Wizard)

Allows you to create a new batch job for the document, edit the existing batch job, or delete the existing batch job.



Schedule a new job - Specifies that you are creating a new batch job for the document. Click **Next** to display the next page of the Schedule Wizard and define a new batch job. For more information, see *Set Batch Job Frequency (Schedule Wizard)*, page 347.

Edit an existing job - Specifies that you want to edit or delete an existing batch job definition. When you select this option, the table at the bottom of the page enables.

➤ **Delete** - Deletes the batch job selected in the table at the bottom of the dialog box. This button is only enabled when a batch job is selected.

Existing batch job table - Lists the batch jobs for the document. To edit a batch job definition, select it in the table and click **Next** to display the next page of the Schedule Wizard. For more information, see *Set Batch Job Frequency (Schedule Wizard)*, page 347.

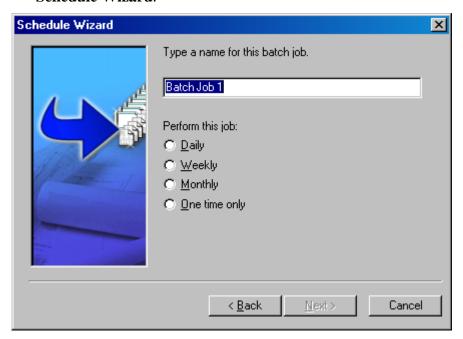
Related Topics

- Edit or Delete Batch Jobs, page 358
- Schedule Wizard Common Tasks, page 344
- Schedule Wizard, page 343
- Updating Documents: An Overview, page 337

Set Batch Job Frequency (Schedule Wizard)

Specifies the frequency with which you want to update this document or set of documents. This page of the wizard appears under two different conditions:

 If the document(s) has no previously schedule batch jobs and you select the Schedule the batch job option on the initial page of the Schedule Wizard. If the document(s) has previously scheduled batch jobs in existence, and you select the Schedule a new batch job option on the initial page of the Schedule Wizard.



Type a name for this batch job. - Specifies the batch job name. The batch job name defaults using the convention **BatchJob1**, **BatchJob2**, and so forth to keep the batch job name unique. You can change the batch job name.

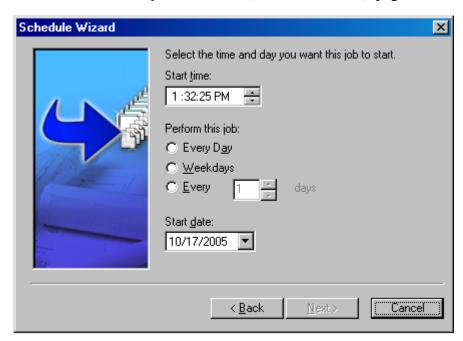
Perform this job: - Specifies the frequency option for updating the batch job. The options are described as follows:

- **Daily** The batch job will update daily at the time specified on the next page of the wizard. For more information, see *Schedule Daily Batch Job* (*Schedule Wizard*), page 349.
- **Weekly** The batch job will update weekly on the day and time specified on the next page of the wizard. For more information, see *Schedule Weekly Batch Job (Schedule Wizard)*, page 350.
- **Monthly** The batch job will update monthly on the month, day, and time specified on the next page of the wizard. For more information, see *Schedule Monthly Batch Job (Schedule Wizard)*, page 351.
- One time only The batch job will update the job one time only. This option goes to the final page of the wizard. For more information, see *Schedule One-Time-Only Batch Job (Schedule Wizard)*, page 352.

- Schedule Wizard Common Tasks, page 344
- Schedule Wizard, page 343
- Updating Documents: An Overview, page 337

Schedule Daily Batch Job (Schedule Wizard)

Specifies the time and day you want the batch job to start. When you click **Next**, the final wizard page displays the batch schedule settings. If the schedule is incorrect, click **Back** to return to previous wizard pages and make corrections. For more information, see *Complete Schedule (Schedule Wizard)*, page 353.



Start time - Specifies the time for the batch job to start. You can select a time using the scroll button or enter a time in the format shown.

Perform the job - Specifies the day option for updating the batch job. The options are described as follows:

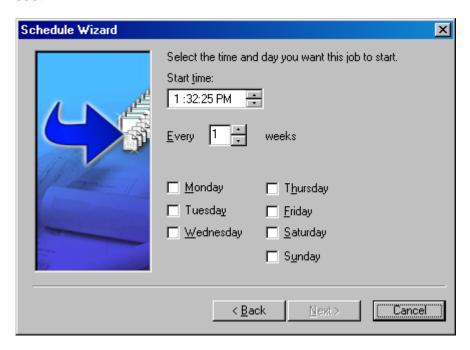
- **Every Day** The batch job will update every day at the time specified at the top of the wizard page.
- **Weekdays** The batch job will update every weekday (Monday through Friday) at the time specified at the top of the wizard page.
- Every You specify a number of days in the field provided. The batch job runs once per the specified count of days. For example, if you specified 2 as the value, the batch job will run once every 2 days.

Start date - Specifies the date on which the batch job updates begin.

- Schedule Wizard Common Tasks, page 344
- Schedule Wizard, page 343
- Updating Documents: An Overview, page 337

Schedule Weekly Batch Job (Schedule Wizard)

Specifies the time and day you want the batch job to start on a weekly basis. When you click **Next**, the final wizard page displays the batch schedule settings. If the schedule is incorrect, click **Back** to return to previous wizard pages and make corrections. For more information, see *Complete Schedule (Schedule Wizard)*, page 353.



Start time - Specifies the time for the batch job to start. You can select a time using the scroll button or enter a time in the format shown.

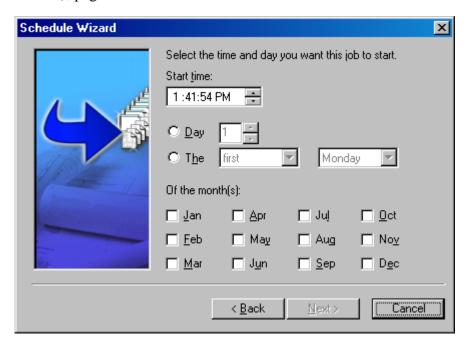
Every - Specifies a number of weeks. The batch job runs once per the specified count of weeks. For example, if you specified **2** as the value, the batch job will run once every 2 weeks.

Day checkboxes - Specifies the day of the week you want the batch job to run.

- Schedule Wizard Common Tasks, page 344
- Schedule Wizard, page 343
- Updating Documents: An Overview, page 337

Schedule Monthly Batch Job (Schedule Wizard)

Specifies the time and day you want the batch job to start and in which months you want the job to run. When you click **Next**, the final wizard page displays the batch schedule settings. If the schedule is incorrect, click **Back** to return to previous wizard pages and make corrections. For more information, see *Complete Schedule (Schedule Wizard)*, page 353.



Start time - Specifies the time for the batch job to start. You can select a time using the scroll button or enter a time in the format shown.

Day - Specify a specific day of the month. The batch job runs once per the day specified. For example, if you specified **2** as the value, the batch job will run on the second day of the selected months.

The set weekday - Specifies the day of the week you want the batch job to run. For example, you can set the batch job to run on the **second Tuesday** of every selected month.

Of the month(s) - Specifies the months you want the batch job to run. You can select multiple months.

- Schedule Wizard Common Tasks, page 344
- *Schedule Wizard*, page 343
- Updating Documents: An Overview, page 337

Schedule One-Time-Only Batch Job (Schedule Wizard)

Specifies the time and day you want the one-time-only batch job to start. When you click **Next**, the final wizard page displays the batch schedule settings. If the schedule is incorrect, click **Back** to return to previous wizard pages and make corrections. For more information, see *Complete Schedule (Schedule Wizard)*, page 353.



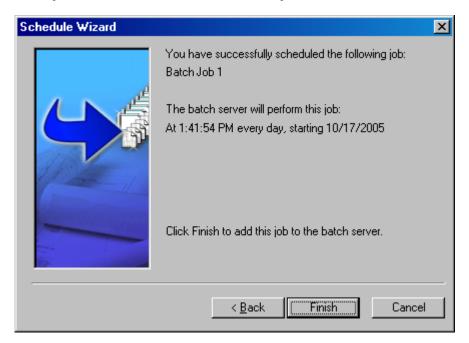
Start time - Specifies the time for the batch job to start. You can select a time using the scroll button or enter a time in the format shown.

Start date - Specifies the date on which the batch job update begins.

- Schedule Wizard Common Tasks, page 344
- Schedule Wizard, page 343
- Updating Documents: An Overview, page 337

Complete Schedule (Schedule Wizard)

Shows the completed schedule setup for the batch job. It displays the name of the batch job and the time and date when the job will run.



If the schedule is incorrect, click **Back** to return to previous wizard pages and make corrections. If the batch job is scheduled correctly, click **Finish** to process the batch job request.

Related Topics

- Schedule Wizard Common Tasks, page 344
- Schedule Wizard, page 343
- Updating Documents: An Overview, page 337

Update a Component

Before you update a component, you can refresh its documents to determine which documents are out-of-date. Right-click a component in the **Management Console** tree view, and then click **Refresh**. Documents that are out-of-date show a red X icon You do not have to refresh before updating, but it can be helpful to determine which documents are out-of-date.

- 1. Right-click a component in the **Management Console** tree view. The component must contain existing drawings or reports.
- 2. On the shortcut menu, click **Update Document(s)**. The icons for the out-of-date documents change to show they are updated. If the Batch Server is configured, the command displays the **Schedule Wizard**.

Schedule Wizard Common Tasks, page 344

Notes

- You can update an individual document by right-clicking the document in the **Detail View** and selecting **Update** or **Update Now** on the shortcut menu.
- If you place drawing property labels on a template, generate a drawing, move the labels on the drawing, and then update the drawing, the software remembers the new position of the labels on the drawing.
- If batch processing is not configured, the command behaves the same as the **Update Now** command, performing a complete regeneration of an entire component contents on the local machine.

Related Topics

• Updating Documents: An Overview, page 337

Update a Single Drawing

- 1. Right-click a document in the **Detail View**.
- 2. On the shortcut menu, click **Update** to update the document on the batch server now or create a schedule to run the batch job. Select **Update Now** to update locally. The icon for the out-of-date document changes to show it is updated .
- 3. If the batch server is configured, the **Schedule Wizard** appears.

Schedule Wizard Common Tasks, page 344

Notes

- When using the Update command and batch processing is configured for
 the selected document, the update is performed on the Batch Server. If
 batch processing is not configured, the command behaves the same as the
 Update Now command, performing a complete regeneration of the entire
 drawing on the local machine.
- If you place drawing property labels on a template, generate a drawing, move the labels on the drawing, and then update the drawing, the software remembers the new position of the labels on the drawing.

Related Topics

• Updating Documents: An Overview, page 337

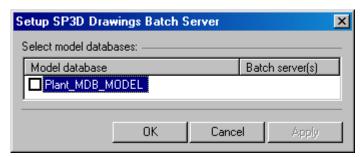
Configuring the Batch Server

The computer on which the batch process runs is called the Batch Server. The Batch Server must have Windows 2000 or Windows XP Professional, Microsoft Message Queuing, and SmartPlant 3D Workstation loaded. Microsoft Message Queuing must

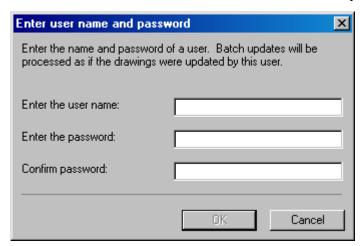
be loaded on both the Batch Server and any clients accessing the server to perform batch operations. You can have one Batch Server per site database.

The user who initially configures the Batch Server must have write permissions or better on the model, the symbols share, and any permission groups that access drawings.

- 1. Configure Microsoft Message Queuing on the Batch Server and all the clients that will use batch processing.
 - Add Message Queuing Services, page 356
- 2. Configure the Batch Server to point to the appropriate site. For more information, refer to the *Update Site Database Name and Path* procedure in the product *Installation Guide*.
- 3. Run the Batch Server Setup utility by double-clicking the SetupBatchServer.exe file, located in \SmartPlant\3D\Drawings\Middle\bin.
- 4. From the list in the **Setup SP3D Drawings Batch Server** dialog box, select all models that may use this server to process batch processing, and click OK.



5. Enter the appropriate user and password information. The user account information must be the same as the account currently logged into the computer.



6. You can verify that the correct Batch Server is associated to the model database by running the utility again and checking the **Batch server(s)** column.

Notes

- Once models have been assigned to the Batch Server, new processes will appear in the Processes tab of the Task Manager dialog box on that computer. The Batch Manager process indicates that at least one model can use this computer as a Batch Server. For each model selected on the Setup 3D Drawings Batch Server dialog box, one Batch Server process appears in the list. If the Batch Manager or Batch Server processes are stopped, the computer will not process batch updates.
- You can also have a Batch Tier process running for each of the selected models. This process is created when the Batch Server process finds a batch job and terminates automatically after the Batch Server has been inactive for a while.
- A user must be logged onto the Batch Server for it to be available to
 process batch updates. The Batch Manager process starts automatically
 whenever any user is logged onto the Batch Server computer but is not
 available when no users are logged on.
- You should set up batch processing to run as a user that has write permissions to any permission group from which a drawing can be submitted. If the write permissions do not exist, the software cannot update the document.
- The default timeout value for updating documents through the Batch Server is 40 minutes. For more information on setting the timeout value, contact Intergraph Support Services. You can find support information on our web site http://ppm.intergraph.com/services/support.asp.

Related Topics

- Removing a Model from the Batch Server, page 357
- Updating Documents: An Overview, page 337

Add Message Queuing Services

Complete these steps on the Batch Server computer and on all clients using batch processing. The instructions below are for Windows XP. Other systems may vary. For more information, contact Intergraph Support Services. You can find support information on our web site http://ppm.intergraph.com/services/support.asp.

- 1. Install the **Microsoft Message Queuing** component on your computer by clicking **Control Panel > Add/Remove Programs**.
- 2. On the left side of the dialog box, click **Add/Remove Windows Components**.

- On the Window Component Wizard dialog box, select the Message Queuing box, and click Next. A status dialog appears while Windows XP starts the Message Queuing component.
- 4. Click **Finish**. **Microsoft Message Queuing** is running on the computer.

Related Topics

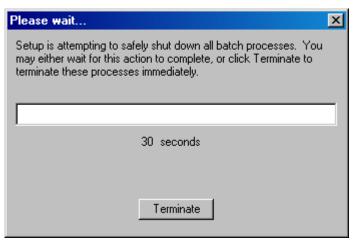
- Configuring the Batch Server, page 354
- Updating Documents: An Overview, page 337

Removing a Model from the Batch Server

- 1. Run the **Batch Server Setup** utility by double-clicking the SetupBatchServer.exe file, located in \SmartPlant\3D\Drawings\Middle\bin.
- 2. From the list in the **Setup SP3D Drawings Batch Server** dialog box, clear any check boxes for models you no longer want this computer to update using batch processing.



3. Click **OK** or **Apply**. A status dialog appears as the utility terminates the Batch Server process for the specified model database. It is recommended that you allow the utility to complete the shut down process instead of terminating immediately.



Note

 Do not try to stop processes from the Task Manager. All models are represented on the Processes tab as a Batch Server process. If you remove some models while leaving others, you cannot tell from the Processes tab which processes should be stopped and which should continue.

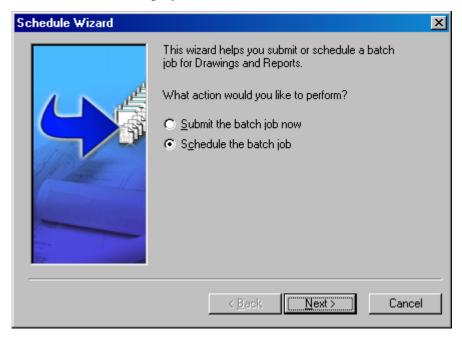
Related Topics

- Configuring the Batch Server, page 354
- Updating Documents: An Overview, page 337

Edit or Delete Batch Jobs

Complete the following steps to edit or delete an existing batch job.

1. Right-click a document that has a batch job scheduled and select **Update**. The **Schedule Wizard** displays.



One or more scheduled jobs already exist for the items you have selected.

What action would you like to perform?

Schedule a new job
Edit an existing job

Batch Job 1

2. The default setting is **Schedule the Batch Job**. Click **Next** to schedule a new batch job or edit the existing batch job.

3. To edit or delete an existing batch job, select the **Edit an existing job** option. The table at the bottom of the wizard page enables.

Next>

Cancel

< Back

Note

- To schedule a new batch job for this document, select the **Schedule new job** option and click **Next** to display the next page of the Schedule Wizard and create a new batch job schedule. For more information, see *Set Batch Job Frequency (Schedule Wizard)*, page 347.
- 4. Select a batch job in the table. To delete the batch job, click **Delete** X. To edit the batch job, click **Next** to display the next page of the Schedule Wizard and edit the batch job properties. For more information, see *Set Batch Job Frequency* (*Schedule Wizard*), page 347.

- Configuring the Batch Server, page 354
- Schedule Wizard Common Tasks, page 344
- Schedule Wizard, page 343
- Update an Existing Batch Job (Schedule Wizard), page 346
- Updating Documents: An Overview, page 337



Deleting Items: An Overview

You can delete many items that are directly or indirectly related to this task. The following list provides examples.

- Components in the **Management Console**
- Documents
- Drawing volumes
- Drawing views

In some cases, deleting an item causes other items to be deleted. For example, when you delete a drawing volume, associated views are deleted. When you delete a drawing view in a generated drawing, associated volumes and documents are deleted.

In addition, the item status and your permissions can affect whether or not you can delete the item. A drawing set to Approved cannot be deleted. However, you may be able to delete a drawing set to Working.

Related Topics

• Delete an Item, page 361

Delete an Item

Right-click a folder, component, or document. On the shortcut menu, click **Delete**.

You cannot undo a delete operation.

Note

• The **Delete** command propagates down the hierarchy. For example, if you delete a volume component, its child components (if any) and all the drawings contained in the components are deleted as well. However, When you delete a single volume drawing, the associated template, volume, and component remain unchanged. You can update the drawing component to re-create the drawing.

- Deleting Items: An Overview, page 361
- *Managing Documents: An Overview*, page 29
- Understanding Components: An Overview, page 33



Save as SmartPlant Review File: An Overview

If you are not using The Engineering Framework (TEF), you can still view your 3D Model Data in SmartPlant Review. When you create a 3D Model Data component, you set it up to save as a SmartPlant Review file. This enables the **Save As SmartPlant Review** command on the right-click menu for the 3D Model Data documents once they are created and up-to-date.

Before you can use the **Save As SmartPlant Review** command, you must install the Intergraph Schema Component on the local client machine. For more information, see *Installation Guide*, available from the **Help > Printable Guides** command.

Notes

- Before creating and setting up your 3D Model Data component, you need to define the workspace to contain all the objects required by the 3D Model Data filter subset.
- You can use SmartPlant Review to review each resulting model of Save as SmartPlant Review action. However, you cannot review multiple models.
- The **Save as SmartPlant Review** command also looks for the EFSchema file (*P3DComponent.xml*). If you are registered with TEF, the software gets the file from TEF. If you are not registered with TEF, the software looks for the file in the Symbol share.

- Save 3D Model Data for SmartPlant Review, page 365
- Save as SmartPlant Review Command, page 364

Save as SmartPlant Review Command

Saves a 3D Model Data document as a SmartPlant Review file. The file can then be viewed in SmartPlant Review. This command is available when you right-click an up-to-date 3D Model Data document if it was setup to save as a SmartPlant Review file. For more information, see *Setup a 3D Model Data Component*, page 320.

The command creates both a property data file in XML format and a graphics file in VUE format (.vue).

Notes

- Before you can use the **Save as SmartPlant Review** command, you must install Intergraph Schema Component on the local client machine. For more information, see the *Installation Guide*, available from the **Help** > **Printable Guides** command.
- The **Save as SmartPlant Review** command also looks for the EFSchema file (*P3DComponent.xml*). If you are registered with TEF, the software gets the file from TEF. If you are not registered with TEF, the software looks for the file in the Symbol share.
- You can publish .zvf files to SmartPlant Foundation and launch SmartPlant Review through SmartPlant Foundation to view the .zvf files. You can also view the .zvf file using **View and Markup** if you have SmartPlant Markup installed. You do not have to use the **Save as SmartPlant Review** command to use the SmartPlant Review features.

Related Topics

- Publishing Documents: An Overview, page 373
- Save 3D Model Data for SmartPlant Review, page 365
- Save as SmartPlant Review File: An Overview, page 363

Save as SmartPlant Review Dialog Box

Specifies the files needed to save documents for viewing in SmartPlant Review.

Note

• You can publish .zvf files to SmartPlant Foundation and launch SmartPlant Review through SmartPlant Foundation to view the .zvf files. You can also view the .zvf file using **View and Markup** if you have SmartPlant Markup installed. You do not have to use the **Save as SmartPlant Review** command to use the SmartPlant Review features.

Data file - Specifies the XML file that contains all the objects and property data that SmartPlant 3D publishes for viewing and that meet the filter query specified for the 3D Model Data component. For more information on the property data that is

published, see *The Engineering Framework Reference Guide* located in **Help > Printable Guides**.

Graphics file - Specifies the VUE file (.vue) to which the document graphics are saved. This is the file you would select in SmartPlant Review with the **File** > **Open** command.

You can browse for either file using the ellipsis button next to the field.

Related Topics

- Save 3D Model Data for SmartPlant Review, page 365
- Save as SmartPlant Review Command, page 364
- Save as SmartPlant Review File: An Overview, page 363

Save 3D Model Data for SmartPlant Review

Before you can use the **Save as SmartPlant Review** command, you must install the Intergraph Schema Component on the local client machine. For more information, see the *Installation Guide*, available from the **Help > Printable Guides** command. The **Save as SmartPlant Review** command also looks for the EFSchema file (*P3DComponent.xml*). If you are registered with TEF, the software gets the file from TEF. If you are not registered with TEF, the software looks for the file in the Symbol share.

The **Save as SmartPlant Review** command is available on the right-click menu for the 3D Model Data documents when they are set up to save as a SmartPlant Review file. For more information, see *Setup a 3D Model Data Component*, page 320.

- 1. Right-click an up-to-date 3D Model Data document and select **Save As SmartPlant Review**.
- 2. On the **Save as SmartPlant Review** dialog box, specify a property data file. This file will be an XML format file.
- 3. Specify a graphics file. This file will be a VUE format file (.vue).
- 4. Click **OK** save the SmartPlant Review files to the specified names and locations.

- Save as SmartPlant Review Command, page 364
- Save as SmartPlant Review Dialog Box, page 364
- Save as SmartPlant Review File: An Overview, page 363

Save as SmartPlant Review File: An Overview	

Revising: An Overview

The document revision process is separate from the publishing process. The **Revise** command is available on the right-click menu for drawings, reports, and 3D Model Data documents. The command creates a revision record in SmartPlant 3D. The revision record is added in the form of a blank row on the **Revision** tab of the document **Properties** dialog box.

In The Engineering Framework (TEF), the revision number is reserved only. You must publish the document to create the revision in TEF.

After reserving the revision number, you right-click the document and select **Properties**. Select the **Revision** tab and edit the **Revision** fields. You should update documents to include any new title block information.

You can now re-publish the document with the new revision information.

Note

 You can use the Revise command if you are registered with The Engineering Framework (TEF). For more information on registering with TEF, see the *Project Management User's Guide* under Help > Printable Guides.

- Publishing Documents: An Overview, page 373
- Reserve Revision Numbers, page 372
- Revise a Document, page 370
- Revise Command, page 368

Revise Command

Reserves revision numbers. This command is on the right-click menu for drawings, reports, and 3D Model Data documents. The command creates a revision record in SmartPlant 3D. The revision record is added in the form of a blank row on the **Revision** tab of the document **Properties** dialog box.

In The Engineering Framework (TEF), the revision number is reserved only. You must publish the document to create the revision in TEF.

After reserving the revision number, you right-click the document and select **Properties**. Select the **Revision** tab and edit the **Revision** fields. You should update documents to include any new title block information.

You can now re-publish the document with the new revision information.



 You can use the Revise command if you are registered with The Engineering Framework (TEF). For more information on registering with TEF, see the *Project Management User's Guide* under Help > Printable Guides.

Related Topics

- Publishing Documents: An Overview, page 373
- Reserve Revision Numbers, page 372
- Revise a Document, page 370
- Revising: An Overview, page 367

Revise Dialog Box

Allows you to revise a document in the database of the authoring tool without publishing it to The Engineering Framework (TEF) or reserve a set of revision numbers for a document within the active project or plant.



Fields with a shaded background are read-only fields and cannot be edited.

Selected documents - Displays a list of the documents selected to be revised or for which you want to reserve a set of revision numbers. You populate this list by selecting documents before you use the **Revise** command.

Engineering Tool - Opens an authoring tool-specific dialog box that allows you to select documents to add to the **Selected documents** list.

File System - Opens a standard Microsoft dialog box that allows you to select documents to add to the **Selected documents** list. When you select a file with this

Select File dialog box, the **Document Properties** dialog box opens, allowing you to specify information about the file, such as whether it is a new file or was previously published; the category, type, and subtype of the document; and the name, description, and title of the document.

Find - Opens the **Find Documents to Revise** dialog box, which allows you to search for documents to add to the **Selected documents** list.

Revision Scheme - If you have selected a new document or a document for which no revision scheme has been selected, choose the revision scheme to be applied from the list of available options. If you have selected a document with a defined revision scheme, that scheme is displayed here in a read-only format.

Current Revision in Tool Major - For existing documents, this field displays the current major revision of the document, as defined in the authoring tool, in a read-only format. For new documents, this field is empty.

Current Revision in Tool Minor - For existing documents, this field displays the current minor revision of the document, as defined in the authoring tool, in a read-only format. If the revision scheme does not use minor revision, or if the selected document has not yet been revised, this field is empty.

! Important

• If you do not use the **Minor** field when revising a document for the first time, the minor revision option will never be available for that document for future revisions.

Revise in Tool Major - From this list box, choose the next available major revision number for the document to revise it locally, without publishing the new information to TEF. If you do not want to revise the document at this time, in other words, if you want to reserve revisions numbers without revising the document, leave this field empty.

Revise in Tool Minor - From this list box, choose the next available minor revision number for the document to revise it locally, without publishing the new information to TEF. If you do not want to revise the document at this time, in other words, if you want to reserve revisions numbers without revising the document, leave this field empty. If minor revisions are not supported for the document, no options are available in this list.

Reserve Revisions in Foundation Count - For a new document, specify the quantity of revision numbers you want to reserve for the active project or plant. For an existing document, the value displayed in this field indicates the quantity of reserved revision numbers unused and still available. In this case, you can reserve more revision numbers by increasing the value displayed in this field.

Note

When you reserve revision numbers, you do so at the level of the plant or
the active project in which you are working. Numbers reserved at the plant
level are not available for use within any project, and numbers reserved at
the project level are available only to the project from which they were
reserved.

Reserve Revisions in Foundation Starting Major - Indicates the next major revision number available to be reserved. For example, if the current version of the document is B and three revision number have been reserved for that document in all the projects and not yet used, this value displayed in this field is F. For a new document that has not yet been revised and for which no revision numbers have been reserved, the default value is the first major revision of the defined revision scheme.

Reserve Revisions in Foundation Starting Minor - For new documents that have not been revised, and for which no revisions have been reserved, this list box contains two options. The first option is the first minor revision of the first major revision in the defined revision scheme. Choosing this option allows for minor revisions to the document now and in the future. The other option is the leave this field empty, which indicates that minor revisions are not now, and will never be, supported for this document.

Note

• In SmartPlant Foundation, version numbers that have been reserved are listed for the document and assigned a status of Reserved. Only versions that have been published through TEF have a status of Working.

Related Topics

• Revise Command, page 368

Revise a Document

You can revise drawings, reports, and 3D Model Data documents if you are registered with The Engineering Framework (TEF). For more information on registering with TEF, see the *Project Management User's Guide* under **Help > Printable Guides**.

1. Right-click a document, and select **Revise**. The **Revise** dialog box appears.



- You can also multi-select documents in the **Detail View**, or select a
 folder in the **Management Console** hierarchy to select all documents
 within the folder.
- 2. For a new document or a document that does not yet have a defined revision scheme, select the revision scheme you want to use from the **Revision Scheme** list. If the selected document already has a defined revision scheme, this field is read-only.

3. In the **Count** field, choose the number of revisions you want to reserve for the active project.

? Tips

- Any revision numbers you reserve are available only to the project from which you reserve them. You can also reserve numbers at the plant level; these numbers are not available to any projects, only to the plant.
- If revision numbers are currently reserved for this project, the number of remaining, unused revision numbers is displayed in this field. For example, if a user previously reserved five revision numbers for this document for the active project and two have been used, the value in this field will be "3".
- You can reserve additional numbers by adding to the value in this field. Using the example from the previous item, if you want to reserve an additional four revision numbers, choose "7" in the **Count** field.
- The Starting Major and Starting Minor fields in the Reserve Revisions in Foundation section display the first revision number available to be reserved. If no reservations have been made, these fields are empty.
- 4. In the **Revise in Tool** section, select the next available major and minor revision numbers.

💡 Tip

- Any revision numbers that have been reserved by another project are
 not available and are skipped in the revision scheme, if applicable. For
 example, if the current version of the document is B and three revision
 numbers are shown as available for the document in all projects, the
 value displayed in this field is F.
- 5. Click **OK**. The document is saved to the model database. The command creates a revision record by adding it to the document Revision properties. The command also reserves the revision number in The Engineering Framework.
- 6. Right-click the document and select **Properties**.
- 7. Go to the **Revision** tab and edit the values in the new revision row.
- 8. Update the document to update any document property title block information. For more information, see *Updating Documents: An Overview*, page 337.
- 9. Re-publish the document. The document stored in SmartPlant Foundation is not updated until you publish it through The Engineering Framework (TEF).

- Revise Command, page 368
- Revising: An Overview, page 367

Reserve Revision Numbers

You can reserve revision numbers for drawings, reports, and 3D Model Data documents if you are registered with The Engineering Framework (TEF). For more information on registering with TEF, see the *Project Management User's Guide* under **Help > Printable Guides**.

1. Right-click a document, and select **Revise**. The **Revise** dialog box appears.



- You can also multi-select documents in the **Detail View**, or select a
 folder in the **Management Console** hierarchy to select all documents
 within the folder.
- 2. For a new document or a document that does not yet have a defined revision scheme, select the revision scheme you want to use from the **Revision Scheme** list. If the selected document already has a defined revision scheme, this field is read-only.
- 3. In the **Count** field, select the number of revisions you want to reserve for the active project.



- Any revision numbers you reserve are available only to the active project. You can also reserve numbers at the plant level. Project numbers are only available at the project level; plant numbers are only available at plant level.
- 4. If a value is shown in the field, it is the number of remaining, unused reserved numbers for this document in the active project. You can change the value if you need additional reserved numbers.



- If the **Starting Major** and **Starting Minor** fields are blank, no reservations have been made previously.
- 5. Click **OK**. The reservation information is communicated with SmartPlant Foundation.

Note

• You must revise the document at least once in order to publish to TEF. A message appears if the document is not revised.

- Revise Command, page 368
- Revising: An Overview, page 367

Publishing Documents: An Overview

Using The Engineering Framework (TEF), you can publish your documents within the Drawings and Reports task. Publishing provides enterprise-wide accessibility to documents and allows you to exchange data among different software tools, thereby avoiding the creation of the same data multiple times in the different tools.

Before you can publish documents in the software, you must configure your computer with TEF. The configuration includes installing SmartPlant Foundation Client and the Intergraph Schema Component and registering the plant with TEF. For more information about TEF configuration, see the section titled *The Engineering Framework* in the *SmartPlant 3D Installation Guide*, available from the **Help** > **Printable Guides** command.

In the Common task, you can use the **Framework > Retrieve** command to create and update the Design Basis objects. For more information, see the *Common User's Guide*, available from the **Help > Printable Guides** command.

The **Publish** command is available for the following document types:

- 3D Model Data (SmartPlant Review file type)
- Orthographic Drawings, including Volume and Snapshot drawings (viewable file with links to data)
- Piping Isometric Drawings (viewable file with links to data)
- Reports (viewable Excel spreadsheet file with links to data)

Notes

- When publishing 3D Model Data, define the workspace to contain all the objects required by the 3D Model Data filter subset before creating and setting up the 3D Model Data component.
- The viewable files created when you publish drawings and reports provide relationship links to the 3D Model Data. You must also publish the 3D Model Data to provide the navigation between the viewable files and the 3D Model Data.

When you publish documents, the software:

Creates a new master document and the first revision in SmartPlant
Foundation the first time you publish a document. From that point on, the
software creates new versions and revisions when the document is
published subsequently. The software relates revisions to the master
document. You can publish subsequent revisions into a workflow, which
can be different from the workflow assigned to the original document.
Changes in the document status of a related revision change the status of
the subsequently published versions and revisions of the document.

- Publishes a visual representation of the document that you can view without SmartPlant 3D. For drawings, this is an Intergraph proprietary file, called a RAD file (.sha). For reports, the viewable file is a Microsoft Excel workbook. You can review and mark up the visual representation of the document using SmartPlant Markup or SmartSketch.
- Publishes associated data into TEF, depending on workflow approval. If the data is approved and loaded, it is used for reporting and subsequent retrieval by other authoring tools when they retrieve the latest data. The software publishes only meaningful engineering data into TEF. The published data is not enough to recreate the document in SmartPlant 3D.
- Places the published XML file and any viewable files in the appropriate SmartPlant Foundation vault. This XML file can be retrieved when users are in any authoring tool.

SmartPlant 3D receives notification when the publish is complete. The software stores the XML file in the appropriate location in the SmartPlant Foundation vault and loads the data into the SmartPlant Foundation database. The XML file can then be retrieved as published data by other tools.

Reasons to Publish

You publish documents and associated data to TEF for several reasons:

- Exchanging of data with other tools
- Sharing common data between tools
- Providing enterprise-wide accessibility to published documents
- Managing change, including workflow history, document revisions, and title block information.

Revisions and Versions of Published Documents

The first time you publish a document, the software creates a new document master and the first revision in SmartPlant Foundation. A revision (major) is an officially recognized change to a document. A version (minor) is an intermediate update that you have published. Revisions can be published for sharing or they can go through an approval process, depending on your needs. Each revisions of a document can have multiple versions in TEF. For more information on revisions, see *Revising: An Overview*, page 367.

You can include revision information within the title block of a drawing by placing drawing property labels within the drawing template. For more information, see *Place a Drawing Property Label on a Template*, page 124.

Note

• When you publish data from any authoring tool, you may not be able to view all the properties that you published in the SmartPlant Foundation client. You can customize view definitions to allow you to see additional properties. For more information on defining view definitions in the Framework schema, see the *Schema Editor User's Guide*. For further assistance with viewing data in SmartPlant Foundation, contact Intergraph Support Services. You can find support information on our web site http://ppm.intergraph.com/services/support.asp.

- Final Publish Command, page 386
- Publish to The Engineering Framework, page 382

Publish Common Tasks

The following tasks are used to publish documents. If the documents are drawings or reports, the **Publish** command publishes a viewable file with links to the data. If publishing a 3D Model Data document, the software creates a SmartPlant Review file and publishes it to The Engineering Framework.

This task list assumes you have already setup the necessary components. Refer to the following **Common Tasks** to:

- Snapshot Drawings Common Tasks, page 155
- Volume Drawings Common Tasks, page 165
- Orthographic Drawings by Query Common Tasks, page 183
- Piping Isometric Drawings by Query Common Tasks, page 195
- Spreadsheet Reports Common Tasks, page 260
- 3D Model Data Component Common Tasks, page 318

Note

• The viewable files created when you publish drawings and reports provide relationship links to the 3D Model Data. You must also publish the 3D Model Data to provide the navigation between the viewable files and the 3D Model Data.

Setting Properties for Publishing

To use the **Publish** command, you just set certain properties on your documents. For more information, see *Set Properties for Publishing Documents*, page 335.

Creating and Reserving Revision Numbers

You should create your documents by right-clicking them and selecting **Create Document(s)**. Once they are created, if you require revision numbers for the documents, use the **Revise** command to reserve the revision numbers. For more information, see *Revising: An Overview*, page 367.

Update the Drawings

Update the drawings, right-click the component and select the appropriate Update command. For more information, see *Updating Documents: An Overview*, page 337.

Save Data as a SmartPlant Review File

If you are not configured with The Engineering Framework (TEF), you can output your 3D Model Data to a SmartPlant Review (.vue) file. For more information, see *Save 3D Model Data for SmartPlant Review*, page 365.

Publish Data to The Engineering Framework

If you are configured with TEF, you can publish your 3D Model data to SmartPlant Foundation for retrieval and use in other tools. For more information, see *Publish to The Engineering Framework*, page 382.

Issue Request Documents

You can also use the **Publish** command to issue a contract request for documents. For more information, see *Issue Request Documents*, page 383.

Performing a Final Publish

You can perform a **Final Publish** when the active project has a **Merged with As-Built** status. For more information, see *Perform a Final Publish*, page 386.

Publish Command

Writes the information in the selected documents to The Engineering Framework (TEF). You can access the **Publish Documents** command by right-clicking a component or document.

The **Publish** command is available for the following types of documents:

- 3D Model Data (SmartPlant Review file type)
- Orthographic Drawings, including Volume and Snapshot drawings (viewable file with links to data)
- Piping Isometric Drawings (viewable file with links to data)
- Reports (viewable Microsoft Excel workbook file with links to data)

Note

• The viewable files created when you publish drawings and reports provide relationship links to the 3D Model Data. You must also publish the 3D Model Data to provide the navigation between the viewable files and the 3D Model Data.

You must define the **Discipline** and **Document Type** properties to enable publishing for the documents. For more information see *Set Properties for Publishing Documents*, page 335.

You may also want to specify documents to be revised, not published, or reserve revision numbers. For more information, see *Revising: An Overview*, page 367.

Related Topics

- Publish to The Engineering Framework, page 382
- Publishing Documents: An Overview, page 373

Publish Dialog Box

Provides a list of documents selected to publish to The Engineering Framework (TEF).

- Publish Command, page 378
- Publish to The Engineering Framework, page 382

Publish Tab (Publish Dialog Box)

Displays properties of the selected document or documents. If only one document is selected in the tree view, the properties displayed on this tab are the properties of that one document. If multiple documents are selected, only the properties with the same value for all documents appear. Any properties with varying values across the documents appear with blank values in these fields.

You can change some of the values assigned to one or more documents by changing the value displayed in the table. The value you enter here overrides any existing values for all selected documents.

Selected documents - Displays a list of the documents selected for publishing. You must populate this list by selecting documents before you use the **Publish** command or by clicking the buttons in the **Add** section of the dialog box. For each document, this list displays the name, the type of document, the workflow from which the document was last published, the revision and version numbers, the revision scheme, and the date when the document was last published.

Engineering Tool - Opens a dialog box to select documents to add to the **Selected documents** list. This functionality is not available in the current release.

File System - Opens a standard Microsoft dialog box that allows you to select documents to add to the **Selected documents** list. When you select a file with this **Select File** dialog box, the **Document Properties** dialog box appears, allowing you to specify information about the file, such as whether it is a new file; the category, type, and subtype of the document; and the name, description, and title of the document.

Find - Opens the **Find Documents to Publish** dialog box, which allows you to search for documents to add to the **Selected documents** list. This functionality is not available in the current release.

Last Published - Indicates the date on which the document or documents were last published.

Name - Displays the name of the document.

Source - Indicates the authoring tool in which the document was created.

Type - Displays the type of document or documents selected.

Issue Only - Allows you to issue request documents without, necessarily, republishing them. Use this option when no changes were made to a drawing, and you only want to add it to a contract.

? Tips

- Even with this option set, you can still publish the documents. If any of the documents have never been published, they must be published to TEF, regardless of this setting.
- You will receive an error message if you select multiple documents and
 activate this option when one or more of the selected documents cannot be
 changed. For example, the error message appears if the selected set of
 documents includes both a new document (for which this field can be set
 only to No) and current or frozen documents (for which this field can be
 set only to Yes). The error message prompts you to select a smaller set of
 documents.

Revision - Displays the current revision number of the selected document or documents.

Tip

• You will receive an error message if you attempt to change the value in this field when you have selected one or more documents that have conflicting revision schemes or different possible revisions. The error message prompts you to select a smaller set of documents.

Revision Scheme - Displays the revision scheme applied to the selected document or documents.



You will receive an error message indicating that this field cannot be
edited if one or more of the documents you have selected are not new or
will have a revision scheme supplied by the authoring tool. The error
message prompts you to select a smaller set of documents.

Version - Indicates the current version of the document or documents.

Workflow - Indicates the workflow to which the selected document or documents are assigned.



You will receive an error message indicating that this field cannot be
edited if one or more of the documents you have selected have conflicting
sets of possible workflows. The error message prompts you to select a
smaller set of documents.

Check and publish released claims for previously deleted items - Specifies that you want to resolve issues where deleted items were restored from an earlier version and the claim on them was released. This check takes additional time and should only be used when deleted items have been restored. This option is not supported in SmartPlant 3D in this release.

💡 Tip

• This check box should also be activated when publishing after a backup is restored or when releasing the claim on an object forces another tool to release the claim on a related object that was previously deleted. In this specific case, the tool fetches the object from As Built again and releases the claim.

Operation - Specifies the operation to perform on the selected documents.

- **Publish now** Selected documents are published immediately.
- **Background publish** Selected documents are published immediately as a separate process, allowing you to perform other tasks at the same time. This option is not supported in SmartPlant 3D in this release.
- Scheduled publish Selected documents are published in the batch mode by the authoring tool. This option is available only for tools that support batch mode and are processed by the authoring tool, not TEF client. The documents are not published immediately. Instead, the selected documents are scheduled for publish at a later time and maybe be scheduled as a recurring operation. This option is not supported in SmartPlant 3D in this release.

Custom - If applicable, opens the **Custom** dialog box. This functionality is available only if defined by your project implementation team.

Related Topics

- Publish Command, page 378
- Publish to The Engineering Framework, page 382

Issue Request Tab (Publish Dialog Box)

Allows you to view the documents associated with a specific issue request and add documents to or remove documents from a request.

Selected documents - Displays a list of the documents selected for publishing. You must select documents before you use the **Publish** command or by clicking the buttons in the **Add** section of the dialog box. For each document, this list displays the name, the type of document, the workflow from which the document was last published, the revision and version numbers, the revision scheme, and the date when the document was last published.

Engineering Tool - Opens a dialog box to select documents to add to the **Selected documents** list. This option is not supported in SmartPlant 3D in this release.

File System - Opens a standard Microsoft dialog box that allows you to select documents to add to the **Selected documents** list. When you select a file with this **Select File** dialog box, the **Document Properties** dialog box appears, allowing you to specify information about the file, such as whether it is a new file; the category, type, and subtype of the document; and the name, description, and title of the document.

Find - Opens the **Find Documents to Publish** dialog box, which allows you to search for documents to add to the **Selected documents** list. This option is not supported in SmartPlant 3D in this release.

Issue to - Contains a list of all objects (contracts) that can support issue requests. When you select an item from this list, the names of any documents associated with that object appear in the table.

Add - Creates a new item in the table for any documents highlighted in the **Selected documents** tree view.

Remove - Deletes a selected document from the table.

Document Name - Displays the names of all documents associated with the object in the **Issue to** field.

Related Topics

- Publish Command, page 378
- Publish to The Engineering Framework, page 382

Publish to The Engineering Framework

Before you can publish documents to The Engineering Framework, you must ensure that your computer is configured to work with The Engineering Framework (TEF). The configuration includes installing SmartPlant Foundation Client and the Intergraph Schema Component and registering the plant with TEF. For more information, see the SmartPlant 3D available from the **Help > Printable Guides** command in the software.

You must use the **Framework > Retrieve** command in one of the 3D tasks to import data from TEF.



- The **Framework** menu is not available in all tasks.
- 1. Right-click a component, and select **Publish**. The **Publish** dialog box appears.
- 2. The documents that appear in the **Selected documents** list are contained within the 3D Model Data component. Add any additional documents to the **Selected documents** list by clicking **Engineering Tool** or **File System**. For more information, see *Publish Dialog Box*, page 378.
- 3. Edit information as necessary for the selected documents.

💡 Tips

- When multiple documents are selected, only property values shared by all the selected documents appear in the table. Changing a value in the table changes that value for all of the selected documents.
- 4. Specify a method of publishing in the **Operation** field: **Publish now** to publish the selected documents immediately; or **Background publish** to publish the selected documents immediately as a separate process, allowing you to perform other tasks at the same time.

Note

- The **Background publish** and **Scheduled publish** options are not supported in SmartPlant 3D in this release.
- 5. Click **OK** to publish the selected documents to The Engineering Framework. For more information, see *Publishing Documents: An Overview*, page 373.

Notes

- Only updated documents can be published.
- You can verify the publishing process by starting SmartPlant Foundation Client on your computer and searching for the published document.

Related Topics

- Issue Request Documents, page 383
- Publish Command, page 378
- Publishing Documents: An Overview, page 373

Issue Request Documents

Before you can publish documents, you must ensure that your computer is configured to work with The Engineering Framework (TEF). The configuration includes installing SmartPlant Foundation Client and the Intergraph Schema Component and registering the plant with TEF. For more information, see the *SmartPlant 3D Installation Guide* available from the **Help > Printable Guides** command in the software.

You must use the **Framework > Retrieve** command in the Common task to import data from TEF.

- 1. Right-click a component, and select **Publish**. The **Publish** dialog box appears.
- 2. Click the **Issue Request** tab.
- 3. In the **Issue to** field, select the contract that you want to assign the document or documents.
- 4. Under **Selected documents**, select the documents that you want to associate with the specified contract.

- 5. Click **Add** to add the documents to the Issue Request list. To remove documents from the list, select them and click **Remove**. Click **Engineering Tools** to add documents from engineering tools, such as P&IDs or PFDs. Click **File System** to add documents from another file system, such as Microsoft Word documents or Microsoft Excel workbooks.
- 6. Click **OK** to issue the contract request for the selected documents to The Engineering Framework.
- 7. After you receive notification that the documents are associated to a contract in SmartPlant Foundation, right-click the document, then select **Refresh**.
- 8. Update the document. For more information, see *Updating Documents: An Overview*, page 337.
- 9. Review the Issue properties. Right-click the document and select **Properties**. Select the **Issue** tab to see the Issue information from TEF. You can also open the document to see the Issue information in the title block.
- 10. Publish the document with the updated Issue information to TEF. For more information, see *Publishing Documents: An Overview*, page 373.

Notes

- Only updated documents can be published.
- You can verify the publishing process by starting SmartPlant Foundation Client on your computer and searching for the published document.

- Publish Command, page 378
- Publish to The Engineering Framework, page 382
- Publishing Documents: An Overview, page 373

Managing Projects: An Overview

The Work Breakdown Structure (WBS) project is shown in the dropdown at the upper left-hand corner of the Drawings and Reports task window, next to the **Permission Group** dropdown. It shows the current active project.



In the Common task, you can create new WBS items and projects or edit existing ones. For more information, see the *Common User's Guide*.

In the Drawings and Reports task, you use projects in conjunction with publishing. The active project must be set before using the **Final Publish** command. For more information, see *Final Publish Command*, page 386.

Related Topics

- Publishing Documents: An Overview, page 373
- Select Active Project Dialog Box, page 385

Select Active Project Dialog Box

Specifies the active project. You can access the **Select Active Project** dialog box by clicking **More** in the **Active Project** box on the main toolbar. You define whether you want to look in the local **Workspace** or in the **Database** for the project by selecting the options at the top of the dialog box. The project hierarchy updates with the selection of the option.

Related Topics

• *Managing Projects: An Overview*, page 385

Final Publish Command

Performs a final publish for all previously published documents for the active project. The **Final Publish** command is enabled only when the SmartPlant 3D project status is **Merged**. For more information on setting the project status, see the *Project Management User's Guide* in **Help > Printable Guide**.

Before running the **Final Publish** command:

- Delete all project documents that are not necessary before performing a Final Publish. All remaining documents are republished as part of the Final Publish operation.
- 2. Select the **Blank Project** for all project documents. This property is on the **WBS** tab of the **Properties** dialog box. For more information, see *Properties Dialog Box*, page 322.
- 3. After clearing the **Project Name** property, update the documents. For more information, see *Updating Documents: An Overview*, page 337.

For a complete list of the steps involved in **Final Publish**, see *Perform a Final Publish*, page 386.

Related Topics

- Managing Projects: An Overview, page 385
- Publish Common Tasks, page 376
- Publishing Documents: An Overview, page 373

Perform a Final Publish

Final Publish is a task that is performed generally by the Project Leader because part of the workflow is performed within the Project Management task. For additional information on performing this task, see the *Project Management User's Guide* accessed under **Help > Printable Guides**.

- In the Project Management task, click Merge with As-Built on the Update
 Project Status dialog box. This changes the Work Breakdown Structure (WBS)
 assignments for all of the objects to as-built. The project status changes to
 Merged.
- 2. Go to the Drawings and Reports task.
- 3. Delete any documents that you do not want to include in the **Final Publish**.
- 4. Right-click the document or folder and select **Properties**. Select the **WBS** tab. Select the **Blank Project** for all project documents. For more information, see *Properties Dialog Box*, page 322.

GraphTip

- To clear the **Project Name** property for all the documents in a folder, right-click the folder and select **Properties**.
- You can also multi-select documents in the **Detail View**, then rightclick and select **Properties** to clear the **Project Name** property.
- 5. Update the documents. Right-click the document or folder and select **Update**. For more information, see *Updating Documents: An Overview*, page 337.
- 6. Set the WBS project in the **Active Project** dropdown on the main toolbar. For more information, see *Managing Projects: An Overview*, page 385.
- 7. Select **Framework > Final Publish**. A dialog box displays the command progress.



- The **Final Publish** command is enabled only when the SmartPlant 3D project status is **Merged**.
- 8. When **Final Publish** is complete, you can return to the Project Management task and set the project status to **Finished**.

- Final Publish Command, page 386
- Publish Common Tasks, page 376
- Publishing Documents: An Overview, page 373

Publishing Documents: An O	verview		

Appendix A: Troubleshooting Drawings and Reports

Log files are used to review activities and errors that occur when working with documents.

Volume and Snapshot Drawings

Two different types of drawings error logs exist: a general Drawings Error Log and error logs for each drawing sheet when generated.

The error logs reside in the following locations.

- The general Drawings Error log (Drawings.log) resides at the location specified in your Temp environment variable. For example, the path to the log might be C:\Documents and Settings\login name\Local Settings\Temp. To view your environment variable, click Start > Settings > Control Panel. Then, double-click System. On the Advanced tab, click Environment Variables.
- The error logs for the generated drawing sheets reside on the Symbols share on the SmartPlant 3D server computer. For example, the path is [Product Directory]\CatalogData\Symbols\Drawings\plant_name\...\component_name\Logs\DrawingNumber1.log, DrawingNumber2.log, and so forth.

Isometric Drawings

Log files for isometric drawings generated in this task reside on the Symbols share on the SmartPlant 3D server. For example, the path is [Product Directory]\CatalogData\Symbols\Drawings\plant_name\...\component_name\piping_ systems. You can view a message file (.mes), a piping component file (.pcf), and log files for the batch process.

Reports

The log file for reports (SP3DReports.log) resides at the location specified in your Temp environment variable. For example, the path to the log might be C:\Documents and Settings\login name\Local Settings\Temp.

Note

 You can specify the settings for drawings error logging by toggling switches in the registry. For more information, contact Intergraph Support Services. You can find support information on our web site http://ppm.intergraph.com/services/support.asp.

- Drawings and Reports: An Overview, page 11
- Troubleshooting Linked Servers: An Overview, page 391

Troubleshooting Linked Servers: An Overview

If your Site/Catalog/Plant database server is different from your Reports database server, you can use linked servers for communication between the data sources. However, if linked servers are not configured correctly, the login may fail when you run queries against a linked server.

For linked servers to work correctly, the following must be true:

- The database link must be created on the Site/Catalog/Plant database server, not the Reports database server.
- The linked server must support Windows Authentication Mode.
- A user must be connected to SQL Server using Windows Authentication Mode on both servers.
- Security account delegation must be available on the client and the sending server.

For more information about setting up linked servers, see Microsoft SQL Server documentation.

Related Topics

Appendix A: Troubleshooting Drawings and Reports, page 389

Appendix A: Troubleshooting Drawings and Reports				

Appendix B: Isometric Drawing Options

The isometric drawing options define the drawing output, which includes drawing symbols, dimensions, layers, drawing frame, attributes, material lists, weld lists, and detail sketches. Each isometric drawing style is associated with a set of options. You can specify the options using the **Edit Options** command.

For more information about the ISOGEN options, see the Alias document titled *Option Switches Reference Guide*, available from the **Help > Printable Guides** command.

The following list describes the option folders:

- **Drawing** Defines general options for the isometric drawing style. For more information, see *Drawing General Options: An Overview*, page 397.
- **Drawing Frame** Specifies the content and format of the drawing frame on a drawing. The drawing frame area can include attribute text such as the drawing number and date. For more information, see *Drawing Frame Options: An Overview*, page 446.
- **Supplementary** Provides additional options for input and output files. For more information, see *Supplementary Options: An Overview*, page 453.
- **Material List** Provides options to control the material list report on the isometric drawing. For more information, see *Material List Options: An Overview*, page 480.
- Weld List- Controls the ISOGEN weld list on the isometric drawing and controls the way that ISOGEN counts welds. For more information, see Weld List Options: An Overview, page 520. Options that deal with the representation of welds on the drawing are in the Welds folder under Drawing. A user defined weld list is allowed only with a backing sheet. That is, if an ISOGEN-generated drawing frame is in use, only the fixed layout and variable layout styles are available.
- **Neutral File** Sets options for the material take-off neutral file. For more information, see *Neutral File Options: An Overview*, page 528.
- **Labels** Specifies options for different labels on the isometric drawing. For more information, see *Labels Options: An Overview*, page 535.
- **Symbol Map** Maps part classes and ISOGEN symbol keys. For more information, see *Symbol Mapping (SymbolMAP): An Overview*, page 543.
- Alternative Texts Specifies text on the drawing that is different from the ISOGEN text. You can substitute your own text terminology or language in place of the standard ISOGEN words on the isometric drawing. For more information, see *Alternative Text Options: An Overview*, page 548.

• **Attribute Map** - Maps the ISOGEN properties with user-defined strings. For more information, see *Attribute Mapping: An Overview*, page 549.

- Edit Options Command, page 212
- Isometric Style Options Browser Dialog Box, page 212

Alias Documentation

The software delivery includes several documents published by Alias, the company that makes ISOGEN. These documents include:

- AText Reference Guide
- Option Switches Reference Guide
- ISOGEN SKEY Definitions

You can open these documents from the **Help > Printable Guides** command in the software.

To learn more about Alias, visit their web site (http://www.alias.ltd.uk).

- Alternative Text Options: An Overview, page 548
- Appendix B: Isometric Drawing Options, page 393

Graphical Representation of Options

When specifying options, you might find it helpful to view a picture of how an option changes the isometric drawing output.

For many of the options, you can refer to the Alias document titled *Option Switches Reference Guide*, available from the **Help > Printable Guides** command. You also can visit the <u>Alias web site</u> (http://www.alias.ltd.uk), from which you can download the Alias documentation.

Related Topics

• Alias Documentation, page 395

Drawing General Options: An Overview

Sets general options for the isometric drawing style.

The **Drawing** folder contains the following groups of options:

- **Content** Defines various options about the content of the drawing, such as whether or not to display enclosures and coordinates. For more information, see *Content (Drawing)*, page 399.
- **Controls** Defines various system controls on the isometric drawing. For more information, see *Controls (Drawing)*, page 403.
- **Dimensions** Controls the dimensions of the different components on the isometric drawing. For more information, see *Dimensions (Drawing)*, page 414.
- **Layers** Changes the default color for layers. For more information, see *Layers* (*Drawing*), page 421.
- **Format** Defines various options about the format of the drawing, including the isometric type and enclosure shapes. For more information, see *Format (Drawing)*, page 423.
- **Welds** Specifies information about welds on the isometric drawing. For more information, see *Welds* (*Drawing*), page 438.
- **Definitions** Customizes line weight, size, and output level of the data on the isometric drawing. For more information, see *Definitions (Drawing)*, page 443.

Property	Value	Description	Option Switch/Attribute
BorderTemplate	.sha file	Specifies the customized border template file (.sha) to use. If this option is undefined, the software uses the default backing sheet template from the Catalog. Click the ellipsis button to display the Open dialog to locate the desired file.	SmartPlant 3D
DrawingClass	Standard System Penetration Trim	Specifies the type of drawing. For Iso_Piperun and Iso_Pipeline isometrics, select the Standard option. For System isometrics, select the System option, and for Penetration Spool isometrics, select the Penetration option.	SmartPlant 3D
Visible	True/False	Controls whether the drawing is generated.	21 pos 1, 2

Content (Drawing)

Sets options for the content on isometric drawings.

Property	Value	Description	Option Switch/Attribute
BoreOutputUnit	True/False	Prints all bores in inches on the isometric drawing.	Intergraph option 43
ExcludeContinuationParts	True/False	Specifies whether continuation parts are excluded from drawing content. If set to True, continuation parts are excluded.	SmartPlant 3D
ExcludeUnconnectedTap	True/False	Specifies whether unconnected taps are excluded from drawing content. If set to True, unconnected taps are excluded.	SmartPlant 3D
GenerateSpools	True/False	Specifies whether to break pipelines into spools.	SmartPlant 3D
InstrumentTagLenLimit	True/False	Specifies the number of characters after which the software inserts a line feed in instrument tag numbers for display in an instrument bubble. If you set this value to 1, the software inserts a line feed after the first character. If you set this value to 10, the software replaces the embedded dash within the tag number with a line feed.	SmartPlant 3D

Property	Value	Description	Option Switch/Attribute
NozzleTagLabel	True/False	Uses the system-defined nozzle connection note. The default setting for this option is True for all isometric styles.	SmartPlant 3D
RpadItemCode	True/False	Specifies whether an item code and material description for reinforcement pads on the material list is generated.	77 pos 3
		If you set this option to True , the software generates an item code and material description for reinforcement pads on the material list, and plots the symbols for reinforcement pads on the drawing. The default setting for this option is False .	
ShowBIP	True/False	Shows connection coordinates at break-in points.	5 pos 8
ShowClosedEnd	True/False	Shows connection coordinates at closed ends of pipe work.	5 pos 4
ShowCoordAtRefItem	True/False	Shows connection coordinates at reference items. The default setting for this option is False .	66 pos 5
ShowCoordAtSplitPts	True/False	Shows connection coordinates at split points.	66 pos 4
ShowCptFlowArrows	True/False	Shows flow arrows on components.	17 pos 1

Property	Value	Description	Option Switch/Attribute
ShowCptTags	True/False	Shows component tags or names on graphics.	60 pos 1
ShowDrainPosition	True/False	Shows connection coordinates at drain positions.	5 pos 6
ShowEquipConn	True/False	Shows connection coordinates at equipment connections.	5 pos 2
ShowFlangeRotation	True/False	Shows flange rotation angles on the drawing.	124 pos 1
ShowMiscPosition	True/False	Shows connection coordinates at miscellaneous positions.	5 pos 7
ShowNozzles	True/False	Displays dotted nozzles on the isometric drawing.	111 pos 1
ShowOpenEnd	True/False	Shows connection coordinates at open ends of pipe work.	5 pos 3
ShowPipeLineCont	True/False	Shows coordinates for pipeline continuations.	5 pos 1
ShowRPads	True/False	Controls the automatic generation of an item code and material description on the material list and a plotted shape for the pad on the isometric drawing. The default is False for No generation of Item Code / Material Description and No plotted Pad shape.	77 pos 3
ShowSiteAssemblyTable	True/False	Controls whether the flange assembly table appears on the isometric drawing or not. The default is False for the Site Assembly Table not to be shown.	79 pos 4

Property	Value	Description	Option Switch/Attribute
ShowSupports	True/False	Shows supports on the drawing.	40 pos 1
ShowTapBrnchCoords	True/False	Shows coordinates at tapped branches.	122
ShowVentPosition	True/False	Shows connection coordinates at vent positions.	5 pos 5

Controls (Drawing)

Sets options for system controls.

Property	Value	Description	Option Switch/Attribute
AppendBoltLength	True/False	Appends the bolt length to the description in the material list.	SmartPlant 3D
AutoDrawingSplit	True/False	Invokes automatic splitting of drawings.	7 pos 1
DiagnosticLevel	LOW MEDIUM	Specifies an ISOGEN diagnostic level in the message file.	55 pos 1
	HIGH	LOW: Message file receives program version and plot messages.	
		MEDIUM: Message file receives plot and module run messages.	
		HIGH: Message file receives plot, module run, and ISOPLOT messages.	
DisconnectionMessage	True/False	Specifies an IDFGEN- generated diagnostic. True displays a message at locations where an increased tolerance is used to connect pipelines.	55 pos 2

Property	Value	Description	Option Switch/Attribute
DrawingSize	A0	Specifies a standard paper	14 pos 1, 2
	A1 A2 A3 A4 ANSI A ANSI B ANSI C	Size for the drawing. European Sizes A0: 841 x 1189 mm A1: 594 x 841 mm A2: 420 x 594 mm A3: 297 x 420 mm A4: 210 x 297 mm ANSI Sizes	
	ANSI D ANSI E	A: 8.5" x 11" B: 11" x 17" C: 17" x 22" D: 22" x 34" E: 34" x 44"	
DrawingsPerFile	One per sheet (default) One per file	Specifies the number of XML sheets that are created per drawing. If set to One per sheet , the software generates a single .sha file with one drawing page per sheet. If set to One per file , the software generates one .sha file for each drawing.	SmartPlant 3D
GenerateImpliedMat	True/False	Generates implied materials, including bolts.	SmartPlant 3D

Property	Value	Description	Option Switch/Attribute
IgnoreUserSplitPoints	True/False	Specifies whether the software ignores user split points.	7 pos 3
		If you set this option to True , the software ignores the user-specified drawing split points.	
IncludePipeSupport	True/False	Includes pipe supports in implied materials.	SmartPlant 3D
IsoScale	0 or 100 for no scaling change Integer number for scaling	Controls the finished isometric drawing size. You can increase or decrease the final plotted isometric size from the size defined by DrawingSize, or CustomHeight and CustomWidth.	32 pos 0

Property	Value	Description	Option Switch/Attribute
IsometricType	Combined Erection/O ffshore Fabrication Only Flat Spool (Flat) Flat Spool (Parallel) Spool Combined Material Erection Material Fabrication Material Spool Material	Selects the required isometric type. Combined specifies fabrication, erection, and offshore information. Erection/Offshore specifies an erection/offshore type isometric drawing. Fabrication Only specifies a fabrication only isometric drawing. Flat Spool (Flat) specifies individual flat spool isometric drawings (maximum legs flat). Flat Spool (Parallel) specifies individual flat spool isometric drawings (maximum legs parallel). Spool specifies individual spool isometric drawings (maximum legs parallel). Spool specifies individual spool isometric drawings as built orientation. Combined Material specifies a material drawing that combines fabrication, erection, and offshore information. Erection Material specifies an erection material drawing. Fabrication Material specifies a fabrication material drawing. Spool Material specifies a spool material drawing.	21 pos 1, 2

Property	Value	Description	Option Switch/Attribute
KeepIDF	True/False	Saves the IDF (Intermediate Data File) when an isometric drawing is generated. ISOGEN reads the IDF and uses it to create the isometric drawing. Third- party software packages (such as SPOOLGEN) use the IDF to create pipe fabrication spool drawings.	SmartPlant 3D

Property	Value	Description	Option Switch/Attribute
LinearWeightDenominator	None Metric Imperial	Metric linear weight denominator, regardless of the units used for	41 pos 4
		None - The linear denominator is the default. For example: LBS/FT when weight is in LBS, and KGS/M when weight is in KGS.	
		Metric - The linear denominator will be in meters. For example: LBS/M when weight is in LBS, and KGS/M when weight is in KGS.	
		Imperial - The linear denominator will be in feet. For example: LBS/FT when weight is in LBS, and KGS/FT when weight is in KGS.	
		You can specify the weight using the Drawing > Controls > WeightOutput option.	
MinAngleOffset	Value in degrees	Specifies the minimum angle to treat as a real skew.	115

Property	Value	Description	Option Switch/Attribute
NoOfDrawings	Value in range 1-99	Specifies the number of drawing sheets required for split control.	7 pos 1, 2
		☑ Note	
		This option dictates how many drawing sheets are extracted for a pipeline. Therefore, you should use this option only for special cases for single pipe runs as the setting applies to all the pipelines in a run.	
NoSymbolMapOK	True/False	Continues to extract the isometric pipeline if the software cannot find the SKEY of a component.	Intergraph option 17
NorthDirection	Top Left Top Right Bottom Left Bottom Right	Specifies the direction that the north arrow points on the drawing.	42
OverwritePlotFile	True/False	Overwrites and deletes any plot files existing with the same name.	31 pos 2
PCFOutputOnly	True/False	Specifies that the software creates the Piping Component File (PCF) only.	SmartPlant 3D

Property	Value	Description	Option Switch/Attribute
PipeLineScale	O or 100 for default scale Integer number to increase or decrease the scaling as a percentage	Increases or decreases the scale of the pipeline portion of the isometric drawing only, while leaving all other parts such as the drawing frame, material list, title block and line summary areas un-altered. Using this property, you can rescale the pipeline picture and all text pointing to it.	34
PipeLineSplitting	Value in range 90-110	Controls the amount of drawing data that triggers a drawing split. This value is a scale factor expressed as a percentage. Setting a value less than 100 causes less of the pipeline to appear on each drawing before automatic splitting occurs, resulting in a less crowded drawing.	38
		This option is intended only for modest modifications of the drawing whitespace. You should specify a value in the range 90-110. If a particular value does not work, gradually adjust the value and re-extract the drawing to check the results.	

Property	Value	Description	Option Switch/Attribute
PipeLineSplittingInTube	Normal Reject Message	Controls pipeline splitting - in tube. Normal means that the drawing splits as normal and repeatability data is generated. Reject includes an output warning in the message file. Message includes an Unacceptable message on the isometric drawing, with no repeatability data generated.	108
PipeNameInContMessages	True/False	Controls whether the pipeline name is output as part of the drawing continuation message on the isometric drawing.	38 pos 4
PlotFilePath	Path to a folder	Specifies the folder into which drawings are written. Click the browse button at the right side of the cell to display a dialog box and browse for the required directory.	ISOGEN

Property	Value	Description	Option Switch/Attribute
PostScriptSize	A0	Specifies the PostScript plotter size.	14 pos 3 and 4
	A1 A2 A3 A4 ANSI A ANSI B ANSI C ANSI D ANSI E	European Sizes A0: 841 x 1189 mm A1: 594 x 841 mm A2: 420 x 594 mm A3: 297 x 420 mm A4: 210 x 297 mm ANSI Sizes A: 8.5" x 11" B: 11" x 17"	
		C: 17" x 22" D: 22" x 34" E: 34" x 44"	
PrintedOutputPageLen	Integer value	Specifies the number of lines per page on printed output.	63
SetPipeLineDisconnected	True/False	Specifies an IDFGEN- generated diagnostic. True sets a pipeline to disconnected when an increased tolerance is used to connect pipelines.	55 pos 3
TrueScale	True/False	Adjusts the length of sections of pipe to be proportional to their actual physical length if set to True . When set to False , no adjustment is made.	95 pos 3

Property	Value	Description	Option Switch/Attribute
Units	Imperial/In ch Bores Metric/Inc h Bores Metric/M M Bores	Specifies the unit format for dimensions. The combination MM bore, FT-IN dimensions is invalid.	41 pos 1
ValOpr2ndOrient	True/False	Enables secondary orientation notes for valve operators.	SmartPlant 3D
WeightOutput	True/False	Specifies whether weight information appears on the drawing.	41 pos 5

Dimensions (Drawing)

Sets options for dimension display on the isometric drawing.

The **Drawings Dimensions** folder contains the following groups of options:

• **Column Reference** - Specifies the column references for isometric drawings. For more information, see *Column Reference (Drawing Dimensions)*, page 420.

Property	Value	Description	
AddRoundOff	True/False	Controls treatment of dimension round-offs. If you set this option to True , the software adds round-off to the next dimension. If you set it to False , the software does not carry round-off beyond branches.	117
AdditionalAllowances	True/False	Shows additional allowances with pipe dimensions. The default setting for this option is False .	9 pos 5
BranchBoreLimit	Value in range 0-999	Specifies a branch bore limit to suppress dimensions.	81 pos 4, 5, 6
BranchCptLimit	Value in range 0-99	Specifies a branch component count to suppress dimensions.	81 pos 7, 8
CoordOutputBOP	True/False	Generates BOP coordinates at changes in pipeline elevation.	SmartPlant 3D
CoordOutputBends	None Elev at changes Elev and Coords at changes Full	Controls the coordinate display at bends. Full means that a full set of coordinates is printed at bends on the drawing.	66 pos 2

Property	Value	Description	
CoordOutputBranches	None Elev at changes Elev and Coords at changes Full	Controls the coordinate display at branch intersections.	66 pos 3
CoordSupports	None Fabrication Erection Offshore All	Shows coordinates at different types of pipe supports.	66 pos 6
CoordType	Arrowed Witness Lines	Controls the output type for supplementary coordinates, that is, coordinates at bends or at branches. You can choose output with arrows or witness lines. This option does not apply to end connection coordinates. It also does not apply when the CoordOutputBends or CoordOutputBranches options are set to Full.	66 pos 1
DoubleUnits	True/False	Shows both imperial and metric units. The default setting for this option is False .	9 pos 7
Format	Composite Support Reference Only Basic Full String	Controls dimension format. Reference Only means that dimensions are reference only. Basic means that the dimensions are in basic string format, and Full String means that the dimensions are in full string format.	9 pos 1

Property	Value	Description	
FtInTolerance	Value in whole inches	Defines a value below which a ft-in dimension appears in inches.	41 pos 6, 7
Gaskets	None Included Separate	Specifies gasket dimensions. You can choose to not show gasket dimensions, to include the dimension with the component, or to treat the dimension separately.	9 pos 2
ImperialFormat	Standard Standard Dash Space Dash Stacked Dash Stacked	Controls the format of imperial measurements. Standard means ft/in format: 1' 2.3/4". Standard Dash includes a dash: 1'-2 3/4". Space Dash is 1'-2 3/4". Stacked Dash uses stacked fractions and a dash. Stacked uses stacked fractions and no dash.	41 pos 3
Inches	True/False	Uses only inches and no feet, if plotting with imperial units. This option overrides the FtInTolerance option. The default setting for this option is False .	41 pos 6
MetresCoordinates	mm m	Controls the output of coordinates in either mm or M.mm format.	41 pos 2
MetresDimensions	mm m	Controls the output of dimensions in either mm or M.mm format.	41 pos 2
MinOffset	Value in 1/100 millimeter	Specifies a minimum distance to be recognized as real movement. This value is in 1/100 millimeters.	116
NominalSize	True/False	Specifies whether nominal sizes appear at branches or reducers.	41 pos 8

Property	Value	Description	
NonLinearValves	True/False	Produces arrowed dimensions for angle, 3-way, and 4-way valve and instrument legs. Otherwise, the software prints a message. The default setting for this option is False .	9 pos 8
Overall	None Across Branches Stop At Branches Valves/Across Branches Valves/Stop At Branches	Defines information about overall dimensions. You can specify (1) no overall dimensions, (2) overall dimensions across branches, (3) overall dimensions that stop at branches, (4) overall dimensions to valve centers and across branches, or (5) overall dimensions to valve centers, stopping at branches.	118 pos 1
SeparatePulledBend	True/False	Plots separate dimensions for the pipe and bend sections of pulled bends. The default setting for this option is False .	9 pos 6
StandOutComposite	Value in range 0-99	Specifies the standout distance for string-composite dimensions.	8
StandOutOverall	Value in range 0-99	Provides overall dimension standout in millimeters.	118 pos 2,
StandOutVertical	True/False	Positions the standout dimension vertically, if possible. Otherwise, the software uses normal standout rules. The default setting for this option is False .	8 pos 4
SuppDimAsDim	True/False	Specifies whether support dimensions are on the same side (True) or opposite side (False) as normal dimensions. When this option is set to False, the software ignores any distance set in the Drawing > Dimensions > SuppStandOut option.	40 pos 2

Property	Value	Description	
SuppFormat	None String Overall	Specifies the format for support dimensions. You can choose string format or overall format. Or, you can specify that support dimensions not be shown.	40 pos 1
SuppOptions	All Fabrication Erection Offshore Erection and Offshore	Defines selective support dimensioning. You can specify that (1) all supports are dimensioned, (2) only fabrication supports are dimensioned, (3) only erection supports are dimensioned, (4) only offshore supports are dimensioned, or (5) only erection and offshore supports are dimensioned.	40 pos 5
SuppStandOut	Value in range 0-99	Specifies the support dimension standout. The software ignores this option when the Drawing > Dimensions > SuppDimAsDim option is set to False .	40 pos 3, 4
TapOnCpt	Full None Pipe Only	Specifies the dimensioning for tapped branches on inline items/flanges. Pipe Only means that dimensions display for pipes and pipe type components (elbows, tees, reducers, and so forth) only.	121 pos 1
TapOnPipe	Full None Pipe Only	Specifies the dimensioning for tapped branches on piping.	121 pos 2
USAStyle	True/False	Uses the USA dimensioning style.	9 pos 3
ValveBW	True/False	Displays the dimension to the center of butt weld valves.	80 pos 1
ValveCP	True/False	Displays the dimension to the center of compression valves.	80 pos 2

Property	Value	Description	
ValveFL	True/False	Displays the dimension to the center of flanged valves.	80 pos 5
ValveHY	True/False	Displays the dimension to the center of hygienic valves.	80 pos 7
ValveLimit	Value in range 0-999	Provides the bore limit. If you set this option and the valve is above this limit, the software generates end dimensions.	81 pos 1, 2, 3
ValvePL	True/False	Displays the dimension to the center of plain valves.	80 pos 6
ValveSC	True/False	Displays the dimension to the center of screwed valves.	80 pos 3
ValveSW	True/False	Displays the dimension to the center of socket weld valves.	80 pos 4
VertOption	Normal	Provides information about vertical pipe dimensions.	119
	Suppressed	Normal means that normal	
	Elevation	vertical pipe dimensions display, with elevations at	
		intersection points when the	
		level changes. Suppressed means that all pipeline	
		dimensions are suppressed and	
		replaced by elevations.	
		Elevation means that vertical	
		dimensions and elevations	
		display at all normal	
		dimensioning locations.	

Column Reference (Drawing Dimensions)

Specifies the columns for isometric drawing dimensions.

Name	Value	Description	
DimensionType	Prime	Specifies the dimension type as either prime or skewed.	
	Skew		
Enable	True/False	Allows placement of reference dimensions from either a structural column or grid line to one item on the isometric drawing. If set to True, the placement occurs.	
Enclosure	None	Specifies the type of enclosure to use for the column name or grid callout.	
	Square Ends		
	Round Ends		
	Diamond Ends		
	Circle		
	Double Circle		
	Ellipse		
Placement	Start Point 1st Component	Specifies the placement point used for the structural reference.	
ReferenceLocation	Column	Specifies the structural reference location as either a column on the object or a gridline.	
	GridLine		
StructureReferenceFilter	Selected Filter	Specifies a filter to use as the structural reference filter. Displays the Select Filter dialog box.	

Related Topics

Layers (Drawing)

Sets the default color for the layer.

The **Layers** folder contains the following groups of options:

• Column - Lists the layers of the isometric drawing and the content for each layer. With the isometric drawing open in the 2D Drawing Editor, you can click **Tools > Display Manager** to see a list of the layers for the isometric drawing. For more information, see *Column (Drawing Layers)*, page 422.

When using layers in a template, keep in mind that the software preserves the **Default** layer and any layer that begins with *User* (for example, a layer named **UserAnnotationLayer**) when you update drawings. Manual markups on other layers are not preserved.

Property	Value	Option Switch/ Attribute
DefaultColour	Identifies an integer value that corresponds to a color. This color is the default color for all layers, if a color is not set explicitly.	ISOGEN

Related Topics

Column (Drawing Layers)

Lists the layers of the isometric drawing and the content for each layer. With the isometric drawing open in the 2D Drawing Editor, you can click **Tools > Display Manager** to see a list of the layers for the isometric drawing.

When using layers in a template, keep in mind that the software preserves the **Default** layer and any layer that begins with *User* (for example, a layer named **UserAnnotationLayer**) when you update drawings. Manual markups on other layers are not preserved.

Layer Number	Name	LayerColour
Identifies an integer number for the layer.	Specifies a name that describes the content of the layer. For example, delivered layers include PIPE, FITTINGS, DIMTEXT, DIMLINES, MATLIST, and others.	Specifies the optional color integer number to override the default color.

Related Topics

Format (Drawing)

Sets options for the isometric drawing format, including text size and skew dimensions.

Property	Value	Description	Option Switch
AdditionalEnclosureStyle	None Type 1 Square Type 1 Round Type 1 Diamond Type 2 (as PartNoStyl e) Type 3 Weld Enclosure	Specifies the style for part number enclosures.	73 pos 5, 76 pos 1
AngleStyle BendRadius	None All except 90/180 degree All Value	Controls the output of angle information for bends and elbows in the pipeline. Sets the bend radius in 1/10 million terms.	67 pos 2 ISOGEN
	between 30 and 90	millimeters. You should use this option only if you set the bend representation to round.	
BendRepresentation	Square Round	Defines how bends look on the isometric drawing.	ISOGEN

Property	Value	Description	Option Switch
ComponentLegLength	Value in millimeters	Specifies a plotted leg length for elbows, tees, and crosses.	95
	None	☑ Note	
		Use this option with care as it can produce undesirable results on the drawing. The maximum suggested value is 18 (18 mm), and the minimum suggested value is 6 (6 mm).	
CptTagsStyle	Boxed Unboxed	Defines the style for tag numbers on inline items.	60 pos 1
DottedSymbologyEnabled	True/False	Enables dotted symbology for parts specified by DottedDimensionedFilter or DottedUnDimensionedFilter options. If both parts connected at a weld are dotted, the weld is dotted also. If a part is returned by both the filters, the part is dimensioned. When set to True, the Drawing.Format.DottedSymbology options are enabled. For more information, see DottedSymbology (Drawing Format), page 437.	Intergraph
ElbowRadius	Value between 30 and 90	Sets the elbow radius in 1/10 millimeters. You should use this option only if you set the elbow representation to round.	ISOGEN
ElbowRepresentation	Square Round	Defines how elbows look on the isometric drawing.	ISOGEN

Property	Value	Description	Option Switch
FallCutOff	Value in the units defined by FallRepres entation	Defines the minimum slope treated as a fall.	20
FallRepresentation	None Angle Ratio Gradient Percentage Imperial Incline Metric Incline	Specifies falling line indication for sloping pipelines.	19
FlangeRotationStyle	No Box Diamond Ends Round Ends Square Ends	Specifies the style of the enclosure box for flange rotation angles.	124 pos 2
FlowArrowScale	Value in range 5-15	Scales flow arrows on pipelines.	112 pos 1, 2

Property	Value	Description	Option Switch
GhostGapDimension	0 or value between 18 mm and 60 mm	Controls the plotted length of a ghost gap element. A ghost gap element is a physical gap on the plotted isometric drawing that can be used when generating individual pipeline isometric drawings, or as a link between related, but unconnected pipelines on a system type isometric drawing. 0 - Default for the normal minimum gap as controlled by ISOGEN. Value - User defined	110
		minimum gap dimension in whole millimeters.	
		The smallest allowable value is 18 mm, and the largest allowable value is 60 mm.	
InstIDEnclosureSize	No Balloon	Specifies the size of the enclosure for instrument text.	123 pos 1
	1 character	You can choose 1 through 4	
	2 character	characters on each of the two available lines within an	
	3 character	instrument balloon, or you can specify no balloon.	
	4 character		
	5 character		

Property	Value	Description	Option Switch
InstIDEnclosureStyle	No Box Diamond Ends Round Ends Triangular Ends Square Ends	Specifies the shape of the instrument name box.	123 pos 3
InstIdentification	Use Tag Use Item Code	Specifies whether to use the instrument tag or item code in the material list.	59
InsulationStyle	Alongside Pipe None Alongside Pipe and Component s	Specifies whether to show insulation along pipe only or along pipe and components.	61 pos 1

Property	Value	Description	Option Switch
MessageCircleEnclosure	Data string consisting of @, \$, and ?	Controls the physical size of the circle enclosure used to contain user input messages.	ISOGEN
	characters	The data string consists of a combination of @, \$, and ? characters.	
		The @ character signals a blank space.	
		The \$ character signals a new line.	
		The ? character indicates where the actual data from the declared record should be located in the padded record.	
MessageDiamondEnclosur e	Data string consisting of @, \$, and ? characters	Controls the physical size of the diamond enclosure used to contain user input messages.	ISOGEN
		For information about the data string, see Drawing.Format.MessageCi rcleEnclosure in this topic.	
MessagePointedEnclosure	Data string consisting of @, \$, and ?	Controls the physical size of the pointed enclosure used to contain user input messages.	ISOGEN
	characters	For information about the data string, see Drawing.Format.MessageCi rcleEnclosure in this topic.	
MessageRoundEnclosure	Data string consisting of @, \$,	Controls the physical size of the round enclosure used to contain user input messages.	ISOGEN
	and? characters	For information about the data string, see Drawing.Format.MessageCi rcleEnclosure in this topic.	

Property	Value	Description	Option Switch
MessageTriangleEnclosure	Data string consisting of @, \$, and ? characters	Controls the physical size of the triangle enclosure used to contain user input messages. For information about the data string, see Drawing.Format.MessageCircleEnclosure in this topic.	ISOGEN
NorthArrowBox	True/False	Adds a box around the north arrow.	42
OrientationFrom	Previous Position Primary Direction	Controls the output of Skew orientation. Previous Position presents the orientation direction message showing rotation relative to the previous position. Primary Direction presents orientation direction message showing rotation from primary direction.	70 pos 3
PartNoEnclSize	Default User Auto	Controls the number of characters used for the size of the circle, double circle, and ellipse enclosure styles. The default is 2, and the valid entries are from 1 to 8 characters, or set to Auto, which dynamically sizes the enclosure.	76 pos 3

Property	Value	Description	Option Switch
PartNoEnclosureLS	None Square Ends Round Ends Diamond Ends Circle Double Circle Ellipse	Generates correct diameter for double circle.	76 pos 2
PartNoEnclosureStyle	None Square Ends Round Ends Diamond Ends	Specifies the part number enclosure box shape.	73 pos 5, 76 pos 1
PartNoSpaces	Value in range 0-99	Controls the number of spaces for a Type 1, 2, or 3 enclosure.	73 1, 2/3, 4/6, 7
PartNoVisible	True/False	Suppresses the plotting of part numbers and associated enclosure boxes on the drawing.	76 pos 1
SegmentFlowArrowPlacem ent	None Pipe Run Branch	Controls flow arrow placement. Pipe Run means the flow arrow appears on the longest header pipe feature of the run. Branch means the flow arrow appears on the branch pipes connected to header pipes.	SmartPlan t 3D

Property	Value	Description	Option Switch
ShowAllFlowArrows	True/False	Displays flow arrows on the drawing. When you set this option to True , you also must set the Drawing.Format.SegmentFlowArrowPlacement option to a value other than None .	112 pos 1, 2
ShowBendAngle	True/False	Displays the bend angle for pipe bends and elbows on the drawings.	SmartPlan t 3D
ShowBendRadius	True/False	Displays the bend radius for pipe bends and elbows on the drawings.	SmartPlan t 3D
SiteAssemblyIDEnclSize	Default User Auto	Controls the number of characters used for the size of the circle, double circle, and ellipse enclosure styles. The default is 2, and the valid entries are from 1 to 8 characters, or set to Auto, which dynamically sizes the enclosure.	79 pos 9
SiteAssemblyIDEnclosure	None Diamond Ended Round Ended Triangular Diamond Rectangula r Circular	Controls the enclosure style used for site assemblies on the isometric drawing.	79 pos 3

Property	Value	Description	Option Switch
SiteAssemblyIDs	Off Numeric	Controls site assembly information on the isometric drawing.	79 pos 1
	Alphabetic	Off - Flange assembly identification not output	
		Numeric - Numeric flange assembly identification	
		Alphabetic - Alphabetic flange assembly identification	
SiteAssemblyIDsPerDrawi ng	True/False	Controls whether site assemblies are numbered per drawing or per pipeline.	79 pos 2
		True - Flange assembly identification per drawing	
		False - Flange assembly identification per pipeline	
SkewDimStandout	Value in range 0-99	Specifies the dimension line standout on skews.	100
SkewDimStyle	Box - normal standout	Specifies the style of the skew dimension.	99 pos 1
	Triangle - normal standout		
	Triangle - skew standout		
	Triangle - alternative		

Property	Value	Description	Option Switch
SkewHatchCptGap	Value in range 0-99	Specifies the gap left unhatched for inline components. This value is in 1/10 millimeters.	102 pos 3, 4
SkewHatchCutOff	Value in range 0-99	Specifies the hatch line length cut off in millimeters.	101 pos 2, 4
SkewHatchDimText	Value in range 0-99	Specifies the gap left unhatched for dimensions and text. This value is in 1/10 millimeters.	102 pos 5, 6
SkewHatchPipeGap	Value in range 0-99	Specifies the gap left unhatched for pipelines. This value is in 1/10 millimeters.	102 pos 1, 2
SkewHatchSpacing	Value in range 0-99	Specifies the hatching spacing in 1/10 millimeters.	101 pos 1, 2
SkewHatching	True/False	Specifies whether skew hatching is on or off.	101 pos 1, 2
SkewInVerticalBranch	True/False	Specifies whether to show skews on branches of falling lines.	68
SkewMinCpts	Value in range 0-99	Defines the minimum number of components in a branch before a skew is shown.	70
SkewMixed	True/False	Shows 3D skew boxes and 2D skew triangles.	99 pos 2
SkewOverall	Overall Individual	Specifies the skew depiction. Overall means a single overall box or triangle encloses the skew. Individual means individual boxes or triangles enclose each branch.	97
SkewRepresentation	3D box 2D Skew + Fall	Specifies the skew depiction. 3D box means the full 3D box/triangle is shown; 2D Skew + Fall means the 2D skew box or triangle and the fall indicator are shown.	67
SpecBrkRepresentation	Single Dual	Specifies single or dual specification break indication boxes.	114

Property	Value	Description	Option Switch
SpoolIDEnclosure	Square Brackets Diamond Ends Round Ends Triangular Box Diamond Box Square Box None	Specifies the enclosure shape for the spool identifiers.	39 pos 3
SpoolIDEnclosureStyle	Data string consisting of @, \$, and ? characters	Increases the size of the SpoolID enclosure. The data string consists of a combination of @, \$, and ? characters. The @ character signals a blank space. The \$ character signals a new line. The ? character indicates where the actual data from the declared record should be located in the padded record.	ISOGEN

Property	Value	Description	Option Switch
SpoolIDType	Numeric None Alphabetic	Turns on and off the display of pool identifiers. Uses either numeric or alphabetic sequencing for spool piece identifiers. Note This option can be used in the place of Drawing.Content.Sh owSpoolID, which is no longer supported by ISOGEN.	39 pos 1
SupportIdentification	No Tags/Spec Ref Unboxed Tags/Tags Boxed Tags/Spec Ref Unboxed Tags Boxed Tags	Specifies whether support names are shown on the isometric drawing and whether the names are boxed or unboxed. This option also controls whether supports are in the material list (BOM) and whether the tag or specification reference (item code) is in the item code field.	64 pos 1
TapBrnchInlineScale	Value in range 75-125 (%)	Provides a tapped branch scaling factor for taps on inline items.such as valves and flanges. The value represents a percentage of the main pipeline size. The minimum value is 75 and the maximum is 125.	120 pos 1, 2

Property	Value	Description	Option Switch
TapBrnchPipeScale	Value in range 1-99 (%)	Provides a tapped branch scaling factor for taps on pipe type components. Examples of pipe type components are elbows, tees, reducers, and so forth. The value represents a percentage of the main pipeline size. The minimum value is 1 and the maximum is 99.	120 pos 4, 5
TextFont	Font name	Specifies a font corresponding to a font entry in the .fif file.	4 pos 7, 8,
TextSize	Small: 2.1 Medium: 2.5 Large: 2.8 XLarge: 3.5 XXLarge: 4.2 XXXLarge: 4.9 millimeters User	Specifies the size of the text characters on the drawing.	4
TextWeight	Value in range 0-9	Specifies the thickness of the text.	4 pos 5
TextWidth	Value in range 10- 99 (in 1/10 millimeters	Specifies the character width if you are using a fixed-width font.	4 pos 3, 4

Property	Value	Description	Option Switch
TracingStyle	Alongside Pipe	Specifies where heat tracing is shown.	62
	None		
	Alongside		
	Pipe and		
	Component		
	S		
UserTextSize	Integer in range 10- 99	Specifies a user-defined size in 1/10 millimeters. You should use this option only when you set TextSize to	4 pos 1, 2
		User.	

• Appendix B: Isometric Drawing Options, page 393

DottedSymbology (Drawing Format)

Provides options for setting dotted symbology on specific isometric parts. These options are only available when the **Drawing.Format.DottedSymbologyEnabled** option is set to **True**.

Name	Value	Description
DottedDimensionedFilter	User- specified filter	Specifies a filter that selects all the parts that need to be shown dotted and dimensioned on the isometric drawing.
DottedUnDimensionedFilter	User- specified filter	Specifies a filter that selects all the parts that need to be shown dotted and undimensioned on the isometric drawing.

Related Topics

- Appendix B: Isometric Drawing Options, page 393
- Format (Drawing), page 423

Welds (Drawing)

Sets options for the display of welds on the isometric drawing.

Property	Value	Description	Option Switch/Attribute
ErectSupportWeldEnclosure	Default Circle	Specifies the style of the weld number enclosure	78 pos 3
	Diamond End	for this weld type.	
	Round End		
	Small Triangle		
	Small Diamond		
	Square		
	Dynamic Circle		
	None		
ErectWeldEnclosure	Default Circle	Circle enclosure for erection welds.	75 pos 3
	Diamond End		
	Round End		
	Small Triangle		
	Small Diamond		
	Square		
	Dynamic Circle		
	None		

Property	Value	Description	Option Switch/Attribute
FabSupportWeldEnclosure	Default Circle	Specifies the style of the weld number enclosure for this weld type.	78 pos 1
	Diamond End		
	Round End		
	Small Triangle		
	Small Diamond		
	Square		
	Dynamic Circle		
	None		
FabWeldEnclosure		enclosure for fabrication	75 pos 1
	Diamond End	welds.	
	Round End		
	Small Triangle		
	Small Diamond		
	Square		
	Dynamic Circle		
	None		

Property	Value	Description	Option Switch/Attribute
OffshoreSupportWeldEnclosure	Default Circle	Specifies the style of the weld number enclosure	
	Diamond End	for this weld type.	
	Round End		
	Small Triangle		
	Small Diamond		
	Square		
	Dynamic Circle		
	None		
OffshoreWeldEnclosure	Default Circle	Sets the style of enclosure for offshore welds.	75 pos 5
	Diamond End	weius.	
	Round End		
	Small Triangle		
	Small Diamond		
	Dynamic Circle		
	None		
SP3DModelWeldInfo	True/False	Generates weld data using the model data.	SmartPlant 3D
ShowErectSupportWeldNos	True/False	Displays weld numbers for the specified weld type.	78 pos 3

Property	Value	Description	Option Switch/Attribute
ShowErectWeldNos	True/False	Displays the erection weld numbers on the isometric drawing.	75 pos 3
ShowFabSupportWeldNos	True/False	Displays weld numbers for the specified weld type.	78 pos 1
ShowFabWeldNos	True/False	Displays the fabrication weld numbers on the isometric drawing.	75 pos 1
ShowOffshoreSupportWeldNos	True/False	Displays weld numbers for the specified weld type.	78 pos 5
ShowOffshoreWeldNos	True/False	Displays the offshore weld numbers on the isometric drawing.	75 pos 5
ShowWeldNumbers	True/False	Turns the display of weld numbers on or off.	53 pos 1
ShowWelds	True/False	Turns the display of welds and weld numbers on or off.	53 pos 1, 54
SupportWeldNumbers	True/False	Numbers support welds separately. The default is False .	53 pos 6
SupportWeldSeq	Continuous Per Drawing	Controls the numbering of support welds. If you set the SupportWeldNumbers option to True, the SupportWeldSeq option controls weld numbering, similar to the WeldNumberSequence option.	53 pos 6

Property	Value	Description	Option Switch/Attribute
SupportWeldTypeNos	True/False	Controls the numbering of support welds. If you set this option to True , the software numbers each type of support weld in a different sequence. The default is False .	53 pos 6
WeldNumberLabel	User- specified label	Specifies a label to use with the weld number on the isometric drawing. Click the ellipsis button in the field to display the Catalog Labels dialog box and select a label.	Intergraph
WeldNumberSize	Small (1.5 mm) Medium (1.8 mm) Large (2.1 mm)	Controls the size of plotted weld numbers, if welds appear.	53 pos 1
WeldTypes	All Fabrication only Erection only Offshore only	Controls which welds and weld numbers print on the drawing.	53 pos 1, 4; 54

Definitions (Drawing)

Customizes the layers on the isometric drawing. You can specify the scale and line thickness.

Property	Value
Definition Type	Specifies the type of definition. For example, you can choose Fitting, Variable Bore, and so forth.
Applies To	Specifies the component type. For a list of acceptable types, see the Values for Applies To Property table below.
DScale	Specifies an integer value for the scale. This value must be in the range 75 - 200.
Layer	Defines a number that corresponds to the layer. This number is an integer that identifies the required layer (level) to which the component type is to be assigned. This value is in the range 1 - 50.
	When using layers in a template, keep in mind that the software preserves the Default layer and any layer that begins with <i>User</i> (for example, a layer named UserAnnotationLayer) when you update drawings. Manual markups on other layers are not preserved.
Thickness Actual	Defines the actual thickness (width) of the plotted line in millimeters (a real number). This value is in the range 0.0 to 10.0.
Thickness Logical	Specifies an integer for the required thickness. This value is in the range 0 - 99.
Upper Bore	Defines the maximum bore as an integer or real number. This value must be in the current nominal size units.
Lower Bore	Defines the minimum bore as an integer or real number. This value must be in the current nominal size units.
Colour	Specifies the integer number representing the color as defined in the output drawing software. The color mapping is defined in the Mapping Values for Color Property table below.
Category	Specifies the category for the definition. For example, you can choose Fabrication, Erection, Offshore, and so forth.

Values for Applies To Property

		<u> </u>			
CAPS	CONTINUA TION	COUPLINGS	DIMENSI ON-LINES	DIMENSI ON-TEXT	END- CONNECT ORS
FILTERS	FLANGES	FRAME	FRAME- TEXT	General Fittings	HATCHIN G
HYGIENI C- CONNECT ORS	INSTRUME NTS	ISO-TEXT	LAGGING	MATERIA L-LIST	MISC- COMPON ENT- PLUG
MISC- COMPON ENT- RESTRICT OR	MISC- COMPONE NT- SLIPPLATE	MISC- COMPONEN T-SLIPRING	MISC- COMPON ENT- SPECBLI ND	MISC- COMPON ENTS	NOZZLE
OLETS	PENETRAT ION PLATES	PIPE- BLOCKS	PipeLine	REDUCER S	REFEREN CE-ITEM
SAFETY- DISC	SKEWS	SPEC- BREAKS	SPECIAL- STATUS	SPLIT- POINTS	SUPPORT S
TRACING	TRAPS	UNDIMENSI ONED- BRANCHES	UNIONS	VALVES	VALVES- 3WAY
VALVES- 4WAY	VALVES- ANGLE	WELD-BOX	WELDS		

Mapping Values for Color Property

Numeric Value	Mapped Color
0	Black
1	Blue
2	Cyan
3	Green
4	Magenta
5	Red
6	Yellow
7	White
8	Dark Blue
9	Dark Cyan
10	Dark Green
11	Dark Magenta
12	Dark Red
13	Dark Yellow
14	Dark Gray
15	Light Gray

Related Topics

Drawing Frame Options: An Overview

Sets options for the drawing frame on isometric drawings.

The **Drawing Frame** folder contains the following groups of options:

- **Units of Measure** Specifies the units for drawing frame reporttext. For more information, see *Units of Measure (Drawing Frame)*, page 448.
- **Pipeline List** Creates an embedded Excel workbook object on the isometric drawing. This object contains information about multiple pipe runs on an isometric drawing. For more information, see *Pipeline List* (*Drawing Frame*), page 449.
- CustomMTO Calls a ProgID at the end of an isometric extraction to embed a custom object on the isometric drawing. Example code for this option is delivered in Programming Resources under \\ExampleCode\Rules\PMfgCustProcess and \\ExampleCode\Rules\PMfgIsoMLExport. For more information, see \(CustomMTO\) (Drawing Frame), page 451.
- **Attributes** Maps ISOGEN attributes to attributes in the software and specifies the X- and Y-coordinates of the attribute placement. For more information, see *Attributes* (*Drawing Frame*), page 452.

Property	Value	Description	Option Switch/ Attribute
BottomMargin	Value in millimeters	Specifies the distance between the outer edge of the drawing sheet and the outer line of the drawing frame at the bottom of the drawing.	13
CustomHeight	Value in millimeters	Sets the height of a non-standard paper size.	15
		• Important	
		You must use CustomHeight and CustomWidth together. Set both to suitable values or zero. For PostScript output, set the paper size using the DrawingSize option.	

Property	Value	Description	Option Switch/ Attribute
CustomWidth	Value in millimeters	Sets the width of a non-standard paper size.	16
LeftMargin	Value in millimeters	Specifies the distance between the outer edge of the drawing sheet and the outer line of the drawing frame on the left side of the drawing.	10
ReservedAreaDrawing	Value in millimeters	Provides the distance between the bottom of the drawing area and the outer line of the drawing frame.	35 pos 1, 2, 3
ReservedAreaMatList	Value in millimeters	Provides the distance between the bottom of the material list and the outer line of the drawing frame.	35 pos 4, 5, 6
RightMargin	Value in millimeters	Specifies the distance between the outer edge of the drawing sheet and the outer line of the drawing frame on the right side of the drawing.	11
TopMargin	Value in millimeters	Specifies the distance between the outer edge of the drawing sheet and the outer line of the drawing frame at the top of the drawing.	12
SP3DBorderData	True/False	Turns the plotting of the title block and border data from the SmartPlant 3D database on and off.	SmartPlant 3D

Units of Measure (Drawing Frame)

Specifies the unit of measure format for values given in the drawing frame.

Property	Value
Pressure	Specifies a label for the pressure units of measure. Click the ellipsis button in the Value cell to display the Catalog Labels dialog box and select a label to use.
Temperature	Specifies a label for the temperature units of measure. Click the ellipsis button in the Value cell to display the Catalog Labels dialog box and select a label to use.

Related Topics

Pipeline List (Drawing Frame)

Specifies an Excel workbook object that communicates multiple pipe run information on the isometric drawing.

The **Pipeline List** folder contains the following groups of options:

• **Column** - Specifies the column layout of an Excel workbook object on the isometric drawing. For more information, see *Column (Drawing Frame Pipeline List)*, page 450

Property	Value	Description	Option Switch/Attribute
BoxOriginValue	Value in millimeters	Defines the Y-coordinate of the pipeline list. This value is in millimeters.	SmartPlant 3D
Layer	Integer	Specifies the layer for the pipeline list. This value is in the range 1 - 50.	SmartPlant 3D
ShowPipeLineListBox	True/False	Defines whether to display the pipeline list.	SmartPlant 3D

Note

• When defining embedded report layout (sizing of columns and rows), consider the report usage first. Because of a Microsoft limitation concerning the size of Windows metafile objects within other applications, the data displayed may be incomplete. Therefore, no column should be out of screen when using 100 percent zoom for the report. Otherwise some columns are ignored when the report is embedded within the drawing. The same limitation exists for rows. To preserve the maximum number of rows displayed, the total header row(s) height should be a minimum of the overall report. Using Microsoft Excel default settings, the maximum number of columns is approximately 20 and the maximum number of rows is approximately 75 (including header rows). For more information on setting the defaults in Microsoft Excel, see your Microsoft Excel documentation.

Related Topics

Column (Drawing Frame Pipeline List)

Specifies the column layout of an Excel workbook object on the isometric drawing.

Sequence	HeaderColumn	Pipeline Data Attribute	Width
Specifies the order of the columns from left to right.	Sets the text heading of each column.	Sets the contents of each column. For more information, see <i>PipeLineListBox (Labels)</i> , page 542.	Specifies the width of the column. The width must be greater than zero.

Related Topics

CustomMTO (Drawing Frame)

Calls a **ProgID** at the end of an isometric extraction to embed a custom object on the isometric drawing. Example code for this option is delivered in **Programming Resources** under *ExampleCode**Rules**PMfgCustProcess* and *ExampleCode**Rules**PMfgIsoMLExport*.

Property	Value	Description
Enabled	True/False	Controls whether or not a custom object, such as an MTO, will be included on an isometric drawing.
FramesProgID	COM object ProgID	Specifies a user-defined COM object for the post-process of data, such as MTO data, on the isometric drawing.
Layer	Layer number	Specifies the layer in the drawing in which the custom data is included.
Template Path	Path and file name	Specifies the source location of the report template to use for the custom data.
Program ProgID	COM object name	Identifies the user-defined COM object for the customization of the included data, such as MTO data.

Related Topics

Attributes (Drawing Frame)

Specifies attributes to plot in the drawing frame. You use these options with the **Attribute Map** and other **Drawing Frame** options.

Many of the attributes for the drawing frame originate from the pipe runs in the model. During an extraction, the software obtains the values for pipe run attributes from the run with the largest NPD.

For more information, see Set Drawing Frame Options, page 225.

Host Attribute	XPos	YPos	CharHeight	CharWidth
Specifies an attribute you want to appear in the drawing frame. This text can be any alphanumeric string. You map this string to an ISOGEN attribute in the Attribute Map option before the string will have meaning in the software.	Sets the X-coordinate location of the attribute text in the title block. This value is in millimeters.	Sets the Y-coordinate location of the attribute text in the title block. This value is in millimeters.	Specifies the character height of the attribute text. This value is in millimeters.	Specifies the character width of the attribute text. This value is in millimeters.

Related Topics

Supplementary Options: An Overview

Sets options for various input and output files.

The **Supplementary** folder contains the following groups of options:

- Centre of Gravity Specifies options about the center of gravity for pipes. For more information, see *Centre of Gravity (Supplementary)*, page 455.
- **Equipment Trim** Specifies options about equipment trim, or vessel trim, drawings. For more information, see *Equipment Trim (Supplementary)*, page 457.
- **Repeat File** Controls the repeatability data that is passed back to the software. For more information, see *Repeat File* (*Supplementary*), page 459.
- **Auxiliary Programs** Lists programs that run before or after ISOGEN. For more information, see *Auxiliary Programs (Supplementary)*, page 462.
- **Bending Report** Activates and configures the bending report. For more information, see *Bending Report (Supplementary)*, page 463.
- **Pipe Bending** Activates pipe bending. For more information, see *Pipe Bending (Supplementary)*, page 464.
- **Bending Simulator** Specifies the pipe bending options. For more information, see *Bending Simulator (Supplementary)*, page 465
- **Site Weld File** Defines the site weld file. For more information, see *Site Weld File (Supplementary)*, page 469.
- Additional Data Defines the DDF file. For more information, see *Additional Data (Supplementary)*, page 471.
- **Detail Sketches** Defines detail sketches on the isometric drawing. For more information, see *Detail Sketches (Supplementary)*, page 472.
- **Instrument SKEYs** Specifies user-defined symbol keys. ISOGEN treats these keys as instruments. For more information, see *Instrument SKEYs* (*Supplementary*), page 476.
- **Title Texts** Specifies user-defined text strings to be plotted on an ISOGEN-generated drawing frame. The **Visible** option for the drawing frame must be set to **False**. For more information, see *Title Texts* (*Supplementary*), page 477.
- **Report Files** Provides the names of supplementary output files. For more information, see *Report Files (Supplementary)*, page 478.

• **Data Files** - Provides the names of input data files that ISOGEN uses in various functions. For more information, see *Data Files (Supplementary)*, page 479.

Related Topics

Centre of Gravity (Supplementary)

Specifies options for center of gravity and weight calculations.

Property	Value	Description	Option Switch/Attribute
Enabled	True/False	Controls whether any center of gravity and weight calculations are done.	82 pos 1
ForDryPipe	Not Required Calc C of G Calc C of G with Insulation Calc C of G with and without Insulation	Controls which center of gravity calculations are done for dry (empty) pipe.	82 pos 4
ForWetPipe	Not Required Calc C of G Calc C of G with Insulation Calc C of G with and without Insulation	Controls which center of gravity calculations are done for wet (full) pipe.	82 pos 5

Property	Value	Description	Option Switch/Attribute
PerPipeLine	True/False	Controls whether the center of gravity and weight calculations are per pipeline or per drawing/spool.	82 pos 1
		True - Calculations are per pipeline	
		False - Calculations are per drawing/spool	
ShowInsulationWeight	True/False	Controls whether insulation weight is calculated.	82 pos 3
ShowWetWeight	True/False	Controls whether wet (full) weight is calculated.	82 pos 2

Equipment Trim (Supplementary)

Controls the generation of equipment trim, or vessel trim, drawings and associated material control files.

Property	Value	Description	Option Switch/Attribute
DataFile	One Pipeline per file	Specifies whether the information is a single pipeline or multiple pipeline.	52 pos 3
	Multiple Pipelines per file	One Pipeline per file - Single pipeline type file (only one set of header records).	
		Multiple Pipelines per File - Multiple pipeline type file (multiple sets of header records).	
MaterialFile	Accumulate per item	Controls the content of the material control file for equipment trim.	52 pos 2
	Accumulate per nozzle	Accumulate per item - All components related to a single equipment item are accumulated and written to the material control file under a single pipeline entry. Equipment trim reference is used as the pipeline reference for identification purposes.	
		Accumulate per nozzle - Components for each nozzle are kept separate and written to the material control file using the individual pipeline reference associated with each nozzle.	

Property	Value	Description	Option Switch/Attribute
PartNos	Start at 1 per nozzle	Controls the part number sequencing on the material list for equipment trim drawings.	52 pos 1
	Start at 1 per drawing	Start at 1 per nozzle - Part numbers starting at 1 for each nozzle.	
		Start at 1 per drawing - Parts sequentially numbered (starting at 1) for the whole material list.	

Repeat File (Supplementary)

Controls repeatability information passed back to the software.

Property	Value	Description	Option Switch/Attribute
ByPass	True/False	Controls whether bypass component re-ordering is done.	109 pos 6
ComponentInfo	Off With Numeric With Alphabetic	Controls whether and how component information is written to the return file. Off - Component information is not written to the return file. With Numeric - Component information is written to the return file. With Alphabetic - Full alphanumeric component information	109 pos 5
		is written to the return file. The maximum is four characters.	

Property	Value	Description	Option Switch/Attribute
Content	Standard	Controls the type of return file.	109 pos 1
	Original Revised Original ASCII Revised ASCII	Standard - Default for standard return file type containing only a list of successfully processed pipelines. Original - For a return file containing the original repeat information.	
		Revised - For a return file containing the original repeat information and new records for spool information.	
		Original ASCII - ASCII version of Original.	
		Revised ASCII - ASCII version of Revised.	
MatList	True/False	Controls whether the material list part number information is written to the return file.	109 pos 8
Path	Path to the return file	Provides the path to the return file.	ISOGEN
SplitPoints	True/False	Controls whether the split point information is written to the return file.	109 pos 3

Property	Value	Description	Option Switch/Attribute
SpoolID	Off With	Controls whether and how the Spool ID information is written to the return file.	109 pos 4
	Numeric With Alphabetic	Off - Spool ID information is not written to the return file.	
		With Numeric - Spool ID information is written to the return file.	
		With Alphabetic - Full alphanumeric spool ID information is written to the return file. The maximum is four characters.	
StartPoint	True/False	Controls whether the start point for data extraction is written to the return file.	109 pos 2
WeldNos	True/False	Controls whether weld number information is written to the return file.	109 pos 7

Auxiliary Programs (Supplementary)

Specifies information about pre- and post-processor programs.

Property	Value	Description	Option Switch/Attribute
EndProgram	Path	Specifies a post-processor program to run after ISOGEN.	ISOGEN
StartProgram	Path	Specifies a pre-processor program to run before ISOGEN.	ISOGEN
StartTimeout	Integer	Defines the number of seconds before the start program is terminated and ISOGEN starts.	ISOGEN
StopOnError	True/False	Controls whether ISOGEN runs if the pre-processor program has an error.	ISOGEN

Related Topics

Bending Report (Supplementary)

Controls options for bending pipe reports.

Property	Value	Description	Option Switch/Attribute
Append	True/False	Defines whether the file is opened in appended mode or overwrite mode.	ISOGEN
Enabled	True/False	Controls whether the properties bending report is created.	ISOGEN
Path	Path to the Bending Report	Provides the path to the Bending Report.	ISOGEN
ReportContents	Full Standard	Determines the information included in the Bending Report.	ISOGEN

Related Topics

Pipe Bending (Supplementary)

Controls options for bending pipe.

Property	Value	Description	Option Switch/Attribute
Enabled	True/False	Controls whether the properties of the pipe bending object are written to the Data Definition File (DDF).	ISOGEN

Related Topics

Bending Simulator (Supplementary)

Controls options for bending pipe.

Property	Value	Description	Option Switch/Attribute
AdjacentMaterialSelectionFactor	Integer	Specifies a value for the adjacent material selection factor. This factor provides the relative importance of adjacent material in selection. Values used for selection factors must be exclusive.	ISOGEN
AllowAdditionalAdjacentMaterial	True/False	Controls whether additional adjacent material is allowed.	ISOGEN
AllowAdditionalMaterialAtEnds	True/False	Controls whether additional material at ends is allowed.	ISOGEN
AllowBenderChange	True/False	Controls whether a bender change is allowed.	ISOGEN
AllowHandChange	True/False	Controls whether a hand change is allowed.	ISOGEN

Property	Value	Description	Option Switch/Attribute
AllowMachineChange	True/False	Controls whether a machine change is allowed.	ISOGEN
BenderSelectionFactor	Integer	Specifies a value for the bender selection factor. This factor provides the relative importance of the bender in selection. Values used for selection factors must be exclusive.	ISOGEN
CutAndWeldCutPiece	True/False	Controls whether cut and weld cut piece is allowed.	ISOGEN
DoClashChecking	True/False	Controls whether clash checking is done.	ISOGEN
HandSelectionFactor	Integer	Specifies a value for the hand selection factor. This factor provides the relative importance of the hand in selection. Values used for selection factors must be exclusive.	ISOGEN
IncludeBendSpringback	True/False	Controls whether to include bend springback.	ISOGEN

Property	Value	Description	Option Switch/Attribute
IncludeBendStretching	True/False	Controls whether to include bend stretching.	ISOGEN
IncludeWeldShrinkage	True/False	Controls whether to include weld shrinkage.	ISOGEN
MachineOverBenderPriority	True/False	Controls whether the machine has priority over the bender.	ISOGEN
MachineSelectionFactor	Integer	Specifies a value for the machine selection factor. This factor provides the relative importance of the machine in selection. Values used for selection factors must be exclusive.	ISOGEN
MaterialAtEndsSelectionFactor	Integer	Specifies a value for the machine selection factor. This factor provides the relative importance of the machine in selection. Values used for selection factors must be exclusive.	ISOGEN

Property	Value	Description	Option Switch/Attribute
NominatedBender	String	Specifies the name of the nominated bender.	ISOGEN
RejectInvalidManufacturer	True/False	Controls whether to reject an invalid manufacturer.	ISOGEN
TemplateLength	Integer	Specifies the length of the template for bending.	ISOGEN
UseBenderPriority	True/False	Controls whether to use the bender priority.	ISOGEN
UseMachinePriority	True/False	Controls whether to use the machine priority.	ISOGEN

Site Weld File (Supplementary)

Specifies information about the site weld file.

The **Site Weld File** folder contains the following groups of options:

• **Column** - Specifies the columns for the site weld file. For more information, see *Column* (*Supplementary Site Weld File*), page 470.

Property	Value	Description	Option Switch/Attribute
Delimiter	User- defined value	Specifies a delimiter to use for the header lines that appear at the top of a SiteWeldInfo file.	ISOGEN
Enabled	True/False	Controls whether the site weld file is created.	ISOGEN
FileFormat	Fixed Delimited	Sets the format of the site weld file.	ISOGEN
HeaderLines	String for the column header	Specifies the text string for the column header. Use a backslash (\) to start a new line and pipe character () to produce a vertical break. You must make sure the column headers match the column positions set in the Material List Definition file (MLD).	ISOGEN
Path	Path to the site weld file	Provides the path to the site weld file.	ISOGEN

Related Topics

Column (Supplementary Site Weld File)

Specifies the columns for the site weld file.

Attribute Name	Width	Start
Specifies a property to appear in the column.		Specifies the starting point of the column. The valid range is 1-999.

Related Topics

Additional Data (Supplementary)

Defines additional data for the Data Definition File (DDF).

Property	Value	Description	Option Switch/Attribute
Enabled	True/False	Controls whether the data in the property DDFLines is added to the DDF.	ISOGEN
DDFLines	Text	Specifies a block of text to be added to the DDF.	ISOGEN

Related Topics

Detail Sketches (Supplementary)

Specifies the characteristics of detail sketches on the isometric drawing.

The **Detail Sketches** (**Supplementary**) folder contains the following groups of options:

• **Sketch Mapping** - Maps part classes with symbol files for the purpose of defining detailed sketches. For more information, see *Sketch Mapping* (*Supplementary Detail Sketches*), page 475

Property	Value	Description	Option Switch/Attribute
Colour	Integer	Provides the color of the label on the detail sketch.	ISOGEN
FileFormat	IGR DXF	Sets the format of the detail sketch symbols. In most cases, this setting agrees with the drawing output format. The IGR option corresponds with the Intergraph SmartSketch product.	ISOGEN
LabelLayer	Integer	Specifies the layer on which the label text resides. For example, the label text in Detail A is the letter A .	ISOGEN
LabelType	ALPHA NUMBER	Specifies whether you want sketches ordered alphabetically (A, B,C) or numerically (1, 2, 3).	ISOGEN
LabelX	Value in millimeters	Provides the X-coordinate of the label text, relative to the bottom left corner of a sketch.	ISOGEN
LabelY	Value in millimeters	Provides the Y-coordinate of the label text, relative to the bottom left corner of a sketch.	ISOGEN
NoteFormat	IGR CEL	Specifies the format of informational notes, which refer to a pipeline, spool, or	ISOGEN
	DXF	component in the drawing.	

Property	Value	Description	Option Switch/Attribute
NoteHeight	Value in millimeters	Specifies the height of the informational notes.	ISOGEN
NotePosition	DEFAULT LOCAL OVERFLOW	Specifies the location of informational notes. You can choose to plot the notes along the bottom of the drawing, near the reference, or on a separate sheet.	ISOGEN
NoteWidth	Value in millimeters	Specifies the width of the informational notes.	ISOGEN
Path	Text box	Sets the detailed sketch directory containing SYM or DXF data, or sets the location of a DGN CEL file, which is a cell library. You specify the symbol file name in Supplementary.Detail Sketches.SketchMapping (SketchSymbol column).	ISOGEN
ShowDetailSketch	True/False	Determines whether the detail sketch facility is enabled.	ISOGEN
SketchHeight	Value in millimeters	Sets the height of user- generated sketches. This setting must be the same for all sketches. The value can be an integer or decimal number but should not exceed 50 mm.	ISOGEN
SketchPosition	default local overflow	Specifies the location of the detail sketches. You can choose to plot sketches along the top of the drawing, near the reference, or on a separate sheet.	ISOGEN

Property	Value	Description	Option Switch/Attribute
SketchWidth	Value in millimeters	Sets the width of user- generated sketches. The value can be an integer or decimal number but should not exceed 50 mm.	ISOGEN
TextFont	Font name	Specifies a font corresponding to a font entry in the .fif file.	ISOGEN
TextHeight	Value in millimeters	Specifies the size of the label text. This setting overrides Drawing.Format.Text Size. This value can be an integer or real number.	ISOGEN
TextWeight	Value in millimeters	Specifies the thickness of the label text. This value is an integer.	ISOGEN

Sketch Mapping (Supplementary Detail Sketches)

Maps part classes with symbol files for the purpose of defining detailed sketches.

You can map multiple symbol files to a given part class. The software places each symbol as a detail sketch on the isometric drawing and prints the appropriate number of call-outs adjacent to each occurrence of the part class.

You can also specify a label and value. If the value you specify matches the value returned by the label, the software generates a detail on the drawing.

SP3DPartClass	SketchSymbol	SketchChkLabel	SketchChkValue
Specifies the name of a part class. Part classes and their associated data are defined in the reference data. For information about creating or modifying part classes, see the SmartPlant 3D Reference Data Guide available from the Help > Printable Guides command.	Specifies the name of a symbol file for the detail sketch. The software checks to see if this file exists in the Supplementary.Detail Sketches.Path location.	Names the label, such as SupportPartName.	Defines a value. If this value matches the value returned by the label, then the software generates a detail on the drawing.

Related Topics

Instrument SKEYs (Supplementary)

Specifies user-defined symbol keys.

ISKEYText

Specifies a symbol key text string.

Related Topics

Title Texts (Supplementary)

Specifies user-defined text strings for plotting on an ISOGEN-generated drawing frame.

TitleText

Specifies a text string for the drawing frame.

Related Topics

Report Files (Supplementary)

Provides the names of supplementary output files. One example is a file containing centerline lengths.

☑ Note

• To add a new row to the grid, click in the blank area of the right pane, and press **Insert** on the keyboard.

ReportFileType	FilePath
Specifies the type of output file. In some cases, this option also specifies the behavior of the file, such as append or	Browses your computer or network for a file
overwrite.	name.

Related Topics

Data Files (Supplementary)

Provides the names of input data files ISOGEN uses in various functions. For example, ISOGEN can calculate weights of components if a data file containing weight information is available.

Note

• To add a new row to the grid, click in the blank area of the right pane, and press **Insert** on the keyboard.

DefinitionDataFileType	FilePath
Specifies the type of input file.	Browses your computer or network for a file name.

Related Topics

Material List Options: An Overview

Sets options for controlling position, format, and contents of the parts list.

The **Material List** folder contains the following groups of options:

- **Cut List** Defines the cut pipe list on the isometric drawing. The cut piece list is a report on computed pipe lengths. For more information, see *Cut List (Material List)*, page 490.
- **Fixed Layout** Defines the original ISOGEN material list options. ISOGEN plots the column headings. For more information, see *Fixed Layout (Material List)*, page 497.
- Variable Layout Defines the alternative variable layout material list. This style is a variation of the basic fixed layout type, with additional user options for header text. For more information, see *Variable Layout* (*Material List*), page 500.
- User Defined Defines the user defined material list. The software prints this type of list into a drawing frame that contains a pre-existing table with header text. For more information, see *User Defined (Material List)*, page 506. A user defined material list is allowed only with a backing sheet. That is, if an ISOGEN-generated drawing frame is in use, only the fixed layout and variable layout styles are available.
- **Summary File** Controls the summary file for the material list. For more information, see *Summary File (Material List)*, page 515.
- **Transfers** Reassigns particular record types to a new sort group. For more information, see *Transfers* (*Material List*), page 517.

Fixed Layout produces a simple material list. Variable Layout and User Defined are fully user-definable: Variable Layout is based on columns and lines, and User Defined is based on X- and Y-coordinate positions.

Property	Value	Description	Option Switch/Attribute
ActiveList	Fixed Variable User Defined	Controls the format of the material list. Note You can use only a Fixed material list with an ISOGEN drawing frame. If the Visible option under DrawingFrame is False, you must set ActiveList to Fixed.	23 pos 3
BoltAccumulation	Normal Suppress None	Controls how bolt components are included on or excluded from the material list. None means each occurrence of an item results in an individual entry on the material list. Note This option does not apply to a User Defined material list, nor when the MLType option for the Fixed or Variable styles is set to Special.	74 pos 5

Property	Value	Description	Option Switch/Attribute
BoltDiameterUnits	As Drawing INCH MM	Controls the units for bolt diameter. For the As Drawing setting, see the Drawing.Controls.Units	65 pos 2
BoltLengthUnits	As Drawing INCH MM	option. Controls the units for bolt lengths. For the As Drawing setting, see the Drawing.Controls.Units option.	65 pos 1
Drg1of1OnSingleIsos	True/False	Specifies whether the software adds 10 to the value of the MaterialListOverflow option. If you set this option to True, the software adds 10 to the value of the MaterialListOverflow option. The Drg1of1 option controls the output of the string Drg 1 of 1 on single-sheet isometric drawings.	30

Property	Value	Description	Option Switch/Attribute
ErectAccumulation	Normal Pipe Only Suppress	Controls the accumulation of erection materials. Pipe Only means only piping is accumulated. Suppress means the erection items do not appear in the material list at all. Note This option does not apply to a User Defined material list, nor when the MLType option for the Fixed or Variable styles is set to Special.	74 pos 2

Property	Value	Description	Option Switch/Attribute
FabAccumulation	Normal Pipe Only Suppress None	Controls the accumulation of fabrication materials. Pipe Only means only pipe is accumulated. Suppress means the fabrication items do not appear in the material list at all. None means the software does not accumulate items but lists them separately in the material list. Note Note This option does not apply to a User Defined material list, nor when the MLType option for the Fixed or Variable Layout styles is set to Special.	74 pos 1
GasketAccumulation	Normal Suppress None	Controls the accumulation of gaskets. Note This option does not apply to a User Defined material list, nor when the MLType option for the Fixed or Variable Layout styles is set to Special.	74 pos 4
InstrSpecialtyCode	Specified item code	Specifies the instrument specialty item code to use in the material list.	SmartPlant 3D

Property	Value	Description	Option Switch/Attribute
ItemCodeLength	Value in range 6 - 29	Sets the number of characters in the material list item code.	28
		☑ Notes	
		The maximum number of item code characters is 29.	
		Changes to this option cause the material list to be wider or narrower.	
LinearQuantityStyle	n.n M n.n ft n MM n ft n in	Specifies the style for linear quantities in the material list. You can choose from:	24 pos 2
	n.nnn M n.nnn ft	Meters (M) or Feet: 1 decimal place	
		Millimeters (MM) or Feet and Inches	
		Meters or Feet: 3 decimal places	
MaterialListOverflow	Same Sheet Continuation Sheet	Controls the overflow of the material list to subsequent drawings.	30
MaterialListSplitting	Per Drawing Per Pipeline	Controls the plotting of the material list when a pipeline splits onto two or more drawings	24 pos 1

Property	Value	Description	Option Switch/Attribute
OffshoreAccumulation	Normal Pipe Only Suppress	Controls the accumulation of offshore materials. Pipe Only means only piping is accumulated. Suppress means the offshore items do not appear in the material list at all. Note This option does not apply to a User Defined material list, nor when the MLType option for the Fixed or Variable Layout styles is set to Special.	74 pos 3

Property	Value	Description	Option Switch/Attribute
OverflowDrawingID	Next ID Alpha suffix	Controls the numbering of overflow drawing sheets.	30
		☑ Note	
		This option applies only when you set the Material List.Material List Overflow option to Continuation Sheet, in order to produce a second sheet displaying the material list continuation. Next ID means the second isometric drawing has the next sheet number in the sequence. Alpha Suffix means the second isometric drawing has the second isometric drawing has the second isometric drawing has the same sheet number as the first one, with a	
		character suffix, for example, A, B, and so forth.	
PipeOidAsItemCode	Pipe Oid value	Specifies a Pipe Oid value will be used as the item code in the material list.	SmartPlant 3D
TextFont	Font name	Specifies a font corresponding to a font entry in the .fif file.	27 pos 7, 8, 9

Property	Value	Description	Option Switch/Attribute
UserComponentQTY	Integer value	Specified user component quantity.	SmartPlant 3D
UserDefinedItemCode	Specified user-defined bolt item code	Specifies a user-defined bolt item code to use in the material list.	SmartPlant 3D
Visible	True/False	Controls the display of the material list. If you set this option to False , the software turns off all material list types, except the material control file and cut list. The software controls these files separately.	23 pos 1
WeightsStyle	None	Controls the output of component weights.	41 pos 5
	Total Individual	Total - Each component weight is a total weight. For example: quantity x unit weight.	
		Individual - Each component weight is the individual weight of the component type. The quantity is not multiplied by the unit weight.	

Property	Value	Description	Option Switch/Attribute
WeldAccumulation	Normal Suppress None	Controls the accumulation of welds. Suppress means the welds do not appear in the material list at all. None means the software does not accumulate items but lists them separately in the material list. Note This option does not apply to a User Defined material list, nor when the MLType option for the Fixed or Variable Layout styles is set to Special.	74 pos 6

Cut List (Material List)

Sets options for the cut list, which provides a list of pipes that are cut into smaller lengths during construction.

The **Cut List** folder contains the following groups of options:

- **Fixed Layout** Defines the fixed layout cut list format. For more information, see *Fixed Layout (Material List Cut List)*, page 492.
- **User Defined** Defines the user-defined cut list format. For more information, see *User Defined (Material List Cut List)*, page 492.
- **Summary File** Sets options for the cut list summary file. For more information, see *Summary File (Material List Cut List)*, page 495.

Property	Value	Description	Option Switch/Attribute
ActiveList	Fixed UserDefined	Controls the format of the cut list.	ISOGEN
CutPieceID	Numeric Alphabetical	Specifies the cut piece identifier type at the appropriate positions along pipelines.	2 pos 3
CutPieceSequence	Continuous Per Drawing	Specifies whether cut piece numbers increase across drawings. If you set this option to	2 pos 1
		Continuous, the cut piece numbers increase across drawings. Otherwise, the cut piece numbers start at 1 (or A) on each drawing.	
DecimalInchOutput	True/False	Specifies the output of cut piece lengths in decimal inches for imperial units used.	2 pos 4

Property	Value	Description	Option Switch/Attribute
LengthToCL	True/False	Specifies the calculation of branch cut length. You can specify that the length be measured to the actual branch connection point or to the run centerline.	2 pos 5
OffShoreAllowance	Value in range 0-999	Controls the allowance for FFW (Field Fit Weld) or loose flanges that are offshore.	22 pos 4-6
OnShoreAllowance	Value in range 0-999	Controls the allowance for FFW (Field Fit Weld) or loose flanges that are onshore.	22 pos 1-3
ShopWeldAllowance	Value in range 0-999	Sets the shop test weld allowance.	22 pos 7-9
Visible	True/False	Turns the display of the cut piece list and cut piece identifiers on or off.	2 pos 1

Related TopicsAppendix B: Isometric Drawing Options, page 393

Fixed Layout (Material List Cut List)

Defines the fixed layout cut list format.

Property	Value	Description	Option Switch/Attribute
Content	Standard	Specifies a limited or full cut list.	2 pos 2
	Extended	Standard - Specifies the old style cut list with limited information.	
		Extended - Specifies the new style cut list with full information.	

Related Topics

• Appendix B: Isometric Drawing Options, page 393

User Defined (Material List Cut List)

Defines the user-defined cut list format.

The User Defined (Cut List) folder contains the following groups of options:

• **Column** - Specifies the columns for the user defined cut list. For more information, see *Column (Material List Cut List User Defined)*, page 494

Property	Value	Description	Option Switch/Attribute
AllowOverflowDrawings	True/False	Controls whether the cut list causes an overflow drawing.	ISOGEN
DrawingLayer	Integer in the range 1 - 50	Sets the drawing layer for the cut list text.	ISOGEN
HorizontalDirection	Left Right	Sets the horizontal direction for the cut list.	ISOGEN

Property	Value	Description	Option Switch/Attribute
HorizontalSpacing	Integer	Sets the horizontal spacing required between each column of the cut pipe list output when the table is horizontally formatted or two-dimensional.	ISOGEN
Layout	Single Vertical Single Horizontal Multiple Vertical Multiple Horizontal	Controls whether the table is horizontal, vertical, or a multiple.	ISOGEN
MaxColumns	Integer	Defines the maximum number of columns before a new row is started when using multiple tables, or the maximum number of entries for a single horizontal table.	ISOGEN
MaxRows	Integer	Defines the maximum number of rows before a new column is started when using multiple tables, or the maximum number of entries when the table is vertical.	ISOGEN
StartX	Integer	Defines the bottom left X position of the first line of the cut list.	ISOGEN
StartY	Integer	Defines the bottom left Y position of the first line of the cut list.	ISOGEN

Property	Value	Description	Option Switch/Attribute
TextFont	Font name	Specifies a font corresponding to a font entry in the .fif file.	ISOGEN
TextHeight	Real number	Sets the height of the text in the cut list.	ISOGEN
TextWeight	Real number	Sets the character thickness in millimeters.	ISOGEN
TextWidthFactor	Integer	Defines the ratio of width to height of the text in the cut list.	ISOGEN
VerticalDirection	Up Down	Sets the vertical direction for the cut list.	ISOGEN
VerticalSpacing	Integer	Sets the vertical spacing required between each row of the cut pipe list output when the table is vertically formatted or two-dimensional.	ISOGEN

• Appendix B: Isometric Drawing Options, page 393

Column (Material List Cut List User Defined)

Specifies the columns for the user defined cut list.

AttributeName	XPosition	MaxChars
Specifies a property to appear	1 01	Sets the width of the
in the column.	of the column.	column.

Related Topics

Summary File (Material List Cut List)

Sets options for the cut list summary file.

The **Summary File (Material List Cut List)** folder contains the following groups of options:

• Column - Specifies the columns for a material list summary file. For more information, see *Column (Material List Cut List Summary File)*, page 496

Property	Value	Description	Option Switch/Attribute
Append	True/False	Defines whether the file is opened in appended mode or overwrite mode.	ISOGEN
Enabled	True/False	Specifies whether the summary file is created.	ISOGEN
HeaderLines	String for the column header	Specifies a text string for the column header. Use a backslash (\) to start a new line, and pipe character () to produce a vertical break. You must ensure that the column headers match the column positions set in the Material List Definition file (MLD).	ISOGEN
		The HeaderLines width is defined by the length of the last line, terminated with the pipe character (). The length of the last line should be at least as long as the total width of data to be displayed below it. Column description text in the HeaderLines field must lie in the	
		column bounds as defined. Note	
		Do not put text in columns that are not within the specified range of column.	

Property	Value	Description	Option Switch/Attribute
Path	Path to a directory	Specifies a path to the summary file.	ISOGEN
		☑ Note	
		Do not include special characters in the summary file name.	
ShowEnclosure	True/False	Specifies whether cut list enclosures are shown.	ISOGEN
ShowHeaders	True/False	Specifies whether the header lines are shown.	ISOGEN
ShowTitles	True/False	Specifies whether the title line is shown.	ISOGEN
TitleText	String	Specifies the text for the title.	ISOGEN
UserDefined	True/False	Specifies whether the software creates the old standard material control file, or the software uses the properties of the summary file object to control the data output to the material control file.	23 pos 4

• Appendix B: Isometric Drawing Options, page 393

Column (Material List Cut List Summary File)

Specifies the columns for a material list summary file.

Attribute Name	Start	Width	Justification
Specifies a property to appear in the	Specifies the starting point of the column. The valid range is 1-	Sets the width of the column.	Sets the alignment. You can choose from left- or right-justified, or numeric.
column.	999.	Column.	right justified, of numeric.

Related Topics

Fixed Layout (Material List)

Sets options for the fixed layout material list format. This format produces a simple material list. You can use either a user-defined drawing frame or the standard ISOGEN drawing frame.

Property	Value	Description	Option Switch/ Attribute
InsertBlankLines	True/False	Sets whether the software inserts blank lines. If you set this option to True , the software inserts blank lines after single line entries in the material list.	26 pos 2
Layout	Standard Continuous	Specifies the material list layout pattern. Select Standard for the standard sectioned type with group headings and component sub-group headings. Select Continuous for a layout without headings and sub-headings.	23 pos 5
LineSpacing	Value in range 75-125	Controls the line spacing in the material list. This value is a ratio that applies to standard line spacing. For example, a value of 90 reduces the line spacing, and a value of 115 increases the line spacing.	29
MLType	Normal Special Normal with alpha pointers	Controls the type of material list. Normal means the software does not create a special type of material list. Special means every item is given a separate entry except gaskets and bolts, which are accumulated. Pulled bends get U prefixes, and welds have W or B prefixes. Normal with alpha pointers means the software does not create a special type of material list. The software uses an alpha system of material list pointers (A, B, C) instead of the default numeric system.	23 pos 2
RightSide	True/False	Sets the position of the material list. If you set this option to True , the material list appears on the right side of the drawing; otherwise, it appears on the left side.	25 pos 1

Property	Value	Description	Option Switch/ Attribute
ShowDividingLines	True/False	Specifies lines in the material list. If you set this option to True , dividing lines appear in the material list between groups, such as PIPES, FLANGES, or FITTINGS.	23 pos 5
		This option provides a way to separate the different groups in the material list when the list does not have headings and subheadings.	
		☑ Note	
		This option works only when the MaterialList.FixedLayout.Layout option is set to Continuous and the MaterialList.FixedLayout.MLType is set to Special.	
ShowItemDescriptions	True/False	Displays descriptions in the material list.	26 pos 1
TextSize	Small (2.1 mm)	Controls the material list text size.	27
	Medium (2.5 mm)		
	Large (2.8 mm)		
	XLarge (3.5 mm)		
	XXLarge (4.2 mm)		
	XXXLarge (4.9 mm)		
	User		
TextWeight	Value in range 0-9	Specifies the thickness of the text.	27 pos 5
TextWidth	Value in range 10-99 (in 1/10 millimeter)	Specifies the character width if you are using a fixed-width font.	27 pos 3, 4

Property	Value	Description	Option Switch/ Attribute
TitleblockPos	Pipeline MatList	Controls the position of the title block. Pipeline means the title block is positioned at the bottom right corner of the pipeline drawing area. MatList means the title block is positioned at the bottom of the material list.	25 pos 2
TitleblockVisible	True/False	Controls whether the title block appears on the drawing.	25 pos 2
UserTextSize	Integer in range 10-	Specifies a user-defined size in 1/10 millimeters. You should use this option only if you set TextSize to User .	27 pos 1, 2

Variable Layout (Material List)

Sets options for the variable layout material list format. This format includes options to define column headers. You can use either a user-defined drawing frame or the standard ISOGEN drawing frame.

The **Variable Layout (Material List)** folder contains the following groups of options:

• **Column** - Specifies the columns for a variable layout material list. For more information, see *Column (Material List Variable Layout)*, page 505

Property	Value	Description	Option Switch/ Attribute
DefaultDescriptionWidth	Value in range 0- 130	Specifies the default number of characters if you use the DESCRIPTION attribute. You can override this option by setting the Width property for this column. The column width for fixed-width fonts is important only for this attribute.	ISOGEN

Property	Value	Description	Option Switch/ Attribute
HeaderLines	String for the column header.	Specifies a text string for the column header. Use a backslash (\) to start a new line, and a pipe character () to produce a vertical break. You must ensure that the column headers match the column positions set in the Material List Definition file (MLD).	ISOGEN
		The HeaderLines width is defined by the length of the last line, terminated with the pipe character (). The length of the last line should be at least as long as the total width of data to be displayed below it.	
		Column description text in the HeaderLines field must lie in the column bounds as defined.	
		☑ Note	
		Do not put text in columns that are not within the specified range of column.	
InsertBlankLines	True/Fal se	Sets whether the software inserts blank lines. If you set this option to True , the software inserts blank lines after single line entries in the material list.	26
Layout	Standard Continu ous	Specifies the material list layout pattern. Select Standard for the standard sectioned type with group headings and component sub-group headings. Select Continuous for a layout without headings and sub-headings.	23 pos 5

Property	Value	Description	Option Switch/ Attribute
LineSpacing	Value in range 75-125	Controls the line spacing in the material list. This value is a ratio that applies to standard line spacing. For example, a value of 90 reduces the line spacing, and a value of 115 increases the line spacing.	29
MLType	Normal Special Normal with alpha pointers	Controls the type of material list. Normal means there is no special type of material list. Special means every item is given a separate entry except gaskets and bolts, which are accumulated. Pulled bends get U prefixes, and welds have W or B prefixes. Normal with alpha pointers means the software does not create a special type of material list. The software uses an alpha system of material list pointers (A, B, C) instead of the default numeric system.	23 pos 2
RightSide	True/Fal se	Sets the position of the material list. If you set this option to True , the material list appears on the right side of the drawing; otherwise, it appears on the left side.	25

Property	Value	Description	Option Switch/ Attribute
ShowDividingLines	True/Fal se	Specifies lines in the material list. If you set this option to True , dividing lines appear in the material list between groups, such as PIPES, FLANGES, or FITTINGS. This option provides a way to separate the different groups in the material list when the list does not have headings and subheadings. Note This option works only when the MaterialList.VariableL ayout.Layout option is set to Continuous.	23 pos 5

Property	Value	Description	Option Switch/ Attribute
TextSize	Small (2.1 mm) Medium (2.5 mm) Large (2.8 mm) XLarge (3.5 mm) XXLarg e (4.2 mm) XXXLar ge (4.9 mm)	Controls the material list text size.	27
	User	~	
TextWeight	Value in range 0-	Specifies the thickness of the text.	27 pos 5
TextWidth	Value in range 10-99 (in 1/10 millimet er)	Specifies the character width if you are using a fixed-width font.	27 pos 3, 4
UserTextSize	Integer in range 10-99	Specifies a user-defined size in 1/10 millimeters. You should use this option only if you set TextSize to User .	27 pos 1, 2

Column (Material List Variable Layout)

Specifies the columns for a variable layout material list.

Attribute Name	Start	Width	Justification
Specifies a property to appear in the column.	Specifies the starting point of the column. The valid range is 1-999.	Sets the width of the column.	Sets the alignment. You can choose from left- or right-justified, or numeric.

Related Topics

User Defined (Material List)

Sets options for the user-defined material list format. The user-defined material list offers more flexibility than the other two material list styles.

Note

• This style of material list requires a user-defined backing sheet.

The User Defined (Material List) folder contains the following groups of options:

- **Column** Specifies the columns for a user-defined material list. For more information, see *Column* (*Material List User Defined*), page 510.
- One Section Sets options for the user-defined material list format. This format displays one section in the material list. For more information, see *One Section (Material List User Defined)*, page 510.
- **Two Section** Sets options for the user-defined material list format. This format displays two sections in the material list. For more information, see *Two Section (Material List User Defined)*, page 511.
- **Three Section** Sets options for the user-defined material list format. This format displays three sections in the material list. For more information, see *Three Section (Material List User Defined)*, page 512.
- **Remarks Box** Sets options for the user-defined material list remarks box. For more information, see *Remarks Box (Material List User Defined)*, page 514.

Property	Value	Description	Option Switch/Attribute
ActiveSection	1, 2, 3	Defines the number of sections in the material list.	ISOGEN
CategoryHeadingX	True/False	Controls whether category headings are displayed.	ISOGEN
CategoryUnderline	True/False	Controls whether category headings are underlined.	ISOGEN

Property	Value	Description	Option Switch/Attribute
DefaultDescriptionWidth	Value in range 0-130	Specifies the default number of characters if you use the DESCRIPTION attribute. You can override this option by setting the Width property for this column. The column width for fixed-width fonts is important only for this attribute.	ISOGEN
DrawingLayer	Integer, range 1-50	Sets the drawing layer for material list text. For more information about drawing layers, see <i>Definitions (Drawing)</i> , page 443.	ISOGEN
GroupHeadingX	Integer (mm)	Specifies the displacement from the StartX setting for the group heading.	ISOGEN
GroupUnderline	True/False	Controls whether group headings are underlined.	ISOGEN

Property	Value	Description	Option Switch/Attribute
MLType	Normal Special Normal with alpha pointers	Controls the type of material list. Normal means the software does not create a special type of material list. Special means every item is given a separate entry except gaskets and bolts, which are accumulated. Pulled bends get U prefixes, and welds have W or B prefixes. Normal with alpha pointers means the software does not create a special type of material list. The software uses an alpha system of material list pointers (A, B, C) instead of the default numeric system.	23 pos 2
ShowCategoryHeadings	True/False	Specifies whether or not the category headings appear.	ISOGEN
ShowGroupHeadings	True/False	Specifies whether or not the group headings appear.	ISOGEN
ShowRemarks	True/False	Controls the appearance of a remarks list, as defined by the separate Remarks options. For more information about remarks, see <i>Remarks Box (Material List User Defined)</i> , page 514.	ISOGEN
StartX	Integer	Defines the bottom left X position of the first line of the material list.	ISOGEN
StartY	Integer	Defines the bottom left Y position of the first line of the material list.	ISOGEN

Property	Value	Description	Option Switch/Attribute
TextFont	Font name	Specifies a font corresponding to a font entry in the .fif file.	ISOGEN
TextHeight	Real number	Specifies the text height in millimeters. The text height applies to all sections in the material list.	ISOGEN
TextWeight	Integer	Specifies the text weight.	ISOGEN
TextWidthFactor	Integer	Defines the ratio of width to height of the text in the material list.	ISOGEN
VerticalSpacing	Real number	Specifies the vertical spacing between the lines of data. The vertical spacing applies to all sections in the material list.	ISOGEN

Column (Material List User Defined)

Specifies the columns for a user-defined material list.

Attribute Name	XPosition	MaxChars
Specifies a property to	Specifies the starting point of the	Sets the width of
appear in the column.	column. The valid range is 1-999.	the column.

Related Topics

• Appendix B: Isometric Drawing Options, page 393

One Section (Material List User Defined)

Sets options for the user-defined material list format. This format displays one section in the material list.

Property	Value	Description	Option Switch/ Attribute
Content	CONTINUOUS FAB ERECT OFFSHORE ERECT/OFFSHORE	Selects the category of data for the material list content. CONTINUOUS means ALL items appear in the material list. The default for this option is CONTINUOUS.	ISOGEN
ListDown	True/False	Specifies the plotting of the material list. If you set this option to False , the software plots data from the bottom up. The default for this option is True .	ISOGEN
MaxEntries	Positive Integer	Sets the maximum number of entries in the material list.	ISOGEN

Related Topics

Two Section (Material List User Defined)

Sets options for the user-defined material list format. This format displays two sections in the material list.

Property	Value	Description	Option Switch/ Attribute
Section1Content	FAB ERECT OFFSHORE ERECT/OFFSHORE	Selects the category of data for the material list content in Section 1. The default for this option is FAB .	ISOGEN
Section1Down	True/False	Specifies the plotting of the material list. If you set this option to False , the software plots data from the bottom up in Section 1. The default for this option is True .	ISOGEN
Section1MaxEntries	Integer	Sets the maximum number of entries in Section 1.	ISOGEN
Section1YOffset	Integer	Controls the Y offset for section 1 of the material list with reference to the StartX and StartY options under Material List > User Defined.	ISOGEN
Section2Content	FAB ERECT OFFSHORE ERECT/OFFSHORE	Selects the category of data for the material list content in Section 2. The default for this option is ERECT . This option cannot be same as Section1Content .	ISOGEN
Section2Down	True/False	Specifies the plotting of the material list. If you set this option to False , the software plots data from bottom up in Section 2. The default for this option is True .	ISOGEN

Property	Value	Description	Option Switch/ Attribute
Section2MaxEntries	Integer	Sets the maximum number of entries in Section 2.	ISOGEN
Section2YOffset	Integer	Controls the Y offset for section 2 of the material list with reference to the StartX and StartY options under Material List > User Defined.	ISOGEN

• Appendix B: Isometric Drawing Options, page 393

Three Section (Material List User Defined)

Sets options for the user-defined material list format. This format displays three sections in the material list.

Property	Value	Description	Option Switch/ Attribute
ListDown	True/False	Sets the plotting of the material list. If you set this option to False , the software plots data from the bottom up. The default for this option is True .	ISOGEN
Section1Content	FAB ERECT OFFSHORE ERECT/OFFSHORE	Selects the category of data for the material list content in Section 1. The default for this option is FAB .	ISOGEN
Section1MaxEntries	Integer	Sets the maximum number of entries in Section 1.	ISOGEN

Property	Value	Description	Option Switch/ Attribute
Section1YOffset	Integer	Controls the Y offset for section 1 of the material list with reference to the StartX and StartY options under Material List > User Defined.	ISOGEN
Section2Content	FAB ERECT OFFSHORE ERECT/OFFSHORE	Selects the category of data for the material list content in Section 2. The default for this option is ERECT . This option cannot be the same as Section1Content .	ISOGEN
Section2MaxEntries	Integer	Sets the maximum number of entries in Section 2.	ISOGEN
Section2YOffset	Integer	Controls the Y offset for section 2 of the material list with reference to the StartX and StartY options under Material List > User Defined.	ISOGEN
Section3Content	FAB ERECT OFFSHORE ERECT/OFFSHORE	Selects the category of data for the material list content in Section 3. The default for this option is OFFSHORE . This option cannot be the same as Section1Content or Section2Content .	ISOGEN
Section3MaxEntries	Integer	Sets the maximum number of entries in Section 3.	ISOGEN
Section3YOffset	Integer	Controls the Y offset for section 3 of the material list with reference to the StartX and StartY options under Material List > User Defined.	ISOGEN

• Appendix B: Isometric Drawing Options, page 393

Remarks Box (Material List User Defined)

Sets options for the user-defined material list remarks box.

Property	Value	Description	Option Switch/ Attribute
DrawingLayer	Integer in the range 1-50	Sets the drawing layer for material list text. For more information about drawing layers, see <i>Definitions (Drawing)</i> , page 443.	ISOGEN
MaxCharacters	Positive integer	Sets the maximum number of characters per line in the remarks box.	ISOGEN
MaxEntries	Positive integer	Sets the maximum number of entries in the remarks box.	ISOGEN
StartX	Positive real number in millimeters	Specifies the X-coordinate (in millimeters) of the starting position for the remarks box.	ISOGEN
StartY	Positive real number in millimeters	Specifies the Y-coordinate (in millimeters) of the starting position for the remarks box.	ISOGEN
TextHeight	Positive real number in millimeters	Specifies the height of the characters in remarks text.	ISOGEN
TextWeight	Positive real number in millimeters	Specifies the weight of the characters in remarks text.	ISOGEN
VerticalSpacing	Positive real number in millimeters	Sets the vertical spacing between the horizontal lines of remark text.	ISOGEN

Related Topics

Summary File (Material List)

Sets options for the material list summary file.

The Summary File (Material List) folder contains the following groups of options:

• **Column** - Specifies the columns for a material list summary file. For more information, see *Column (Material List Summary File)*, page 516.

Property	Value	Description	Option Switch/Attribute
Append	True/False	Defines whether the file is opened in appended mode or overwrite mode.	ISOGEN
Enabled	True/False	Specifies whether the summary file is created.	ISOGEN
HeaderLines	String for the column header	Specifies a text string for the column header. Use a backslash (\) to start a new line, and pipe character () to produce a vertical break. You must ensure that the column headers match the column positions set in the Material List Definition file (MLD). The HeaderLines width is defined by the length of the last line, terminated with the bar character . The length of the last line should be at least as long as the total width of data to be displayed below it. Column description text in the HeaderLines field must lie in the column bounds as defined. Note Note Do not put text in columns that are not within the	ISOGEN
Include Delte	True/False	specified range of column.	65 mag 1
IncludeBolts	True/raise	Specifies whether bolts are included in the summary file.	65 pos 1

Property	Value	Description	Option Switch/Attribute
Path	Path to a directory	Specifies a path to the summary file.	ISOGEN
		☑ Note	
		Do not include special characters in the summary file name.	
ShowHeaders	True/False	Specifies whether the header lines are shown.	ISOGEN
ShowTitles	True/False	Specifies whether the title line is shown.	ISOGEN
TitleText	String	Specifies text for the title.	ISOGEN
UserDefined	True/False	Specifies whether the software creates the old standard material control file, or the software uses the properties of the summary file object to control the data output to the material control file.	23 pos 4

• Appendix B: Isometric Drawing Options, page 393

Column (Material List Summary File)

Specifies the columns for a material list summary file.

Attribute Name	Start	Width	Justification
Specifies a property to appear in the column.	Specifies the starting point of the column. The valid range is 1-999.	Sets the width of the column.	Sets the alignment. You can choose from left- or right-justified, or numeric.

Related Topics

Transfers (Material List)

Moves components from one sort group to another. For example, by default a blind flange is listed in the Valves/Inline item group in the material list. Using the Transfers functionality, you can specify that blind flanges are listed in the Flanges group in the material list instead.

Note

When you transfer components on a Variable Layout or User Defined material list, you must also change the material list headings by specifying AText -307, -308, -309, -311, -312, -313, -314, -315, -339, and -375. For more information, see the AText Reference Guide, available from the **Help** > Printable Guides command in the software or from the Alias web site (http://www.alias.ltd.uk).

RecordID	NewGroup
Specifies the record ID of the component type. Valid IDs are integers in the range 0-999. For more information, see <i>Record Identification Numbers</i> , page 518.	Specifies a new group for the component type. Available sort groups include:
page 316.	PIPE (pipe)
	VALV (valves and inline items)
	FITT (fittings)
	INST (instruments)
	FLAN (flanges)
	SUPP (supports)
	GASK (gaskets)
	MISC (miscellaneous components)
	BOLT (bolts)
	WELD (welds)

Related Topics

Record Identification Numbers

Spool and In-Line Fitting Records

Spool/Fitting Type	In Leg	First Branch Leg	Second Branch Leg	Out Leg	Remarks
Bend	30	-	-	31	
Elbow	35	-	-	36	
Olet	40	41	-	42	Plus a 0 record
Tee	45	46	-	47	Plus a 0 record
Cross	50	51	52	53	Plus a 0 record
Reducer (Con/Ecc)	55	-	-	-	Plus a 0 record
Tee Reducer (Con/Ecc)	60	61	-	62	Plus a 0 record
Reducing Flange	65	-	-	-	Plus a 0 record
Tee Bend/Elbow	70	71	-	72	Plus a 0 record
Angle Valve	75	-	-	76	Plus a 0 record
3 Way Valve	80	81	-	82	Plus a 0 record
4 Way Valve	85	86	87	88	Plus a 0 record
Instrument	90	91	92	93	A straight through type instrument has only 90 and 93 record ID numbers. Instrument dials have only a 90 record.
Pcom (Misc. Pipe Component)	95	-	-	96	
Pipe (Tube)	100	-	-	-	
Fixed Length Pipe	101	-	-	-	
Pipe Block (Fixed Length)	102	-	-	-	
Pipe Block (Variable Length)	103	-	-	-	
Flange	105	-	-	-	
Lap Joint Stub End	106	-	-	-	

Spool/Fitting Type	In Leg	First Branch Leg	Second Branch Leg	Out Leg	Remarks
Blank Flange (Blind)	107	-	-	-	
Gasket	110	-	-	-	
Bolt	115	-	-	-	
Weld	120	-	-	-	
Cap	125	-	-	-	
Coupling	126	-	-	-	
Union	127	-	-	-	
Valve	130	-	-	-	
Trap	132	-	-	-	
Vent	134	-	-	-	
Filter	136	-	-	-	
User Positioned Comment	149	-	-	-	Used to define position
Pipe Hanger/Support	150	-	-	-	
	180 to 199 inclusive	-	-	-	Reserved for internal ISODAT processing.
Bore Record	0	-	-	-	
End of File Marker	999	-	-	-	

Related TopicsTransfers (Material List), page 517

Weld List Options: An Overview

Sets options for the ISOGEN weld list. For options that control weld representation on the drawing, see *Welds (Drawing)*, page 438.

Note

• The weld list is the report that appears on the drawing, while the weld list summary file is the weld data formatted in a text file.

The **Weld List** folder contains the following groups of options.

- **Fixed Layout** Defines the fixed layout weld list. For more information, see *Fixed Layout (Weld List)*, page 521.
- Variable Layout Defines the alternative variable layout weld list. The fixed layout list is a fixed-format list that you cannot modify. The variable layout list is a column-based list. For more information, see *Variable Layout (Weld List)*, page 522.
- **User Defined** Defines the alternative user defined weld list. The software prints this type of list into a drawing frame that contains a pre-existing table with header texts. The user defined list is a collection of start positions expressed in millimeters. For more information, see *User Defined (Weld List)*, page 523.
- **Summary File** Defines the titles section of a user-defined weld summary file and contains a series of column definitions. You can suppress the display of title and column header information to produce a more readily machine-readable summary file. For more information, see *Summary File* (*Weld List*), page 525.

Property	Value	Description	Option Switch/Attribute
ActiveList	Fixed Variable UserDefined	Controls the format of the weld list.	53 pos 2
Visible	True/False	Turns on or off the display of the weld or operations summary on the drawing.	53 pos 2

Related Topics

Fixed Layout (Weld List)

Sets options for the fixed layout weld list format. This format produces a simple weld list. You can use either a user-defined drawing frame or the standard ISOGEN drawing frame.

Property	Value	Description	Option Switch/Attribute
ShowOperationsBox	True/False	Controls whether the weld operations box is shown or not.	53 pos 2
		True - The weld operations box is shown. This operations box replaces the original weld list if the WeldList.Visible option is set to True.	
		False - The weld operations box is not shown. The original weld list is plotted if the WeldList.Visible option is set to True.	

Related Topics

Variable Layout (Weld List)

Sets options for the variable layout weld list format. This format includes options to define column headers. You can use either a user-defined drawing frame or the standard ISOGEN drawing frame.

The Variable Layout (Weld List) folder contains the following groups of options:

• **Column** - Specifies the columns for a variable layout weld list. For more information, see *Column* (*Weld List Variable Layout*), page 522.

Property	Value	Description	Option Switch/Attribute
HeaderLines	String for the column header.	Controls the column header text in the material list. Notes Check that the column headers match the column positions when you use this option. Type \ to start a new line in the column header. Type for a vertical break.	ISOGEN

Related Topics

• Appendix B: Isometric Drawing Options, page 393

Column (Weld List Variable Layout)

Specifies the columns for a variable layout weld list.

Attribute Name	Start	Width	Justification
Specifies a property to appear in the column.	Specifies the starting point of the column. The valid range is 1-999.	Sets the width of the column. The width must be zero or greater.	Sets the alignment. You can choose from left- or right-justified, or numeric.

Related Topics

User Defined (Weld List)

Sets options for the user-defined weld list format. The user-defined weld list offers more flexibility than the other two weld list styles.

The **User Defined (Weld List)** folder contains the following groups of options:

• **Column** - Specifies the columns for a user-defined weld list. For more information, see *Column (Weld List User Defined)*, page 524.

Property	Value	Description	Option Switch/Attribute
DrawingLayer	Integer, range 1-50	Sets the drawing layer for weld list text. For more information about drawing layers, see <i>Definitions (Drawing)</i> , page 443.	ISOGEN
ListDown	True/False	Sets the plotting of the weld list. If you set this option to False , the software plots data from the bottom up. The default for this option is True .	ISOGEN
MaxEntries	Integer	Sets the maximum number of entries for the weld list.	ISOGEN
StartX	Integer	Defines the bottom left X position of the first line of the weld list.	ISOGEN
StartY	Integer	Defines the bottom left Y position of the first line of the weld list.	ISOGEN
TextHeight	Real number	Specifies the text height in millimeters.	ISOGEN
TextWeight	Integer	Specifies the character thickness in millimeters.	ISOGEN
VerticalSpacing	Real number	Specifies the vertical spacing between each horizontal line.	ISOGEN

Related Topics

Column (Weld List User Defined)

Specifies the columns for a user-defined weld list.

Attribute Name	XPosition	MaxChars
Specifies a property to appear in the column.	Sets the width of the column. The width must be zero or greater.	Specifies the starting point of the column. The valid range is 1-999.

Related Topics

Summary File (Weld List)

Sets options for the titles and column headers in a user-defined weld summary file. You can suppress the display of title and column header information in order to produce a more readily machine-readable summary file.

The **Summary File (Weld List)** folder contains the following groups of options:

• Column - Specifies the columns for a weld list summary file. For more information, see *Column (Weld List Summary File)*, page 527

Note

• In order to specify a summary file, you must also enter data in the **Weld List.Summary File.Column** grid.

Property	Value	Description	Option Switch/Attribute
Append	True/False	Specifies an append file for welds. If you set this option to True , the software creates an append file. The current pipeline is added to an accumulation file. If you set this option to False , the software creates a new file.	ISOGEN
Delimiter	User- defined value	Specifies a delimiter to use for the header lines that appear at the top of a Weld List Summary file.	ISOGEN
Enabled	True/False	Controls whether the summary file is created. If you want to set this option to True , you first must specify the columns under Weld List.Summary.Column .	ISOGEN
FileFormat	Fixed Delimited	Sets the format of the site weld file.	ISOGEN

Property	Value	Description	Option Switch/Attribute
HeaderLines	String containing column header lines	Controls the column header text in the weld list. Notes	ISOGEN
		Check that the column headers match the column positions when you use this option.	
		Type \ to start a new line in the column header.	
		Type for a vertical break.	
Path	Path to a file	Browses your computer or network for a file name.	ISOGEN
		☑ Note	
		Do not include special characters in the summary file name.	
ShowHeaders	True/False	Controls whether column headers appear.	ISOGEN
ShowTitles	True/False	Controls whether titles appear in the weld summary file. This option overrides the setting in TitleText .	ISOGEN
TitleText	String containing title text	Constructs the weld summary file title block.	ISOGEN

Column (Weld List Summary File)

Specifies the columns for a weld list summary file.

Attribute Name	Start	Width	Justification
Specifies a property to appear in the column.	Specifies the starting point of the column. The valid range is 1-999.	Sets the width of the column.	Sets the alignment. You can choose from left- or right-justified, or numeric.

Related Topics

Neutral File Options: An Overview

Sets options for the material take-off neutral file. This file has a .b extension and is saved in the output folder.

Note

• To exclude items from the neutral file, you can set properties on the items in the model. On the **Occurrence** tab of the **Properties** dialog box for an item, set the **Reporting Requirements** property to **Not to be reported**. Set the **Reporting Type** property to **<undefined value>**.

The **Neutral File** folder contains the following groups of options:

- User Attributes Specifies the user attributes for the material take-off neutral file. For more information, see *User Attributes (Neutral File)*, page 530.
- **Part Data** Specifies the part data for the material take-off neutral file. For more information, see *Part Data (Neutral File)*, page 531.
- **Weld Data** Specifies the weld data for the material take-off neutral file. For more information, see *Weld Data* (*Neutral File*), page 532.
- **Bolt Data** Specifies the bolt data for the material take-off neutral file. For more information, see *Bolt Data (Neutral File)*, page 533.
- **Gasket Data** Specifies the gasket data for the material take-off neutral file. For more information, see *Gasket Data (Neutral File)*, page 534.

Property	Value	Description	Option Switch/ Attribute
Enabled	True/False	Produces the Intergraph MTO file.	SmartPlant 3D
GeneratedBySheet	True/False	Splits the MTO neutral file by sheet.	SmartPlant 3D
GroupBPIP	True/False	Specifies whether pipe bends (BPIP) are treated as pipe in the neutral file rather than being listed separate in the .b file. When set to False , the .b file contains separate entries for pipe bends and pipe. If set to True , you only get one entry for pipe.	SmartPlant 3D

Property	Value	Description	Option Switch/ Attribute
GroupPerRun	True/False	Specifies whether a single *.b file is supported for each ISO extraction. If the ISO line count is greater than 1, Enabled is set to True , and GroupPerRun is set to True , the group neutral b file is created using the first line's name. For example, "lineName"-grp.b will contain all the data of each ISO line in the process.	SmartPlant 3D
RecordLength	256 1024	Uses standard neutral file record length of 256 or expanded length 1024.	SmartPlant 3D
Sort	True/False	Sorts the records returned on the MTO neutral file.	SmartPlant 3D

User Attributes (Neutral File)

Specifies the user attributes for the material take-off neutral file.

Name	LabelName
Specifies a name corresponding to the attribute.	Specifies the catalog label for the attribute. You can click the browse button to navigate the Catalog hierarchy.

Related Topics

Part Data (Neutral File)

Specifies the part information for the material take-off neutral file.

The **Part Data** (**Neutral File**) folder contains the following groups of options:

• Column - Specifies the columns for part data in the MTO neutral file. For more information, see *Column (Neutral File Part Data)*, page 531.

Property	Value	Description	Option Switch/Attribute
Enabled	True/False	Produces part data for the MTO file.	SmartPlant 3D

Related Topics

• Appendix B: Isometric Drawing Options, page 393

Column (Neutral File Part Data)

Specifies the columns for part data in the MTO neutral file.

Attribute	Start	Width
Specifies a property to appear in the column.	Specifies the starting point of the column.	Sets the width of the column.

Related Topics

Weld Data (Neutral File)

Specifies the weld information for the material take-off neutral file.

The Weld Data (Neutral File) folder contains the following groups of options:

• Column - Specifies the columns for weld data in the MTO neutral file. For more information, see *Column (Neutral File Weld Data)*, page 532.

Property	Value	Description	Option Switch/Attribute
Enabled	True/False	Produces weld data for the MTO file.	SmartPlant 3D

Related Topics

• Appendix B: Isometric Drawing Options, page 393

Column (Neutral File Weld Data)

Specifies the columns for weld data in the MTO neutral file.

Attribute	Start	Width
Specifies a property to appear in the column.	Specifies the starting point of the column.	Sets the width of the column.

Related Topics

Bolt Data (Neutral File)

Specifies the bolt information for the material take-off neutral file.

The **Bolt Data** (Neutral File) folder contains the following groups of options:

• Column - Specifies the columns for bolt data in the MTO neutral file. For more information, see *Column (Neutral File Bolt Data)*, page 533.

Property	Value	Description	Option Switch/Attribute
Enabled	True/False	Produces bolt data for the MTO file.	SmartPlant 3D

Related Topics

• Appendix B: Isometric Drawing Options, page 393

Column (Neutral File Bolt Data)

Specifies the columns for bolt data in the MTO neutral file.

Attribute	Start	Width
Specifies a property to appear in the column.	Specifies the starting point of the column.	Sets the width of the column.

Related Topics

Gasket Data (Neutral File)

Specifies the gasket information for the material take-off neutral file.

The Gasket Data (Neutral File) folder contains the following groups of options:

• Column - Specifies the columns for gasket data in the MTO neutral file. For more information, see *Column (Neutral File Gasket Data)*, page 534.

Property	Value	Description	Option Switch/Attribute
Enabled	True/False	Produces gasket data for the MTO file.	SmartPlant 3D

Related Topics

• Appendix B: Isometric Drawing Options, page 393

Column (Neutral File Gasket Data)

Specifies the columns for gasket data in the MTO neutral file.

Attribute	Start	Width
Specifies a property to appear in the column.	Specifies the starting point of the column.	Sets the width of the column.

Related Topics

Labels Options: An Overview

Sets options for labels on the isometric drawing. You must have a Reports database and Schema in order to use labels on isometric drawings.

The **Labels** folder contains the following groups of options:

- **Component Note** Generates labels for specific components. For more information, see *Component Note (Labels)*, page 536.
- **Material List** Generates labels for items in the material list. For more information, see *Material List (Labels)*, page 537.
- **Weld List** Generates labels for items in the weld list. For more information, see *Weld List (Labels)*, page 538.
- **Drawing Frame** Generates labels with drawing frame text. For more information, see *Drawing Frame (Labels)*, page 539.
- **End Connection** Generates labels for end connections. For more information, see *End Connection (Labels)*, page 540.
- **Misc-Spec** Generates labels for piperun properties. For more information, see *MiscSpec* (*Labels*), page 541.
- **PipeLineListBox** Generates labels to use for the PipeLineListBox. For more information, see *PipeLineListBox* (*Labels*), page 542.

Property	Value	Description	Option Switch/Attribute
EnableLabels	True/False	Enables reporting labels.	SmartPlant 3D
MiscSpec	True/False	Enables the use of miscellaneous specifications.	SmartPlant 3D

Related Topics

Component Note (Labels)

Generates notes that point to specific components on the isometric drawing. The report label definition determines the different types of notes.

LabelAttribute2	LabelName	MessageEnclosure
Selects a component type, such as PIPE, VALVE, or PIPE- SUPPORT.	Provides the name of the label. You click the Browse button to open a dialog box and select a label definition from the catalog. Commonly, you will choose a label from the Iso category of labels.	Provides the type of enclosure for the label.

Related Topics

Material List (Labels)

Generates notes that correspond to items in the material list.

LabelAttribute	LabelName
Selects an	Provides the name of the label. Click the browse button to open a
attribute.	dialog box and select a label definition from the catalog.
	MATERIAL-USER0 through MATERIAL-USER9 are user-
	defined attributes reported in the material list.

Related Topics

Weld List (Labels)

Generates notes that correspond to items in the weld list.

LabelAttribute	LabelName
Selects an attribute.	Provides the name of the label. Click the ellipsis button to open the Catalog Labels dialog box and select a label to use. WELD-ATTRIBUTE1 through WELD-ATTRIBUTE10 are user-defined weld properties reported in the weld box or weld summary report.

Related Topics

Drawing Frame (Labels)

Associates drawing frame formatted text with a report label definition. You can use this option folder to specify labels for the drawing frame. For more information, see *Attribute Mapping: An Overview*, page 549.

LabelAttribute	LabelName	OidType
Selects an attribute.	Provides the name of the label. Click the ellipsis button to open the Catalog Labels dialog box and select a label to use.	Specifies that an Oid type can be associated with the drawing frame.

Related Topics

End Connection (Labels)

Associates a drawing end connection note with a catalog report label template.

LabelAttribute3	LabelName	MessageEnclosure
Select an attribute for the label.	Provides the name of the label. Click the browse button to open a dialog box and select a label definition from the catalog. Commonly, you will choose a label from the Iso category of labels.	Provides the type of enclosure for the label.

Related Topics

MiscSpec (Labels)

Generates notes that correspond to items in a miscellaneous specification.

LabelAttribute	LabelName
Selects an attribute.	Provides the name of the label. Click the browse button to open a dialog box and select a label definition from the catalog. MISC-SPEC1 through MISC-SPEC5 are user-defined piperun properties used to call out attribute breaks in the isometric drawing.

Related Topics

PipeLineListBox (Labels)

Generates labels to use as part of the PipeLineListBox.

LabelAttribute	LabelName
Selects an attribute.	Provides the name of the label. Click the browse button to open a dialog box and select a label definition from the catalog. The defined values show up in DrawingFrame.PipeLineList.Columns so you can give the
	column a name and the defined attributes that map to a label. For more information, see <i>Column (Drawing Frame Pipeline List)</i> , page 450.

Related Topics

Symbol Mapping (SymbolMAP): An Overview

Defines the symbol mapping between the software and ISOGEN, which is the third-party software used to create isometric drawings. The symbol mapping is defined for each isometric style.

The symbol map must contain all matching data for objects in the model. If the software fails to find a matching symbol name for each object during the extraction process, a message appears. Your system administrator then must update the symbol map accordingly.

The content in this option category is similar to the PDS-to-ISOGEN map in PDS (Plant Design System).

The **SymbolMAP** folder also provides supplement definitions. For more information, see *Supplement (SymbolMAP)*, page 546.

SP3DPartClass	EndPrepCode	IsoGenSkey	ComponentClass
Specifies a part class name for the piping component. These part class names are sheet names in the Piping.xls workbook.	Specifies the codelist number for the end preparation of the piping component. For more details about the End Preparation codelist, see the AllCodeLists.xls workbook.	Specifies the ISOGEN symbol key for the piping component. Most components are mapped to an SKEY using an end preparation of 0. You can specify a wildcard for the end type. For example, when ** is specified for the end type, the software reads the End Prep from the part and assigns the correct ISOGEN end type. Note An important exception is the nipple. You must map two different SKEYs and associated component identifiers based on the end preparation of the nipple. In other words, the SKEY mapping for nipples always requires end-prep information. For more information, see Nipple Symbol Mapping, page 547.	Specifies the Piping Component File (PCF) identification text for the piping component. This ID must be a valid ISOGEN Component Type Identifier as described in the ISOGEN documentation, which is accessible from the Help > Printable Guides command.

Note

• In the current version of the software, any part class that represents a spectacle blind, slip ring, paddle spacer, or other part that needs to have a primary direction output on it must be mapped to one of the following SKEYs: SP, SR, or SB. Likewise, it should be mapped to the component class MISC-COMPONENT.

Because these three SKEYs are user-definable, you can change the graphics of the SKEYs if necessary, but the names of the SKEYs cannot be changed if the direction output is needed.

Related Topics

- Appendix B: Isometric Drawing Options, page 393
- Appendix D: Symbols and Symbol Keys, page 555

Supplement (SymbolMAP)

Defines supplement symbol mapping for end preparation codes.

EndPrep1	EndPrepCode	EndPrepMap	Description
Provides a user reference for the symbol mapping.	Specifies a numerical value from the EndPreparation 3D codelist.	Specifies the ISOGEN allowable end condition code. For more information, see the <i>Symbol Keys Reference Guide</i> available from Help > Printable Guides .	Provides a description of the end preparation mapping.

Related Topics

- Appendix B: Isometric Drawing Options, page 393
- Appendix D: Symbols and Symbol Keys, page 555
- Supplement (SymbolMAP), page 546
- Symbol Mapping (SymbolMAP): An Overview, page 543

Nipple Symbol Mapping

You can map most components using an end preparation of 0.

However, a notable exception is the nipple. For nipples, you must use two different SKEYs and PCF component identifiers, based on the end preparation. The SKEY mapping for nipples always requires end-prep information.

The table below shows an example of SKEY mapping for the nipple.

CodeList	PartClassName	SKEY	PCFComponentID
301	Nipple	FPPL	PIPE-FIXED
331	Nipple	NRSC	COUPLING
391	Nipple	FPPL	PIPE-FIXED
301	NIP	FPPL	PIPE-FIXED
331	NIP	NRSC	COUPLING
391	NIP	FPPL	PIPE-FIXED

Related Topics

Alternative Text Options: An Overview

Sets options for substitute text on the isometric drawing.

AText is an abbreviation for alternative text, an ISOGEN feature that allows you to change or remove any text on the isometric drawing.

AText operates by assigning a unique identification number to each standard text string. The software refers to this number whenever the text string that the number represents needs to change. By definition, a standard text string can be a single character, a single word, or a group of words. Furthermore, some AText is set, by default, to an all-blank word.

AText allows you to substitute your own text terminology or language in place of the standard ISOGEN words on the isometric drawing. To change alternative text, you do not have to replace all of the standard AText; you can change only one word, if needed.

For more information about AText, see the Alias document titled *AText Reference Guide*, accessible from the **Help > Printable Guides** command.

Important

 Although the AText feature has a considerable degree of built-in flexibility, you must exercise a certain amount of care when defining your own words, particularly with respect to word lengths. As a general rule, newly defined words or word strings should be about equal in length or shorter than the text that is being replaced.

ISOGENTextID	AlternateText
Provides a value corresponding to the ISOGEN text to replace. For a listing of these values, see the Alias document titled <i>AText Reference Guide</i> , accessible from the Help > Printable Guides command.	Specifies the text you want to appear on the drawing.

Notes

- You can click any value in the **ISOGENTextID** dropdown list. Do not type a value manually because it may not be supported.
- In the **AlternateText** column, you do not need to type single quotation marks around the text string. You can just type the text.
- You can use AText to control the output of specification break notes. If AText is set to blank, the note is disabled.

Related Topics

Attribute Mapping: An Overview

Maps ISOGEN attributes to the software attributes.

HostAttribute	IsogenAttribute
Specifies a text string for the attribute. You select this text string in the Attributes folder under Drawing Frame .	Selects an ISOGEN attribute.

You can customize the mapping of ISOGEN Attributes **ATTRIBUTE11** through **ATTRIBUTE99**. **ATTRIBUTE1** through **ATTRIBUTE10** are reserved by Intergraph. We recommend that you start adding your own attributes from **ATTRIBUTE21**. The Intergraph reserved attributes are defined as follows:

Attribute Name	Definition
ATTRIBUTE1	Maximum Working Pressure
ATTRIBUTE2	Maximum Testing Temperature
ATTRIBUTE3	Maximum Testing Pressure
ATTRIBUTE4	Extracted By
ATTRIBUTE5	Checked By
ATTRIBUTE6	Approved By
ATTRIBUTE7	Parent Piping System
ATTRIBUTE8	Fluid Code
ATTRIBUTE9	Extraction Date
ATTRIBUTE10	Reserved for Future Use

Related Topics

Appendix B: Isometric Drav	ving Options		

Appendix C: Personal ISOGEN Return Values

If an isometric extraction fails, ISOGEN returns an error number which can be referenced from an error message or the extraction error log within the software. Descriptions for the ISOGEN error message numbers are shown in the table below.

Error Value	Description
-1001	Failed to create the specified Pre/Post Processor.
-1002	Pre/Post Processor has crashed.
-1003	Pre/Post processor has timed out and the external process has been killed.
-1004	Pre/Post processor has timed out but the external process cannot be terminated (and is therefore still running).
-1	Error in Isogen data. Check message file.
-2	Isogen dll failed to load (possible installation problem, missing dependent file, etc.) or crash occurred during execution, which has been trapped by the error handler.
-3	Isogen thread filed to finish possible execution loop which has been terminated by Personal Isogen.
0	Iso created successfully.
1	Personal ISOGEN cannot find the isometric directory specified.
2	Invalid Style for this project. The specified Style cannot be found under the specified project.
3	Specified Project not found. The specified project cannot be found.
4	IDFGEN handshake failed. The handshake passed to the <i>idfdll.dll</i> has not been verified.
	☑ Note
	A number of other "unexpected" failures have been found to generate this error.
5	Could not access specified PCF in TEMFILES.
6	ISOGEN DLL failed to establish current directory.
7	ISOGEN DLL failed to change directory.
8	The handshake passed to the pisodll.dll has not been verified.
9	Personal ISOGEN failed to produce a valid drawing. View message file for details as there are many possible causes.
10	ISOGEN DLL failed to restore working directory.
11	IDFGEN DLL failed to establish current directory.

Error Value	Description
12	IDFGEN DLL failed to change directory.
13	IDFGEN did not complete successfully. View message file for details. One common cause is that the components in the PCF do not form a connected system.
14	IDFGEN DLL failed to restore working directory.
15	IDFGEN could not delete IPISOGEN\PROGRAMS\FOR036.DAT from a previous run.
16	IDFGEN was unable to create ISOGEN.IDX file. Error opening <i>isogen.idx</i> . Error closing <i>isogen.idx</i> .
17	IDFGEN was unable to copy ISOGEN.FLS to the PROGRAMS directory. The following errors may have occurred:
	Error allocating memory to store a line.
	Zero line length encountered.
	Error opening source <i>isogen.fls</i> (in specified project\isotype).
	Error opening target <i>isogen.fls</i> (in pisogen\programs).
	Error closing source <i>isogen.fls</i> (in specified project\isotype).
	Error closing target <i>isogen.fls</i> (in pisogen\programs).
18	No description available from Alias Personal ISOGEN.
19	Personal ISOGEN handshake failed. The handshake passed to the <i>pisogen.dll has not been verified</i> .
20	Unknown Error in Personal ISOGEN. An unexpected path through the program has occurred.
21	PISOGEN DLL could not find Current Working Directory. The current directory cannot be identified.
22	PISOGEN DLL could not Change Directory to \PISOGEN\PROGRAMS.
23	ISOGEN failed to write banner to message file. (Probably cannot find message file path). The following errors may have occurred:
	Error opening isogen.fls.
	Unable to find MESSAGE line in isogen.fls.
	Error closing isogen.fls.
	Error opening message file.
	Error closing message file.
24	PISOGEN DLL has been unable to restore current working directory.

Error Value	Description
26	The following errors may have occurred:
	Error opening <i>i-gen.fls</i> for reading.
	Error opening isogen.fls for writing.
	Error opening options file specified in <i>i-gen.fls</i> .
	Error closing options file specified in <i>i-gen.fls</i> .
	Error opening <i>i-gen.opl</i> for writing.
	Error closing <i>i-gen.opl</i> .
	Error closing <i>i-gen.fls</i> .
	Error closing isogen.fls.
26	Problem with the specified isometric root directory.
27	Problem setting the current isometric root directory.
28	Invalid file extension.
29	Not applicable when running Isogen.
30	Warning occurred creating a POD (intermediate) file.
31	Error occurred creating a POD (intermediate) file. Error occurred during "pass 1" when the input IDF file is preprocessed.
32	Error occurred creating a POD (intermediate) file. Error occurred during "pass 2" when the POD file is being created from Isogen data.
33	Failed to load PODGRAPHICS DLL - used in the creation of graphics output files such as DWG and IGR.
34	PODGRAPHICS process failed - IE. a graphics specific process has failed - for example, user requested IGR output but SmartSketch is not installed.
35-40	Not used at this time.
41	Cannot get path to system temp directory.
42	System temp directory does not exist.
43	Unable to set bore units to those specified in the input file.
44	Unable to perform macro substitution in style ISOGEN.FLS.
45	Unable to restore original style ISOGEN.FLS.
46-49	Not used at this time.
50	Project disk is full - unable to create new output files.
51	File containing temp directory is full - unable to process file.
52-999	Not used at this time.
1000	Expected files missing in Style - E.G. no FLS, no options file.

Error Value	Description
1001	Requested units combination is inconsistent - probably means a combination of metric bore and imperial coordinates.
1002	Invalid drawing format requested.
1003	Problem deleting an ISOGEN message file.
1004	Line in the options file has invalid format.
1005	Too many switches in the options file.
1006	Too few switches in the options file.
1007	MicroStation design file output requested but MS_EXE environment variable is not defined.
1008	MicroStation design file output requested but MicroStation not installed.
1009	MS_EXE environment variable does not end with a \.
1010	ISOGEN.FLS missing in Style.
1011	Options file missing specified in ISOGEN.FLS is missing.

The following warning messages may also appear:

Error Value	Description
1999	Isogen has returned a FAIL for one or more drawings in the set. This means it has been unable to layout the drawing successfully.
2001	Inconsistent units combination (imperial coordinates and metric bores).
2022	IDFGEN has detected disconnected pipeline. Drawings have still been created that will indicate the cause of the disconnection.

Related Topics

- Edit Options Command, page 212
- Isometric Style Options Browser Dialog Box, page 212

Appendix D: Symbols and Symbol Keys

The Drawings and Reports task delivers a symbol library containing ISOGEN symbols. You can use the Alias symbol editor or the PDS Isometric Symbol Editor to modify the symbols.

Each symbol is associated with a unique code called a symbol key (SKEY). SKEYs contain 2-4 letters; the first two letters define the type of component, and the last two letters define the end type such as flanged, butt welded, or screwed. You can specify a wildcard for the end type. For example, when ** is specified for the end type, the software reads the End Prep from the part and assigns the correct ISOGEN end type.

You can map the symbol keys using the **Edit Options** command, which displays the **Isometric Style Options Browser**, the tool used to control all the options related to the appearance and information content of the various styles of isometric drawings.

For more information about symbols, see the Alias document titled *ISOGEN SKEY Definitions*, accessible from the **Help > Printable Guides** command.

Notes

- Before you modify a symbol, review the dimensions and connect points. For more information about the position of SKEY connection points, see *ISOGEN® SKEY Dimensions*, page 557.
- The @ character in the symbol keys can be replaced with an integer value in the range 1 to 9, inclusive, to denote the number of segments. Currently, regardless of the value assigned to @, the software draws the symbol per the SKEY plotted isometric shape.
- The + character in the symbol keys can be replaced with an integer value in the range 1 to 9, inclusive, to denote the bend radius.
- You can map most components using an end preparation of 0. However, a notable exception is the nipple. For nipples, you must use two different SKEYs and Piping Component File (PCF) component identifiers, based on the end preparation. In other words, the SKEY mapping for nipples always requires end-prep information.
- If, during drawing extraction, the software encounters a component that lacks SKEY mapping data, the extraction continues as long as the **Drawing.Controls.NoSymbolMapOk** option is set to **True**. The resulting drawing displays the component with an empty gap, along with an error message that points to the gap. In addition, the message file contains an error message.

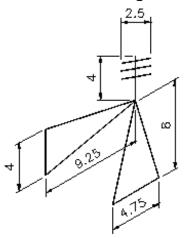
Related Topics

- *Caps (SKEYs)*, page 602
- Couplings (SKEYs), page 603

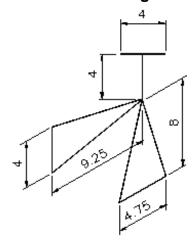
- Crosses (SKEYs), page 604
- Elbows and Bends (SKEYs), page 605
- End Prep Connections, page 607
- Fixed Length Pipes (SKEYs), page 608
- Flanges (SKEYs), page 609
- Inline Filters (SKEYs), page 610
- *Instruments (SKEYs)*, page 611
- *LJSE Type Flanges (SKEYs)*, page 614
- Miscellaneous Items (SKEYs), page 615
- Miscellaneous Pipe Components (SKEYs), page 616
- Olets (SKEYs), page 617
- Operators (SKEYs), page 618
- Other End Connections (SKEYs), page 620
- Reducers (SKEYs), page 621
- Tees (SKEYs), page 623
- Traps (SKEYs), page 624
- Valves (SKEYs), page 625
- *Vents (SKEYs)*, page 627
- Welds (SKEYs), page 628

ISOGEN® SKEY Dimensions

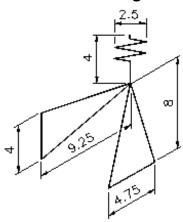
ARFL: Valve - Angle Relief/Vent



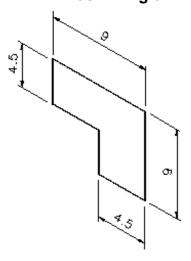
AVFL: Valve - Angle



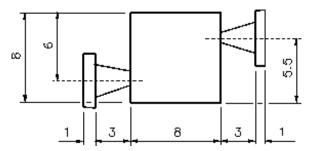
AXFL: Valve - Angle Pressure Reducing



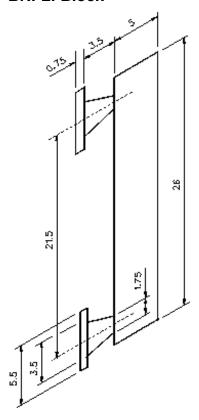
BAFL: Block - Angle



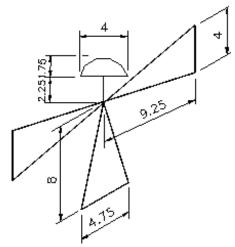
BOFL: Block - Offset



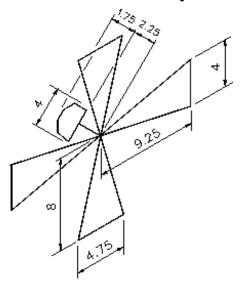
BRFL: Block



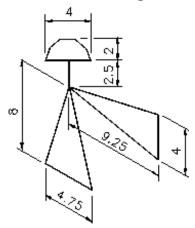
C3FL: Valve - Three-Way Control



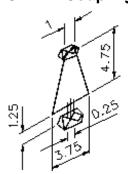
C4FL: Valve - Four-Way Control



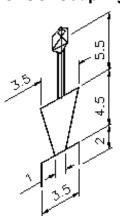
CAFL: Valve - Angle Control



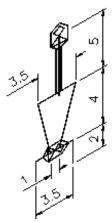
CEBW: Coupling - Elbolet (Butt Weld)



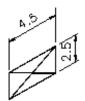
CESC: Coupling - Elbolet (Screwed)



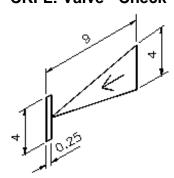
CESW: Coupling - Elbolet (Socket Weld)



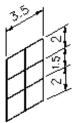
CHFL: Hose Coupling



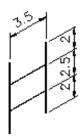
CKFL: Valve - Check



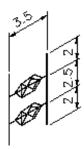
COCP: Coupling - Compression



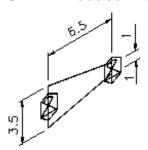
COSC: Coupling - Screwed



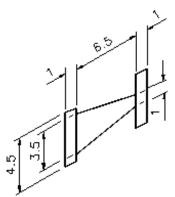
COSW: Coupling - Socket Weld



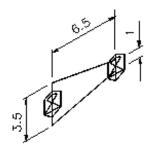
CPBW: Reducer - Concentric Fabricated from Plate



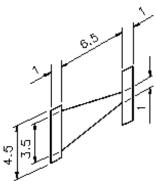
CPFL: Reducer - Concentric Fabricated from Plate (Flanged)



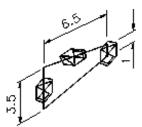
CSBW: Reducer - Concentric Swaged From Plate



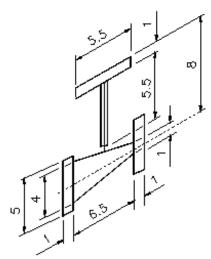
CSFL: Reducer - Concentric Swaged From Plate (Flanged)



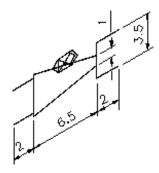
CTBW: Reducer - Concentric with a Connection (Butt Weld)



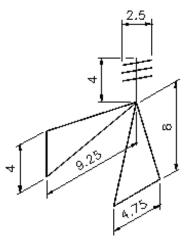
CTFL: Reducer - Concentric with a Connection (Flanged)



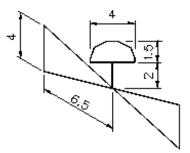
CTSC: Reducer - Concentric with a Connection (Screwed)



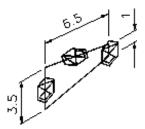
CTSW: Concentric with a Connection (Socket Weld)



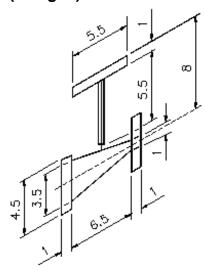
CVFL: Valve - Control



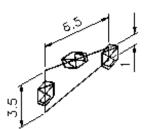
CXBW: Reducer - Concentric with a Connection Swaged from Pipe



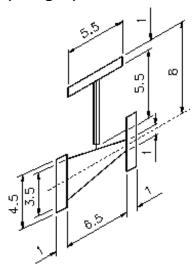
CXFL: Reducer - Concentric with a Connection Swaged from Pipe (Flanged)



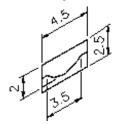
CZBW: Reducer - Concentric with a Connection Fabricated from Plate



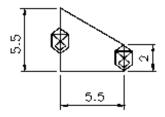
CZFL: Reducer - Concentric with a Connection Fabricated from Plate (Flanged)



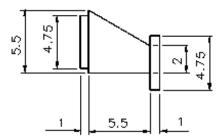
DR: Rupture Disk



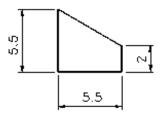
EPBW: Reducer - Eccentric Fabricated from Plate



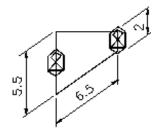
EPFL: Reducer - Eccentric Fabricated from Plate (Flanged)



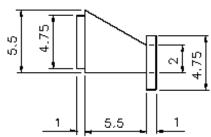
EPSW: Reducer - Eccentric Fabricated from Plate (Socket Weld)



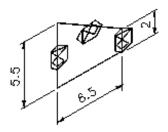
ESBW: Reducer - Eccentric Swaged from Pipe



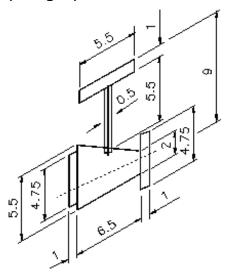
ESFL: Reducer - Eccentric Swaged from Pipe (Flanged)



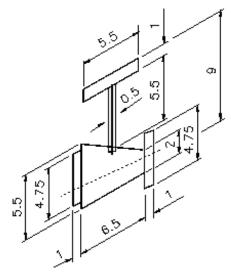
EXBW: Reducer - Eccentric with a Connection Swaged from Pipe



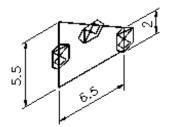
EXFL: Reducer - Eccentric with a Connection Swaged from Pipe (Flanged)



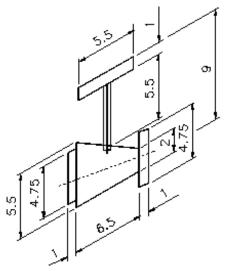
EXFL: Expansion Bellows



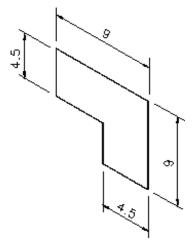
EZBW: Reducer - Eccentric with a Connection Fabricated from Plate



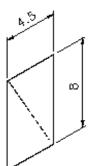
EZFL: Reducer - Eccentric with a Connection Fabricated from Plate (Flanged)



FAFL: Filter/Strainer - Angle



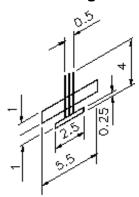
FIFL: Filter/Strainer - Straight Through



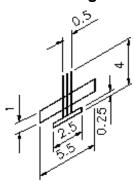
FLBL: Flange - Blind



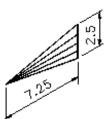
FLFL: Flange - Flared/Loose Backing



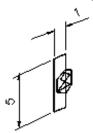
FLLB: Flange - Backing



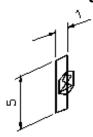
FLOW: Flow Arrow



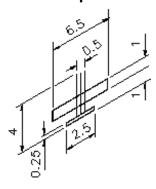
FLRC: Flange - Reducing Concentric



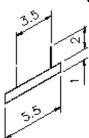
FLRE: Flange - Reducing Eccentric



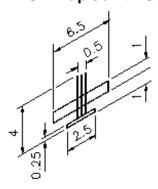
FLRG: Lap Joint - Ring Loose



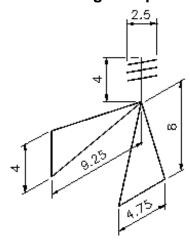
FLSC: Flange - Screwed



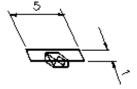
FLSE: Lap Joint - Stub End Loose



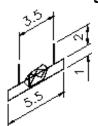
FLSJ: Flange - Slip On with J-type Weld



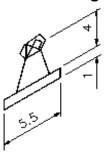
FLSO: Flange - Slip On



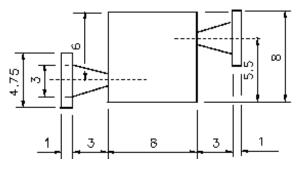
FLSW: Flange - Socket Weld



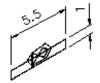
FLWN: Flange - Weld Neck



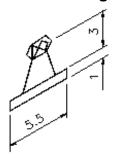
FOFL: Filter/Strainer - Offset



FOSO: Flange - Orifice (Slip On)



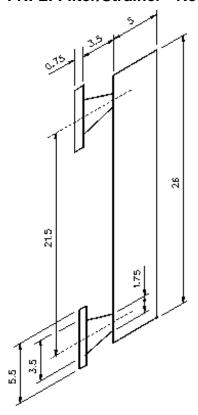
FOWN: Flange - Orifice (Weld Neck)



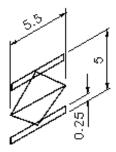
FPPL: Fixed Length Pipe - Without Flanged Ends



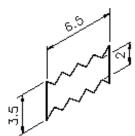
FRFL: Filter/Strainer - Return



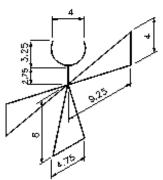
FTFL: Flame Trap



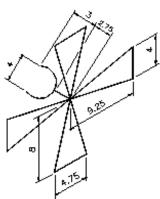
FXFL: Filter/Strainer - Return



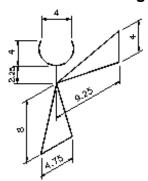
H3FL: Valve - Three-Way Control (Hand Indicator)



H4FL: Valve - Four-Way Control (Hand Indicator)



HAFL: Valve - Angle Control (Hand Indicator)



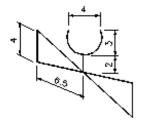
HCSC: Olet - Half Coupling (Screwed)



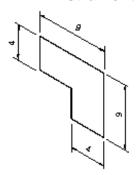
HCSW: Olet - Half Coupling (Socket Weld)



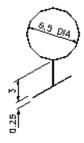
HVFL: Valve - Control Hand Indicator



IAFL: Instrument - Angle



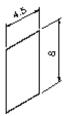
IDFL: Instrument - Dial (Flanged)



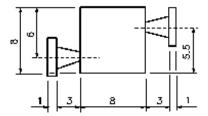
IDPL: Instrument - Dial



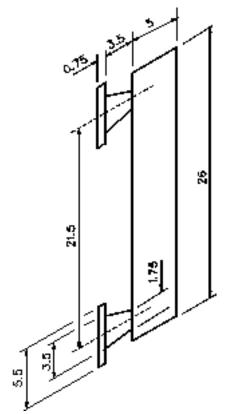
IIFL: Instrument



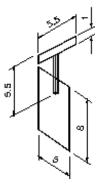
IOFL: Instrument - Offset



IRFL: Instrument - Return



ITFL: Instrument - Tee



KABW: Cap - Butt Weld



KACP: Cap - Compression



KASC: Cap - Screwed



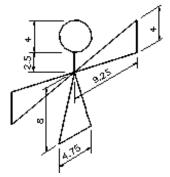
KASW: Cap - Socket Weld



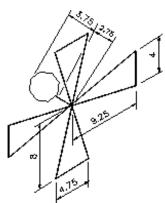
LABW: Olet - Latrolet (Buttweld)



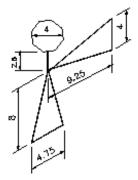
M3FL: Valve - Three-Way Control (Motorized Indicator)



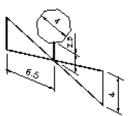
M4FL: Valve - Four-Way Control (Motorized Indicator)



MAFL: Valve - Angle Control (Motorized Indicator)



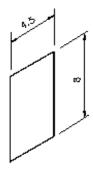
MVFL: Valve - Control (Motorized Indicator)



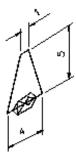
NBSC: Nipple - Screwed



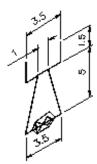
NCFL: Non-Category Item



NIPL: Olet - Nipolet (Plain End)



NISC: Olet - Nipolet (Screwed)



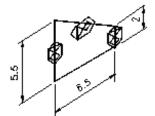
NRSC: Nippled - Screwed



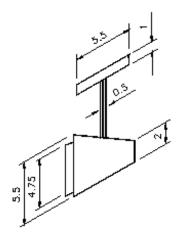
OP: Orifice Plate



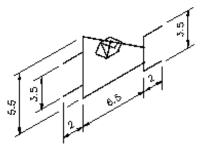
OTBW: Reducer - Eccentric with a Connection (Butt Weld)



OTFL: Reducer - Eccentric with a Connection (Flanged)



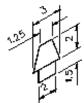
OTSC: Reducer - Eccentric with a Connection (Screwed)



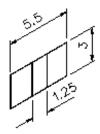
PF: Pipe Block - Fixed Length



PL: Plug



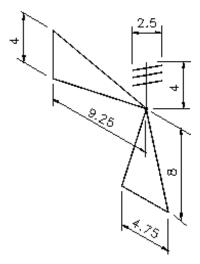
PR: Restrictor Plate



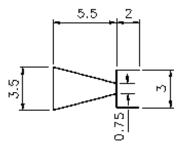
PV: Pipe Block - Variable Length



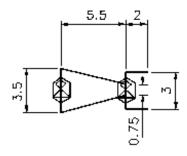
RAFL: Valve - Angle (Relief/Vent)



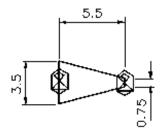
RBSC: Reducer - Concentric (Screwed Bush)



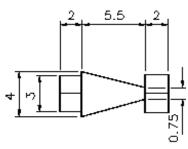
RBSW: Reducer - Concentric (Socket Weld)



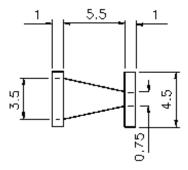
RCBW: Reducer - Concentric (Butt Weld)



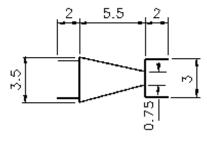
RCCP: Reducer - Concentric (Compression)



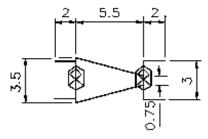
RCFL: Reducer - Concentric (Flanged)



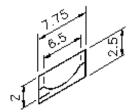
RCSC: Reducer - Concentric (Screwed)



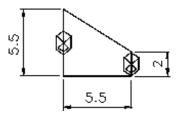
RCSW: Reducer - Concentric (Socket Weld)



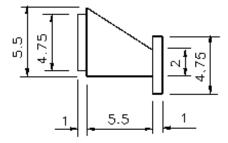
RD: Rupture Disk



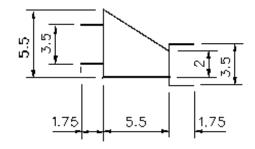
REBW: Reducer - Eccentric (Butt Weld)



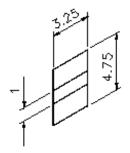
REFL: Reducer - Eccentric (Flanged)



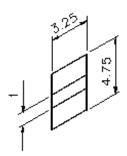
RESC: Reducer - Eccentric (Screwed)



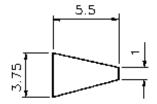
RF: Reducer - Special Reducing Flange



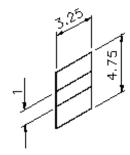
RFPL: Reducer - Connection Block



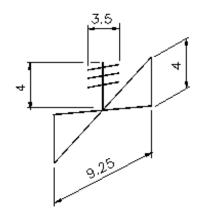
RNSC: Reducer - Concentric (Nipple)



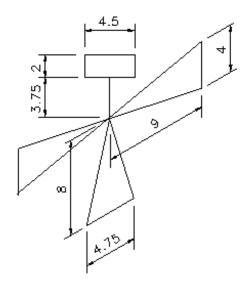
RP: Restrictor Plate



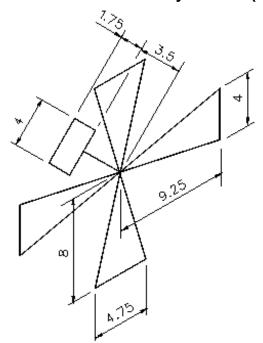
RVFL: Valve - Relief/Vent



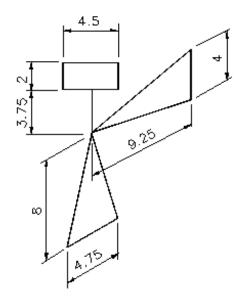
S3FL: Valve - Three-Way Control (Square Indicator)



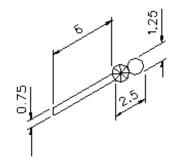
S4FL: Valve - Four-Way Control (Square Indicator)



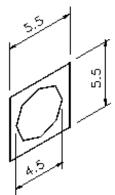
SAFL: Valve - Angle Control (Square Indicator)



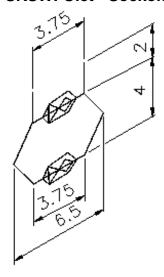
SB: Spectacle - Blind



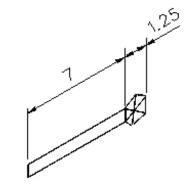
SGFL: Sight Glass



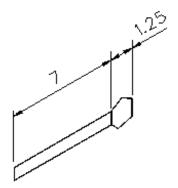
SKSW: Olet - Sockolet



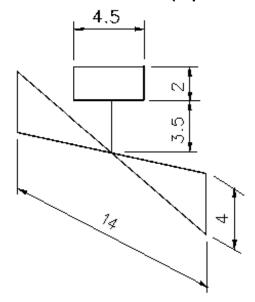
SP: Slip Plate



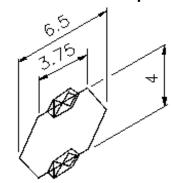
SR: Slip Ring



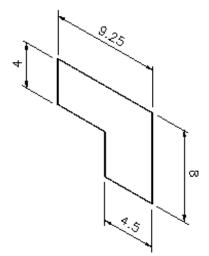
SVFL: Valve - Control (Square Indicator)



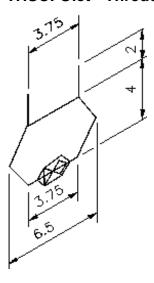
SWBW: Olet Sweepolet



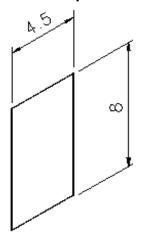
TAFL: Trap - Angle



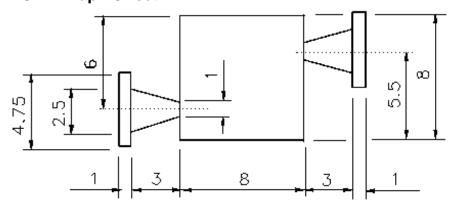
THSC: Olet - Thredolet



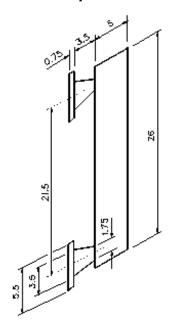
TIFL: Trap - Inline Straight Through



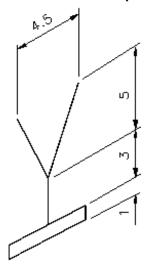
TOFL: Trap - Offset



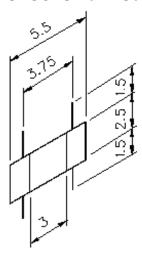
TRFL: Trap - Return



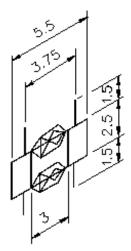
TUFL: Tundish (Funnel)



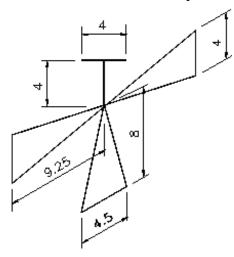
UNSC: Union - Screwed



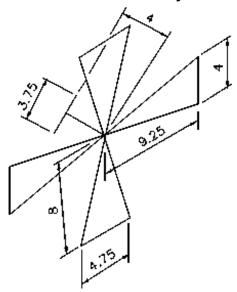
UNSW: Union - Socket Weld



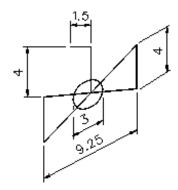
V3FL: Valve - Three-Way



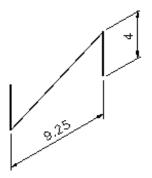
V4FL: Valve - Four-Way



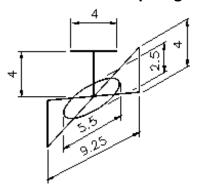
VBFL: Valve - Ball



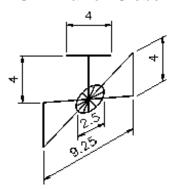
VCFL: Valve - Check



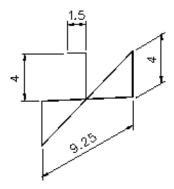
VDFL: Valve - Diaphragm



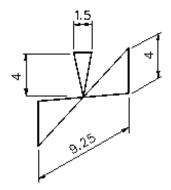
VGFL: Valve - Globe



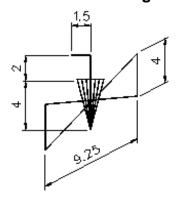
VKFL: Valve - Cock



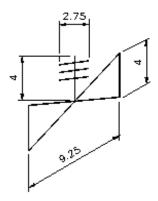
VNFL: Valve - Needle



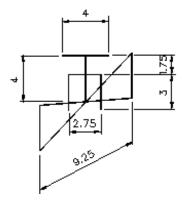
VPFL: Valve - Plug



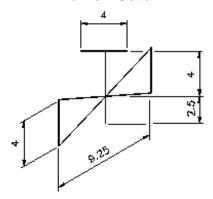
VRFL: Valve - Relief/Vent



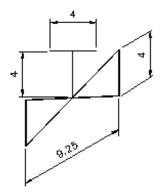
VSFL: Valve - Slide



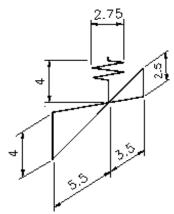
VTFL: Valve - Gate



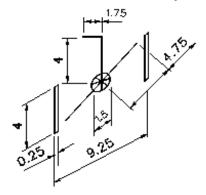
VVFL: Valve - Basic



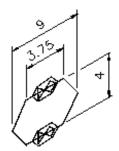
VXFL: Valve - Pressure Reducing



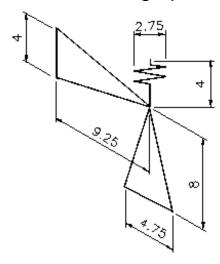
VYFL: Valve - Butterfly



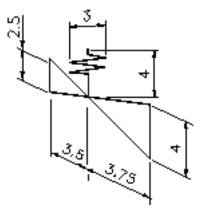
WTBW: Olet - Weldolet



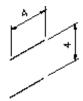
XAFL: Valve - Angle (Pressure Reducing)



XVFL: Valve - Pressure Reducing



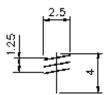
01HG: Support/Hanger



01SP: Used on valves with AV, V3, V4, VD, VG, or VV as the first two characters of their SKEYs



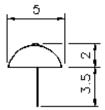
02SP: Used on valves with AR, RA, VR as the first two characters of their SKEYs.



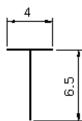
03SP: Used on valves with VB, VK, VY as the first two characters of their SKEYs.



04SP: Not Used



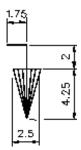
05SP: Used on valves with VT as the first two characters of their SKEYs.



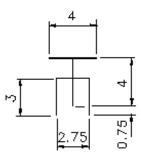
06SP: Used on valves with VN as the first two characters of their SKEYs.



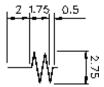
07SP: Used on valves with VP as the first two characters of their SKEYs.



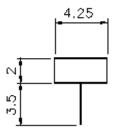
08SP: Used on valves with VS as the first two characters of their SKEYs.



09SP: Used on valves with AX, VX, or XA as the first two characters of their SKEYs.



10SP: Used on valves with SA, SV, S3, or S4 as the first two characters of their SKEYs.



11SP: Used on valves with MA, MV, M3, or M4 as the first two characters of their SKEYs.



Related Topics

• Appendix D: Symbols and Symbol Keys, page 555

Caps (SKEYs)

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Cap - Butt Weld	KABW	D	Y
Cap - Compression	KACP		Y
Cap - Screwed	KASC	3	Y
Cap - Socket Weld	KASW	 - 3	Y

Related Topics

• Appendix D: Symbols and Symbol Keys, page 555

Couplings (SKEYs)

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Nipple - Screwed (Running)	NRSC		Y
Nipple - Screwed (Barrel)	NBSC	\Diamond	Y
Coupling - Compression	COCP	H	Y
Coupling - Screwed	COSC	II	Y
Coupling - Socket Weld	COSW	垂	Y
Elbolet Coupling Butt Weld	CEBW	4	Y
Elbolet Coupling Screwed	CESC	节	Y
Elbolet Coupling Socket Weld	CESW		Y

Related Topics

Crosses (SKEYs)

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Cross - Butt Weld	CRBW	+	N
Cross - Compression	CRCP		N
Cross - Flanged	CRFL	-	N
Cross - Screwed	CRSC	ᅷ	N
Cross - Set On	CRSO	+	N
Cross - Set On Reinforced	CRRF	Reinferend	N
Cross - Socket Weld	CRSW	3 ₩€	N

Related Topics

Elbows and Bends (SKEYs)

The @ character in the symbol keys can be replaced with an integer value in the range 1 to 9, inclusive, to denote the number of segments. Currently, regardless of the value assigned to @, the symbol is drawn per the SKEY plotted isometric shape. The + character in the symbol keys may be replaced with an integer value in the range 1 to 9, inclusive, to denote the bend radius.

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Elbow - Butt Weld (90 and 45)	ELBW	7	N
Elbow - Butt Weld (90 and 45) With A Connection	ETBW	<u> </u>	N
Elbow - Butt Weld 180 Return (U Elbow)	EUBW	\bigcap	N
Elbow - Compression (90 and 45)	ELCP	B	N
Elbow - Compression (90 and 45) With A Connection	ETCP	بهلهم	N
Elbow - Screwed (90 and 45) With Male Ends	EBSC	7	N
Elbow - Screwed (90 and 45) With Female Ends	ELSC	يكر	N
Elbow - Screwed - Female (90 and 45) With A Connection	ETSC	ب ار در	N
Elbow - Socket Weld	ELSW	7	N
Elbow - Socket Weld With A Connection	ETSW	بعكوم	N
Bend - Flanged (All Angles)	BEFL		N
Bend - Flanged (All Angles) With A Connection	BTFL	ہگر	N

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Bend - Flanged 180 Return (U Bend)	BUFL	\bigcap	N
Bend - Miter Flanged	MIFL		N
Bend - Miter Flanged With A Connection	MTFL	PJ.	N
Bend - Miter Butt Weld	MIBW		N
Bend - Miter Butt Weld With A Connection	MTBW		Y
Bend - Lobster Back Flanged	L@FL	\searrow	N
Bend - Lobster Back Flanged With A Connection	T@FL	卢	N
Bend - Lobster Back Butt Weld	L@BW	7	N
Bend - Lobster Back Butt Weld With A Connection	T@BW		N
Bend - Pulled (All Angles)	PB+D	$\overline{}$	N
Bend - Pulled 180 Return (U Bend)	BU+D		N
Bend - Pulled (All Angles) With A Connection	TB+D		N

Related Topics

End Prep Connections

You can replace the ** characters in the symbol keys with one of the following end prep types:

End Type	End Prep Codelist Value Range	End Prep Type (**)
Flanged	2-199	FL (Flanged)
Male	321 - 329	LC (Liner - clamped)
	331 - 339	SC (Screwed)
	341 - 349	LC (Liner - clamped)
	351 - 359	LC (Liner - clamped)
	361 - 369	CP (Compression)
	371 - 379	SC (Screwed)
	381 - 389	SC (Screwed)
Female	420 - 429	SW (Socket Weld)
	440 - 449	SC (Screwed)
All Other Codelist Values		BW (Butt Weld)

Related Topics

Fixed Length Pipes (SKEYs)

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Fixed Length Pipe - With Flanged Ends	FPFL		N
Fixed Length Pipe - Without Flanged Ends	FPPL		Y

Related Topics

Flanges (SKEYs)

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Flange - Blind (Blank)	FLBL		Y
Flange - Flared/Loose Backing	FLFL		Y
Flange - Loose Backing	FLLB	}	Y
Flange - Reducing Concentric	FLRC	} 	Y
Flange - Reducing Eccentric	FLRE	}	Y
Flange - Slip On	FLSO		Y
Flange - Slip On With 'J' Type Weld	FLSJ	}	Y
Flange - Orifice Slip On	FOSO	} 	Y
Flange - Socket Weld	FLSW	₽	Y
Flange - Weld Neck	FLWN	▶	Y
Flange - Orifice Weld Neck	FOWN	⊳ →	Y

Related Topics

Inline Filters (SKEYs)

You can replace the ** characters in the symbol keys with an end prep type. For more information, see *End Prep Connections*, page 607.

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Filter/Strainer - Straight Through	FI**		Y
Filter/Strainer - Angle	FA**		N
Filter/Strainer - Offset	FO**		N
Filter/Strainer - Return	FR**		N

Related Topics

Instruments (SKEYs)

You can replace the ** characters in the symbol keys with an end prep type. For more information, see *End Prep Connections*, page 607.

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Instrument	II**		Y
Instrument - Angle	IA**		N
Instrument - Offset	IO**	[]	N
Instrument - Return	IR**		N
Instrument - Dial	IDPL	9	N
Instrument - Dial Flanged	IDFL	9	N
Orifice Plate	OP	8	Y
Restrictor Plate	PR	8	Y
Rupture Disk	DR	<i>-</i>	Y
Valve - Angle Relief/Vent	RA**	\(\frac{1}{3}\)	Y
Valve - Angle Pressure Reducing	XA**	7	Y

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Valve - Control	CV**	☆	Y
Valve - Angle Control	CA**	烙	Y
Valve - 3-Way Control	C3**	%	N
Valve - 4-Way Control	C4**	Pleas Spindle	N
Valve - Control Square Indicator	SV**	₩	Y
Valve - Angle Control Square Indicator	SA**	烃	Y
Valve - 3-Way Control Square Indicator	S3**	⅓	N
Valve - 4-Way Control Square Indicator	S4**	Phu Spindle	N
Valve - Control Motorized Indicator	MV**	SZ V	Y
Valve - Angle Control Motorized Indicator	MA**	\bowtie	Y
Valve - 3-Way Control Motorized Indicator	M3**	₩	N
Valve - 4-Way Control Motorized Indicator	M4**	Phus Spindle	N
Valve - Control Hand Indicator	HV**	\bowtie	Y
Valve - Angle Control Hand Indicator	HA**	Ŋ	Y

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Valve - 3-Way Control Hand Indicator	H3**	*	N
Valve - 4-Way Hand Indicator	H4**	Plus Spindle	N
Valve - Pressure Reducing	XV**	Ž	Y
Valve - Relief/Vent	RV**	槸	Y

Related TopicsAppendix D: Symbols and Symbol Keys, page 555

LJSE Type Flanges (SKEYs)

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Lap Joint Ring (Loose Backing Flange)	FLRG	H]	Y
Lap Joint Stub End (Loose Backing Flange)	FLSE	∬• - ₹	Y

Related Topics

Miscellaneous Items (SKEYs)

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Flow Indicator	FLOW	-	Y
Hanger/Support	01HG		Y

Related Topics

Miscellaneous Pipe Components (SKEYs)

You can replace the ** characters in the symbol keys with an end prep type. For more information, see *End Prep Connections*, page 607.

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Block - Angle	BA**	<u> </u>	N
Expansion Bellows	EX**	CAN CONTRACT OF THE CONTRACT O	Y
Flame Trap	FT**	\Diamond	Y
Flexible Hose	FX**	∿ ₩	Y
Hose Coupling	CH**	\boxtimes	Y
Non-Category Item	NC**		Y
Block Offset	BO**	[]	N
Plug	PL	[<⊅	Y
Restrictor Plate	RP	8	Y
Block - Return	BR**	<u> </u>	N
Sight Glass	SG**		Y
Slip Plate	SP		Y
Slip Ring	SR		Y
Spectacle Blind	SB	7	Y
Tundish (Funnel)	TU**	Υ	Y

Related Topics

Olets (SKEYs)

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Olet - Half Coupling Screwed	HCSC	ہے۔ ا	Y
Olet - Half Coupling Socket Weld	HCSW	, 	Y
Olet - Latrolet Butt Weld	LABW	5000	Y
Olet - Latrolet Screwed	LASC	5	Y
Olet - Latrolet Socket Weld	LASW	بنج	Y
Olet - Nipolet Screwed	NISC	, 	Y
Olet - Nipolet Plain End	NIPL	,4,	Y
Olet - Sockolet	SKSW	, 4	Y
Olet - Sweepolet	SWBW	<u>کھ</u> ے۔	Y
Olet - Thredolet	THSC	چ <u>ل</u> چ	Y
Olet - Weldolet	WTBW	بطر	Y
Instrument Tee - Flanged	ITFL	二	N

Related Topics

Operators (SKEYs)

Description	Key	Plotted Isometric Shape	User- Definable (Yes/No)
Fitting used on valves with AV, V3, V4, VD, VG, and VV as the first two characters of the SKEY	01SP	T	Y
Fitting used on valves with AR, RA, and VR as the first two characters of the SKEY	02SP	#	Y
Fitting used on valves with VB, VK, and VY as the first two characters of the SKEY	03SP		Y
Fitting used on valves with VT as the first two characters of the SKEY	05SP	T	Y
Fittings used on valves with VT as the first two characters of the SKEY	06SP	\bigvee	Y
Fittings used on valves with VP as the first two characters of the SKEY	07SP		Y
Fittings used on valves with VS as the first two characters of the SKEY	08SP	币	Y
Fittings used on control valves with AX, VX, and XA as the first two characters of the SKEY	09SP	*	Y
Fittings used on control valves with SA, SV, S3, and S4 as the first two characters of the SKEY	10SP	一	Y
Fittings used on control valves with MA, MV, M3, and M4 as the first two characters of the SKEY	11SP	\overline{Q}	Y

Description	Key	Plotted Isometric Shape	User- Definable (Yes/No)
Fittings used on control valves with HA, HV, H3, and H4 as the first two characters of the SKEY	12SP	个	Y
Fittings used on control valves with CA, CV, C3, and C4 as the first two characters of the SKEY	13SP	7	Y

Related Topics

Other End Connections (SKEYs)

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Erection Weld or Connection on fittings with an SW end	SW	£	Y
Erection Weld or Connection on fittings with an SC end	SC	С	Y
Erection Weld or Connection on fittings with a CP end	СР]	Y



• For SW, SC, or CP type end connections, the software fills in the plotted shapes for erection items and leaves the shapes open for fabrication type items. If the connecting pipe fabrication category is erection, then the software overrides whatever is defined in the fitting.

Related Topics

Reducers (SKEYs)

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Reducer - Concentric Butt Weld	RCBW	~ D~	Y
Reducer - Concentric Fabricated From Plate	CPBW	7	Y
Reducer - Concentric Swaged From Pipe	CSBW	7	Y
Reducer - Concentric Butt Weld With a Connection	CTBW		Y
Reducer - Concentric Fabricated From Plate With a Connection	CZBW	-63-	Y
Reducer - Concentric Swaged From Pipe With a Connection	CXBW	-63-	Y
Reducer - Concentric Compression	RCCP	€ ◯ 8	Y
Reducer - Concentric Flanged	RCFL		Y
Reducer - Concentric Fabricated From Plate Flanged	CPFL		Y
Reducer - Concentric Swaged From Pipe Flanged	CSFL		Y
Reducer - Concentric Fabricated From Plate Flanged w/ Connection	CZFL		Y
Reducer - Concentric Swaged From Pipe Flanged w/ Connection	CXFL		Y
Reducer - Concentric Nipple	RNSC	Δ	Y
Reducer - Concentric Screwed	RCSC	-EXE-	Y
Reducer - Concentric Screwed w/ Connection	CTSC	-£%-	Y
Reducer - Concentric Screwed Bush	RBSC	-t D₹-	Y
Reducer - Concentric Socket Weld Bush	RBSW	()-	Y
Reducer - Eccentric Butt Weld	REBW	-6	Y

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Reducer - Eccentric Fabricated From Plate	EPBW	2	Y
Reducer - Eccentric Swaged From Pipe	ESBW		Y
Reducer - Eccentric Butt Weld With a Connection	OTBW	₽	Y
Reducer - Eccentric Fabricated From Plate With a Connection	EZBW	4	Y
Reducer - Eccentric Swaged From Pipe With a Connection	EXBW	Ð	Y
Reducer - Eccentric Screwed	RESC		Y
Reducer - Eccentric Screwed With a Connection	OTSC	- 1	Y
Reducer - Eccentric Flanged	REFL	I	Y
Reducer - Eccentric Fabricated From Pipe Flanged	EPFL		Y
Reducer - Eccentric Swaged From Pipe Flanged	ESFL	I	Y
Reducer - Eccentric Flanged With a Connection	OTFL	₽	Y
Reducer - Eccentric Fabricated From Plate Flanged With a Connection	EZFL	l‡:	Y
Reducer - Eccentric Swaged From Pipe Flanged With a Connection	EXFL	1	Y
Reducing Block	RFPL		Y

Related Topics

Tees (SKEYs)

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Tee - Butt Weld	TEBW	ر ب <u>وام</u> ر	N
Tee - Compression	TECP	,,,,	N
Tee - Flanged	TEFL	r F	N
Tee - Screwed	TEFL	ہے ا د ب	N
Tee - Set On	TESO		N
Tee - Set On Reinforced	TERF	Relations	N
Tee - Socket Weld	TESW	بع الحد	N
Tee - Swept Branch Butt Weld	TSBW	Ţ ,	N
Tee - Swept Branch Flanged	TSFL	<u>L</u>	N
Tee - Swept Branch Compression	TSCP	ب و آو ب	N
Tee - Swept Branch Socket Weld	TSSW	, <u>, , , , , , , , , , , , , , , , , , </u>	N

Related Topics

Traps (SKEYs)

You can replace the ** characters in the symbol keys with an end prep type. For more information, see *End Prep Connections*, page 607.

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Trap - In-Line (Straight Through)	TI**		Y
Trap - Angle	TA**	- -	N
Trap - Offset	TO**		N
Trap - Return	TR**	- [N

Related Topics

Valves (SKEYs)

You can replace the ** characters in the symbol keys with an end prep type. For more information, see *End Prep Connections*, page 607.

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Valve - Angle	AV**	烃	Y
Valve - Angle Pressure Reducing	AX**	戍	Y
Valve - Angle Relief/Vent	AR**	戊	Y
Valve - Ball	VB**	D € 0	Y
Valve -Basic	VV**	⋫	Y
Valve - Butterfly	VY**	استارا	Y
Valve - Check	VC**	7	Y
Valve - Check	CK**	⋈	Y
Valve - Cock	VK**	₩	Y
Valve - Diaphragm	VD**	β \$₹3	Y
Valve - Gate	VT**	⋈	Y
Valve - Globe	VG**	⊅	Y
Valve - Needle	VN**	炒	Y
Valve - Plug	VP**	D ∮ 3	Y
Valve - Pressure Reducing	VX**	₽	Y
Valve - Relief/Vent	VR**	🖄	Y
Valve - Slide	VS**	⋈	Y

Three-Way Valves

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Valve - 3-Way	V3**	₩	N

Four-Way Valves

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Valve - 4-Way	V4**	Phus Spindle	N

Related Topics

Vents (SKEYs)

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Rupture Disk	RD	00	Y

Related Topics

Welds (SKEYs)

Description	Key	Plotted Isometric Shape	User-Definable (Yes/No)
Weld - Site	WS	-	Y
Weld - Field Fitted	WF	-	Y
Weld - Workshop	WW		Y
Weld Mitre (Shop)	WM		N
Weld Mitre (Site)	WM	-	N
Weld Mitre (Offshore)	WM	₩	N
Weld Mitre (Field Fit)	WMF	-	N
Weld Mitre (Field Fit)	WMF	₩	N
Offshore Weld	WO	```	Y
Offshore Weld-Field Fit	WOF	*	Y

Related Topics

Drawings and Reports Glossary

A

alternative text (AText)

An ISOGEN feature that allows you to change or remove any standard ISOGEN text on an isometric drawing.

annotations

Dimensions, notes, symbols, or reports placed in a drawing to provide information or comments.

В

batch extraction

A method of extracting drawings in which many drawings are extracted at a time. You can schedule the extraction process and set its recurrence.

BOM (Bill Of Materials)

An indented and exploded list of parts in a feature.

C

codelist

A set of acceptable values for a particular attribute which can be referred to by an index number or selected in a combo box. For example, the codelist for the material specification allows you to select from a set of standard entries, such as ASTM A183-F316 Stainless Steel.

condition rule

The condition that must be met before any rule is applied in a drawing view style.

cut pipe report

A list that shows the length of each piece of pipe in the pipeline.

D

detail sketch

A small drawing inset on an isometric drawing that provides more information about the fabrication or erection of a component represented in the isometric drawing.

dimension rule

The dimension processing for a drawing view style. This rule controls the style, units, and placement of dimensions in a drawing view.

G

graphic rule

The graphic processing rule for a drawing view style. A graphic rule could be vector hidden line (VHL), for example.

Н

HITS report

A diagnostic tool generated by the ISOGEN interface used to analyze the data collected from the 3D piping model when a problem extracting an isometric occurs.

I

implied piping component

Piping components which are created in the database as part of a feature, but which are not represented graphically.

interactive extraction

A method of extracting drawings in which you can extract only a single isometric at a time. This method is primarily used for testing purposes.

ISOGEN

A software component that generates isometric drawings. Alias, Ltd develops ISOGEN.

isometric

Relating to or being a drafting system characterized by three equal axes at right angles; a view in which the horizontal lines of an element are drawn at an angle to the horizontal and all verticals are projected at an angle from the base.

isometric drawing

A line drawing, always shown in an isometric perspective, that is used for fabricating and erecting piping systems. An isometric drawing usually shows a complete line from one piece of equipment to another and provides all information necessary for fabrication and erection of piping.

isometric drawing style

A set of options that control the drawing output, including format and content. Each style has a unique set of options stored in reference data. You can use the Isometric Style Options Browser to edit the options.

L

label rule

The label processing rule for a drawing view style. A label rule could locate white space in a drawing view, for example.

М

material list

An option category that controls the format and content of the bill of materials.

MTO neutral file

A nongraphic output file that can be fed into a material control system. MTO stands for Material Take-Off.

0

olet

A type of branching fitting that is preshaped to the curvature of the run pipe. Types of olets include sockolets, nipolets, and elbolets.

options file

A set of options that drives the ISOGEN interface.

orthographic

A depiction of an object created by projecting its features onto a plane along lines perpendicular to the plane.

Р

PCF (Piping Component File)

The intermediary file that SmartPlant 3D generates and delivers to the Alias ISOGEN software with the goal of creating an isometric piping drawing.

PDS (Plant Design System)

A comprehensive, intelligent computer-aided design and engineering application for the process and power industries. PDS consists of integrated 2D and 3D modules that correspond to engineering tasks in the plant design workflow.

parts

The physical components that comprise a feature and are generally selected by the software. For example, the flanges, gaskets, and the gate valve itself are examples of the parts comprising the gate valve feature.

port

A connection point to a pipe or a component such as a valve.

pipeline

A set of graphically connected pipe runs including all branches.

pipe run

A pipe run is a connected series of pipe components that have the same nominal piping diameter (NPD) and flow direction, and are governed by the same pipe specification.

pipe specification

A collection of the allowed types of piping commodities and requirements for those types that can be used in the design of a piping system to which the specification applies. These are also known as specification parts. Each individual piping specification includes additional rules that determine the types of parts that must be used in certain design circumstances as well as suggestions for parts that could be used in other circumstances.

R

repeatability

A process in which re-extracted drawings only change where modifications have been made to the model. When a drawing is re-extracted, the software recalls the repeated data to avoid changing drawing split points and part, weld, and spool numbers.

S

spool

A prefabricated portion of a piping system that is an assembly of fittings, flanges, and pipe. A spool does not include bolts, gaskets, valves, or instruments.

symbol key (SKEY)

A code for a symbol on an isometric drawing. For example, FLSO is the SKEY for a slip on flange.

U

update rule

The rule for a drawing view style that determines when to update the view.

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