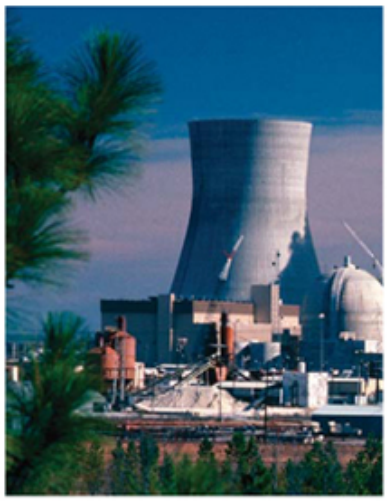
SmartPlant 3D

Common Applications Tutorials

Process, Power & Marine


INTERGRAPH

Version 2009 SP1 April 2009 DSP3D-TP-100009A

Copyright

U.S. Government Restricted Rights Legend

Intergraph Corporation
P.O. Box 240000
Huntsville, AL 35813
Street address: 170 Graphics Drive, Madison, AL 35758

Terms of Use

Warranties and Liabilities

Trademarks

Table of Contents

Session 1: Creating a Session File

Objective:

By the end of this session, you will be able to:

- Create a new session file from a session file template.

Prerequisite Session:

- SP3D Overview

Overview:

You open a session file to begin working in SP3D. You can open an existing session file stored on a disk or create a new session file from a session template. After you create a new session file from a template, you must use the **Define Workspace** command to select the portion of the model you want to view on your computer before you can edit the model.

The session template or session file does not store plant design data. It stores only your personal default settings for the commands, the active task environment, and display geometry. The display geometry is used for display only. It is updated when you execute the **Define Workspace** or the **Refresh** commands. When you select an object to edit or use an object as input for another design operation, the software retrieves the object information needed directly from the server. It is not possible to edit or directly reference outdated object information.

The session file stores the following:

- Last active task environment
- Window layout
- Named views
- Surface style rules to apply (The rule definitions are stored in the model.)
- **Tools > Options** settings
- Graphic geometry for display (Display list)
- Identity of the filter used to create the display list (The filter definitions are stored in the model.)
- Defaults for each command's options and settings

When you exit SP3D, the graphics and command defaults you established while modeling can be stored in a session file on your computer. Later, you can quickly start another design session by opening the saved session file. This gets you back to exactly the same modeling situation you had when you exited SP3D and saved the session file.

This session shows you how to create the session file from a template. Refer to *Session 2* for the procedure for defining a filter for the session file.

Steps for Creating a Session File:

Create a session file and save the file on your desktop.

1. Start SP3D software by clicking **Start > Programs > Intergraph SmartPlant 3D >**

SmartPlant 3D.

2. The **New** dialog box is displayed and the **empty** template is selected by default. Select any template options other than **empty** and click **OK.**

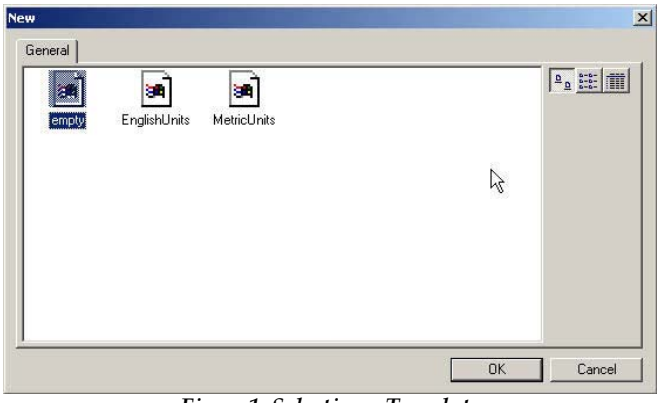


Figure 1: Selecting a Template

Notes:

- Three standard templates are available with SP3D: **empty, EnglishUnits,** and **MetricUnits**.
- Administrators can use the **empty** template to create custom templates. The **EnglishUnits** template displays measurements in the English system, and the **MetricUnits** template displays measurements in the Metric system.
- Your administrator can add templates for you to select from that are customized for the preferences of the different discipline areas.
- Saving a session file without defining a workspace is useful only if you want to create another template. **Refer to** ***Session 2: Define Workspace with a System Filter*** for how to define a filter.

3. Click **Save**.

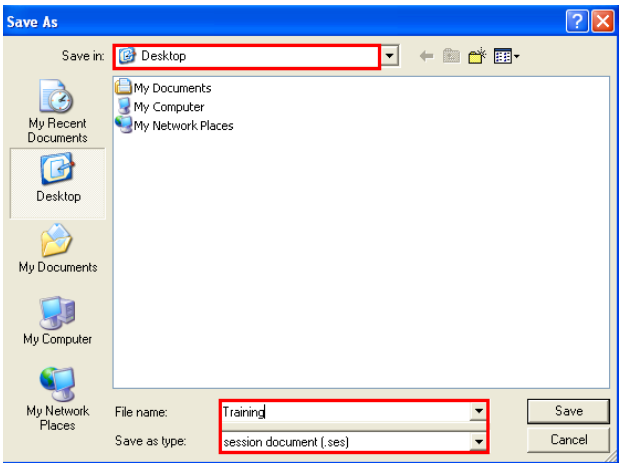


Figure 2: Naming a Session File

Tips:

- The **Save As** dialog box prompts you to specify the name of the session file and the location. You can navigate to and select a folder on a local drive or a network drive. Session files have a .ses extension.
- The default directory proposed when you save a session file is defined under **Tools> Options>File Locations** tab**> WorkSpace** option.

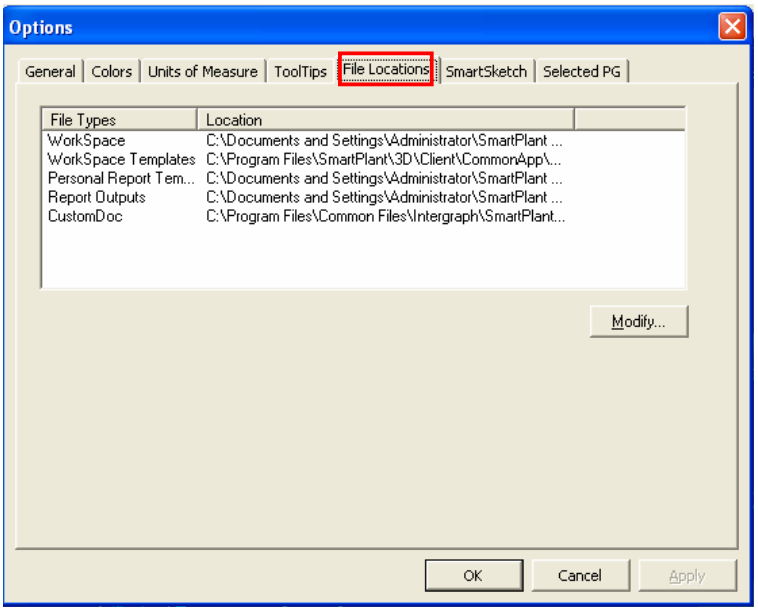


Figure 3: Selecting a Location for Session Templates

For more information related to session files and defining a workspace, refer to the following topics in the user guide *CommonUsersGuide.pdf*:

- *Common: An Overview*
- *Managing Sessions: An Overview*

Session 2: Defining a Workspace Using a System Filter

Objective:

By the end of this session, you will be able to:

- Define a workspace by creating and using a System filter.

Prerequisite Sessions:

- SP3D Overview
- Creating a Session File

Overview:

When you create a session file from a session template, you must define the portion of the plant model you want to view or edit using the **Define Workspace** command. This command queries the model database using a filter that describes the characteristics of the data you want to see. You can select an existing filter or define a new filter. The graphics for the objects you selected are displayed on your computer. When you save the session file, the graphics retrieved from the server by the **Define Workspace** and **Refresh** commands are stored in the session file.

Objects others create or edit in the model database after you use the **Define Workspace** command are not seen in your design session until you run the **Refresh** command to update the graphics displayed on your computer. This command reruns the define workspace filter and retrieves only the new or changed data that meets the filter criteria.

This session shows how to create a filter that uses the system hierarchy grouping of the design data.

Notes:

- The relationships between design objects ensure that any supporting design data required to edit a design object is automatically retrieved from the server when you edit the object. The retrieval of the required data from the server does not depend on the data you included in your session. All edits will automatically use the updated information from the model on the server.
- Opening a session file and running the **Refresh** command is much quicker than running the **Define Workspace** command and reduces the load on the server. The **Define Workspace** command activates all the design objects retrieved by the filter to generate the graphic display. The display graphics are stored in the session file. When you activate a session file and run the **Refresh** command, only the changed and new graphics are retrieved from the server to the client computer (with associated activation of the design object). Once the graphics are included in the session file, the design objects are only activated when you select them for edit or reference. You should maintain a directory for saved

session files that use the different filters needed for your daily work. As a rule, you should begin working in SP3D by opening an existing session file.

Steps for Defining a Workspace Using a System Filter:

Open a session file and define a workspace containing the objects in Unit **U01**, Area **A2**, by creating a System filter, **Unit 1**, for those objects. After the workspace is defined, it should contain the objects as shown in Figure 1.

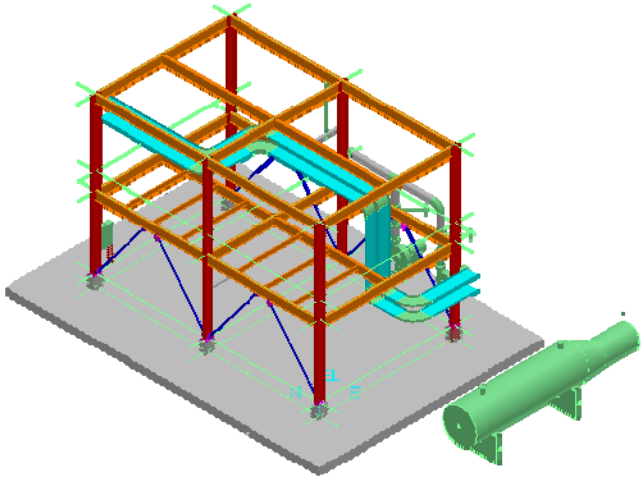


Figure 1: Output: After Defining Workspace for Unit U01

1. Click the **File** menu and select the **Define Workspace…** command.

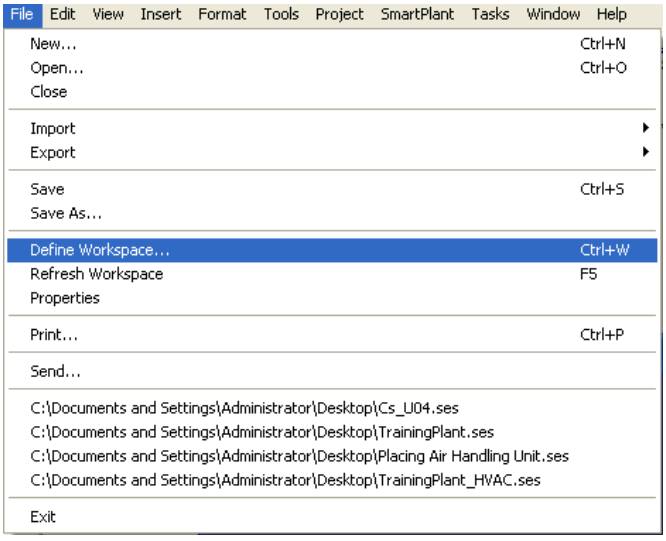


Figure 2: Define Workspace Command on the File Menu

The **Define Workspace** dialog box appears.

Note:

- You can also press the **Ctrl+W** keys to open the **Define Workspace** dialog box.

2. In the **Filter** drop-down list of the **Define Workspace** dialog box, select the **More…** option.

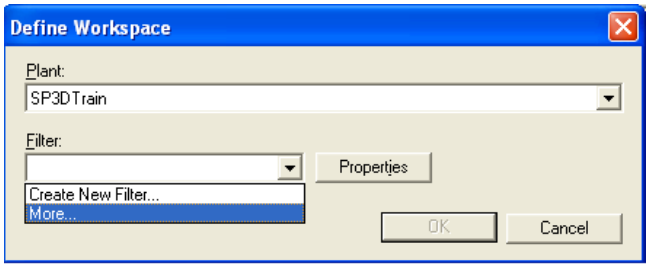


Figure 3: Define Workspace Dialog Box

The **Select Filter** dialog box is displayed.

3. In the **Select Filter** dialog box, select **My Filters** and click the **New Filter (Simple or Asking)** icon to open the **New Filter Properties** dialog box.

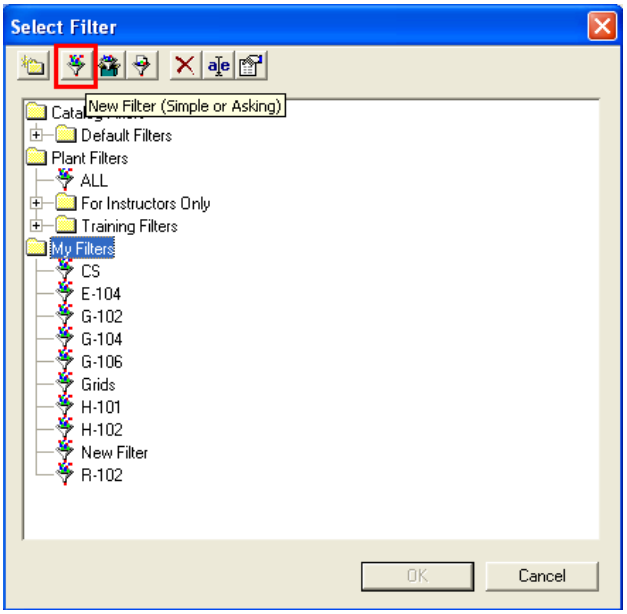


Figure 4: Select Filter Dialog Box

Note:

- You can select an existing filter or create a new filter.
- Catalog filters are stored in the catalog referenced by the model. These filters contain only criteria related to the data model such as object type and

properties.

- Plant filters are stored in the model database and can be based on model objects such as Systems and Named Spaces (volumes).
- You need Read permission to view the Catalog and Plant filters defined by the site administrator.

4. In the **New Filter Properties** dialog box, type **Unit 1** in the **Name** box. Then, expand **A2** and **CS**. Press and hold the **Ctrl** key, and then, select **U01** and **U01 CS**.

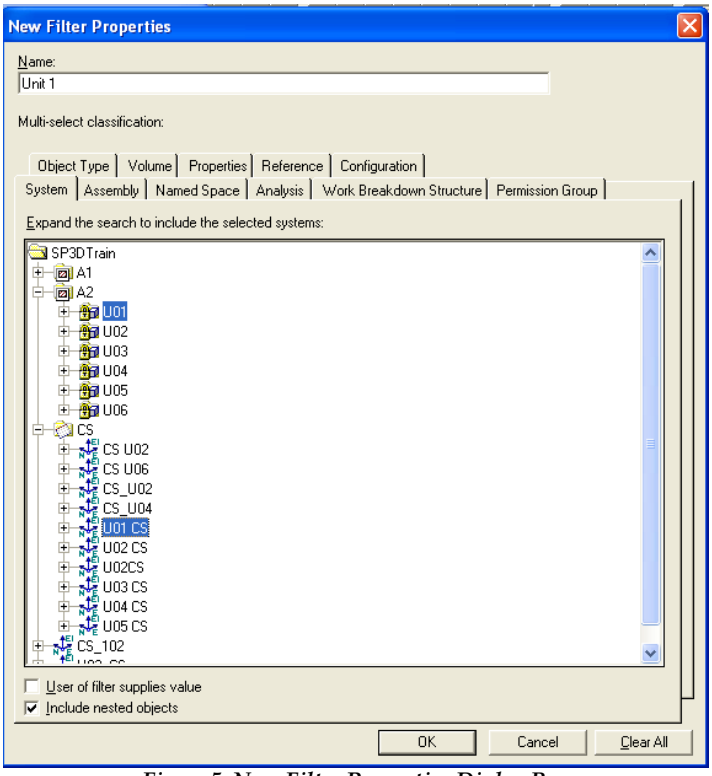


Figure 5: New Filter Properties Dialog Box

Notes:

- You need write permission to the permission group of the parent folder in the Plant or Catalog folders to create a filter. You can always create filters in your **My Filters** folder.
- A filter name can have any alphanumeric characters, including space, but cannot contain apostrophes. The name must be unique in the model database.
- The **New Filter Properties** dialog box consists of several tabs that define search criteria. The **System**, **Assembly**, and **Named Space** tabs expand the search for objects within a model that meet the criteria defined on these tabs. The **Volume**, **Permission Group**, and **Object Type** tabs restrict or limit the objects returned to those that meet the additional criteria, if any, defined on those tabs.
- The filter in this tutorial is called a System filter because only **System** criteria are

used to select the desired data. The system hierarchy is like a folder hierarchy. It is used to organize the model for design purposes (as opposed to construction management). Every part in the plant has one and only one system–parent. So, this can be a primary way to select the portion of the model you want to work on or view.

- If the **Include nested objects** check box is selected, the filter will retrieve all objects nested under the selected systems. If the check box is clear, only the selected system objects themselves will be retrieved.
- You can apply the other criteria if you need to be more selective about the data you want to see. This will be covered in an advanced tutorial on defining filters.

5. Click **OK** to complete the filter definition process.
6. In the **Select Filter** dialog box, select **Unit 1** and click **OK** to set your workspace to display **Unit 1**.

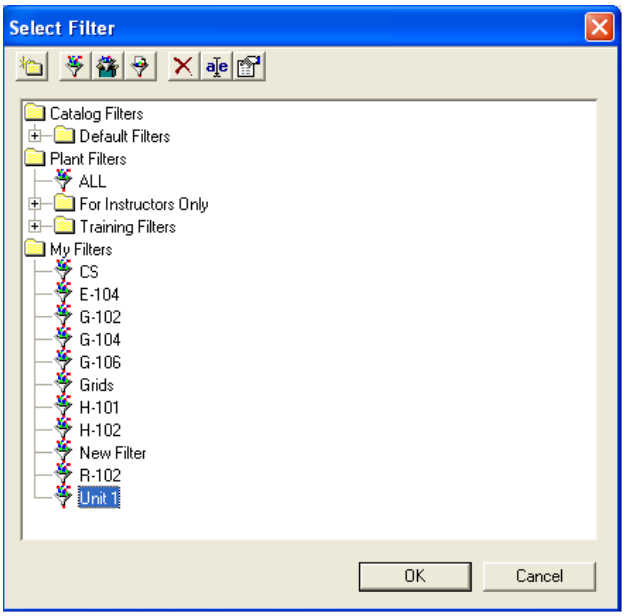


Figure 6: New Filter Added

7. In the **Define Workspace** dialog box, click **OK**.

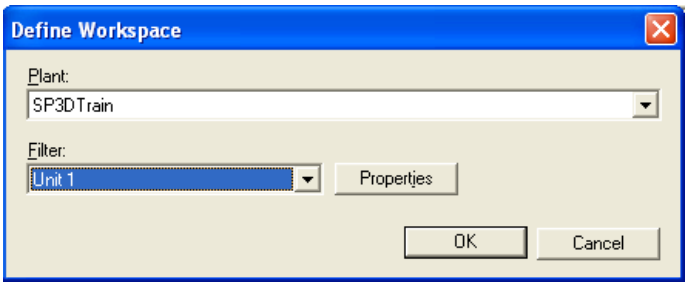


Figure 7: Selecting the Created Filter From Define Workspace Dialog Box

You have defined a workspace by creating the filter **Unit 1**.

Note:

- To fit all objects retrieved by the simple filter **Unit 1** in the active view, click the **Fit** button on the **Common** toolbar.

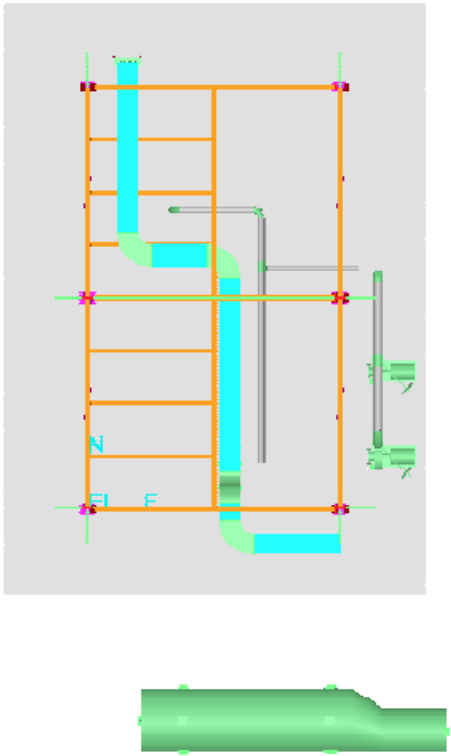


Figure 8: View of the Workspace After Creating a System Filter

For more information related to defining a workspace, refer to the following topics in the user guide *CommonUsersGuide.pdf*:

- *Defining Workspace Content: An Overview*
- *Using the File Menu: An Overview*

Session 3: Defining a Workspace Using a Volume Filter

Objective:

By the end of this session, you will be able to:

- Define a workspace by creating and using a volume filter.

Prerequisite Sessions:

- SP3D Overview
- Creating a Session File
- Defining a Workspace Using a System Filter

Volume Filter:

A filter can restrict the selection of objects to whose geometry range box as found relative to the global coordinate system lies within a given volume or volumes. This restriction can be combined with any other criteria available in the **Filter Properties** dialog box.

Steps to Define a Workspace Using a Volume Filter:

Define a workspace that loads all objects of system **Area A2 / Unit U01** that are in **Volume U01**. After the workspace is defined, it should contain the objects shown in Figure 1.

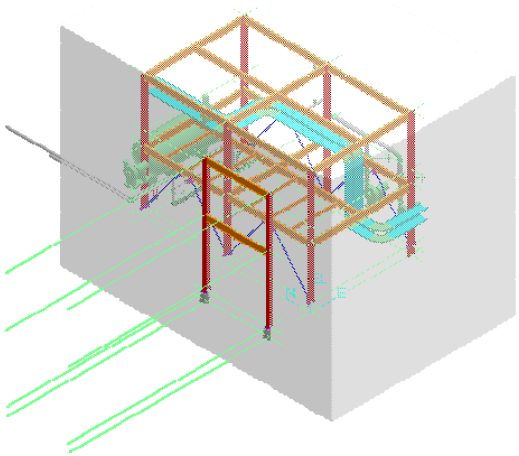


Figure 1: Final Output: Objects Found by the Volume Filter

8. Select Define Workspace command and create a new filter by using the **New Filter (Simple or Asking)** button in the **Select Filter** dialog box.

The **New Filter Properties** dialog box is displayed:

- The **New Filter Properties** dialog box consists of several tabs that define search criteria. The **System**, **Assembly**, and **Named Space** tabs expand the search for objects within a model that meet the criteria defined on these tabs. The **Volume**, **Permission Group**, and **Object Type** tabs restrict or limit the objects returned to those that meet these additional criteria.
- If you don't select anything on the **System**, **Assembly**, or **Named Space** tabs, then the restrictions you define on the other tabs are applied against all objects in the database. So, if you want to make sure your filter gets all graphic objects that meet the volume criteria, regardless of what system the objects belong to, then don't select a system at all and just define the volume criteria (skip step 2 below).
- The **System** tab provides a tree view of all the systems defined in the model. Systems can span disciplines and include many types of objects. A plant is the highest system in the hierarchy and includes all subsystems.
- The **Named Space** tab provides tree view of folders and named spaces in the folders. Named spaces are volumes that represent fire zones, design areas, volumes to be output to drawings, or other non-solid spatial concepts. Including the Named spaces in the workspace enables you to view the size, shape, and position of the volumes.
- Only the SmartMarine product extensively uses the **Assembly** tab. The only objects in SmartPlant which appear on the Assembly tab are piping spools. The assembly hierarchy is used for manufacturing purposes.

9. In the **New Filter Properties** dialog box, type **Unit 1 in Volume U01** in the **Name** box. Then, expand **A2** and **CS**. Press and hold the **CTRL** key and then, select **U01** and **U01 CS**.

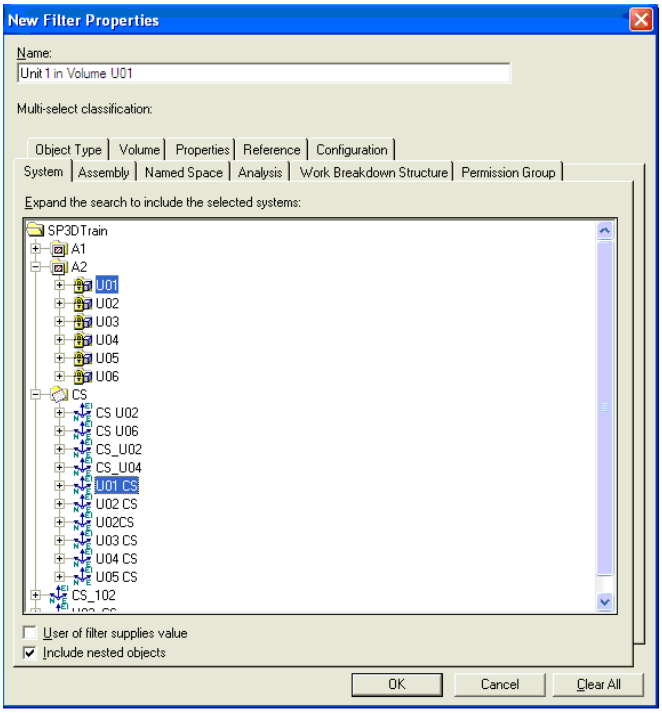


Figure 2: New Filter Properties Dialog Box

10. To retrieve only objects in the selected systems that have geometry in **Volume U01**, click the **Volume** tab and select **Volume U01**.

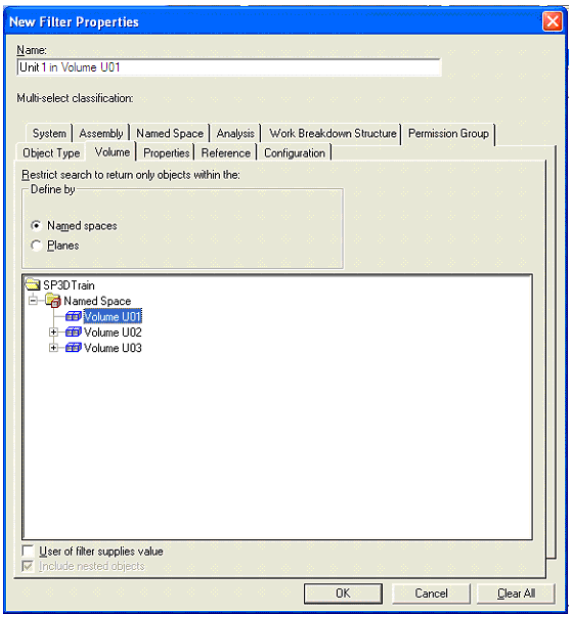


Figure 3: Volume Tab: New Filter Properties Dialog Box

Note:

- There are two options on the **Volume** tab, **Named Spaces** and **Planes**. The **Named Spaces** option allows you to select one or more existing named volumes to define the volume criteria. The **Planes** option allows you to define a single volume using six existing reference planes. (Refer to the tutorial for SP3D Structure Session 2: *Placing Coordinate Systems/Grids*.).

11. Click **OK**.

12. In the **Select Filter** dialog box, select **Unit 1 in Volume U01** and click **OK**.

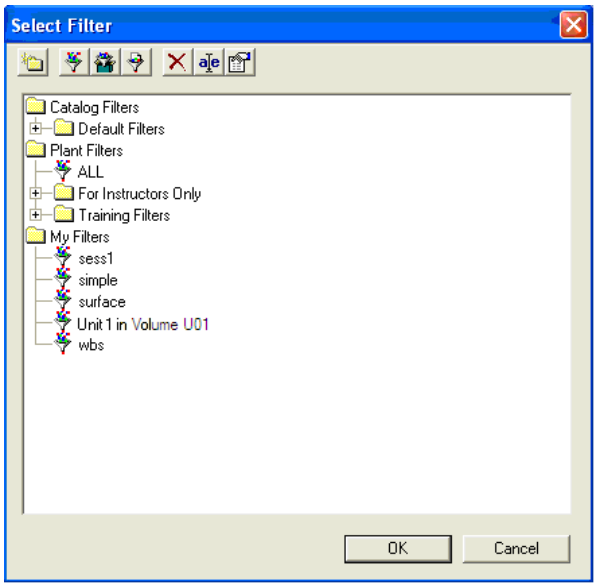


Figure 4: Select Filter Dialog Box

13. In the **Define Workspace** dialog box, click **OK**.

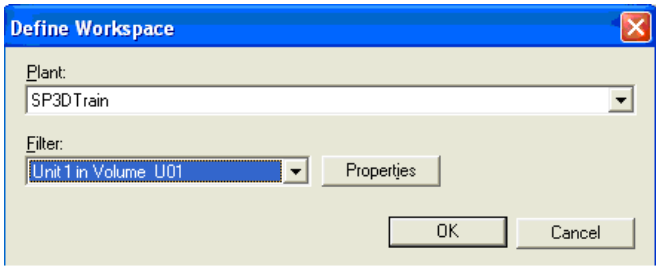


Figure 5: Define Workspace Dialog Box

You have defined a workspace that includes all geometric objects retrieved by the volume filter **Volume_U01**.

Notes:

- To fit all objects retrieved by the volume filter in the active view, click the **Fit** button on the **Common** toolbar. The view will look like Figure 6.

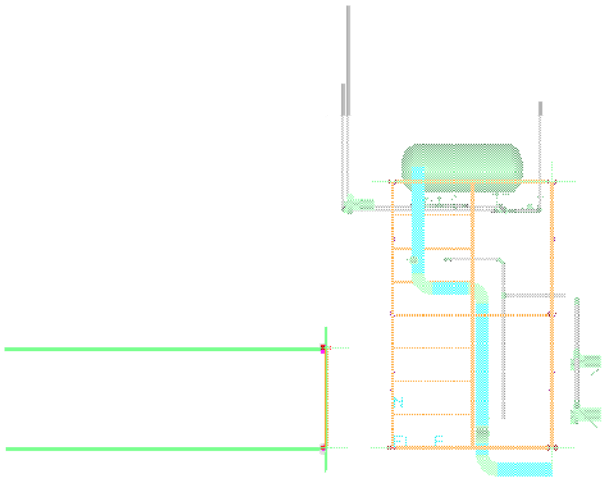


Figure 6: Active View After Clicking the Fit Button

- Save the session file so you can open it to quickly resume your work from where you left off after exiting SP3D.

For more information related to defining a workspace, refer to the following topics in the user guide *CommonUsersGuide.pdf*:

- *Defining Workspace Content: An Overview*
- *Using the File Menu: An Overview*

Session 4: Manipulating Views

Objective:

By the end of this session, you will be able to:

- Create a graphic window and manipulate the view in the active window.

Prerequisite Sessions:

- SP3D Overview
- Creating a Session File
- Defining a Workspace Using a System Filter
- Defining a Workspace Using a Volume Filter

Manipulating Views:

You can manipulate the graphic windows and the view in the windows using the **Window** and **View** menus and the commands on the **Common** toolbar.

The **Window** commands create new graphic windows and arrange the position of the existing windows. The **Common** toolbar has the most commonly used commands to manipulate the views within the windows. These commands can also be accessed from the **View** menu.

Figure 1: Common Toolbar – View Manipulation Buttons

Note:

- The view commands do not cancel an active command. When you complete or cancel the view manipulation command, the interrupted command is reactivated in the same command step where you left it.

The view manipulation buttons on the **Common** toolbar are described below:

Note:

- The orientations used in viewing commands are relative to axes of the Active Coordinate System selected on the PinPoint ribbon.

- **Clip by Object**: Creates a rectangular volume based on the maximum range of the selected objects and then limits or clips the display to show only graphics within that clipping volume.
- **Clip by Volume**: Creates a clipping volume by specifying two area definition points (the cross-section of the volume) and two depth points (the extent of the volume).

- **Clear View Clipping**: Removes all clipping of the display from the active window.

Active View Control: Enables you to graphically edit the parameters of the view, including the clipping by showing active view's eye point, focus point, and viewing volume in another view and allowing you to manipulate them.

Common Views: Displays a small window that sets the view direction of the active view when you click one of the standard viewing angles.

View by Points: Defines the viewing plane by three points.

Rotate View: Rotates a view about a point or an axis.

Named Views: Assigns a name and a description to a view and stores the definition for later reuse. The default views are **Top**, **Front**, **Right**, and **Isometric**.

Note:

- A named view definition includes the clipping but does not define the perspective angle of the view or it's rendering characteristics. The named views you define are saved in the session file.

- **Zoom Tool**: Provides the functions of zoom, zoom area, and pan commands. If you get into the habit of using this command for all its functions, you will save yourself time.
  - Left-button drag zooms out and in.
  - Middle-wheel zooms
  - Two left button clicks zooms to the rectangular area.
  - Middle-button drag pans.

- **Zoom Area**: Zooms to a specific rectangular area of the model.
- **Fit**: Fits the view to the range of the currently selected objects. If no objects are selected, it fits to all displayed objects.

Notes:

- The **Fit** command allows you to fit the objects in one or more views. When you click the **Fit** button, the software fits the active view, and prompts you to select another view to fit. You can also fit all views at the same time by holding the SHIFT+A or CTRL+A keys and clicking the **Fit** button.
- If you press SHIFT and then click the **Fit** button, the fit is to all visible objects in the clipping volume, regardless of the objects that are currently selected.

- Pan: Moves the view in the current view plane.

Steps to Manipulate Views in the Active Window:

The following activity uses the view manipulation commands in a workspace containing the entire Training Plant.

1. Click the **Common Views** button on the **Common** toolbar. The **Common Views** dialog box appears.

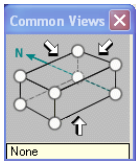


Figure 2: Common Views Window

2. To change the view of the model in the Training Plant, select the **Looking North** view in the **Common Views** dialog box.

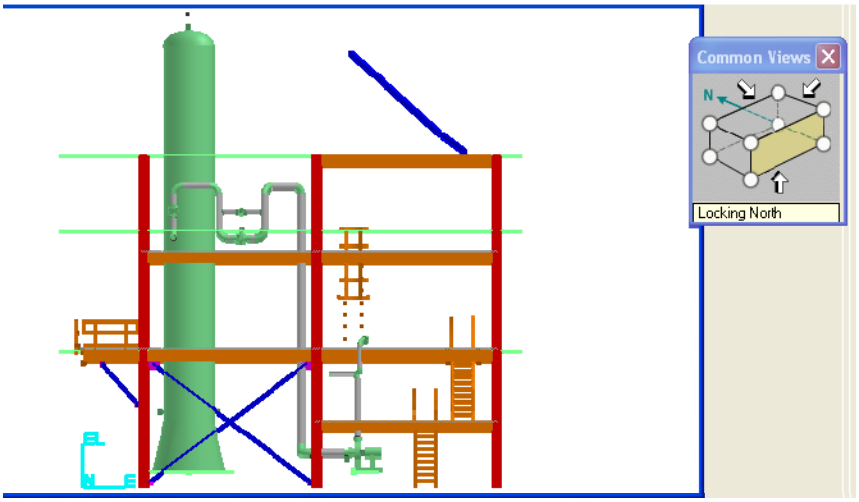


Figure 3: Looking North View

Note:

- You can change your view to **Isometric**, **Looking East**, **Looking South**, **Looking Plan**, and **Looking West** by using the **Common Views** dialog box. You can move this dialog to an unobtrusive location and leave it displayed for quick access while you execute other commands.

3. Change the view to isometric view and magnify the area highlighted in Figure 4 by clicking the **Zoom Area** button on the **Common** toolbar. Choose an area that just includes the vertical tank and the pump.

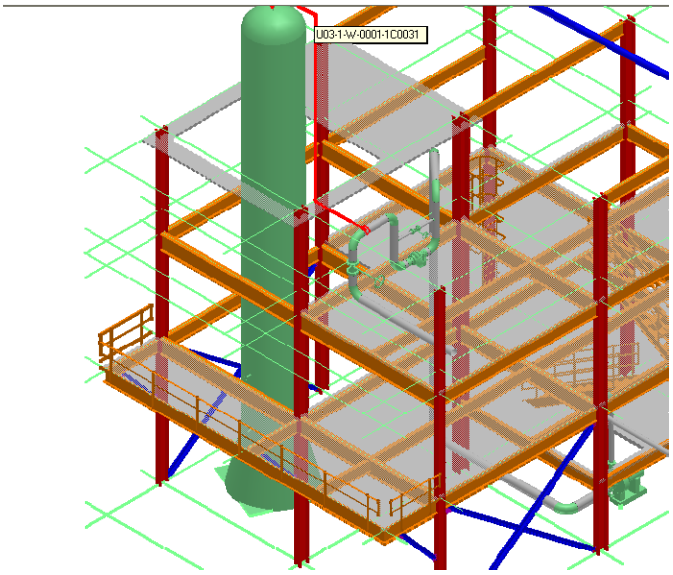


Figure 4: Isometric View

4. Click the **Named View** button on the **Common** toolbar.
5. In the **Named View** dialog box, type **Area 1** in the **Name** box and then type **Area around the equipment T-101** in the **Description** box.

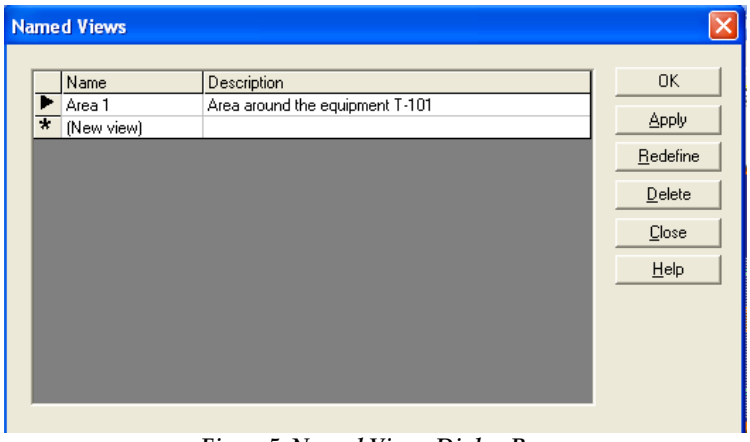


Figure 5: Named Views Dialog Box

6. Click **Apply** and then click **OK**.

Figure 6: Area 1: Area Around the Equipment T-101

The view you created is added to the list of named views and can be restored when required. The named view definition is stored in the session file.

7. To clip to a region around the equipment **P-101**, click the **Clip by Object** button and select the equipment and two beams above the equipment. The objects to define the range of the clipping. If the objects are already selected when you pick the **Clip by Object** command, then the clipping will use the selected objects without prompting you to select objects.

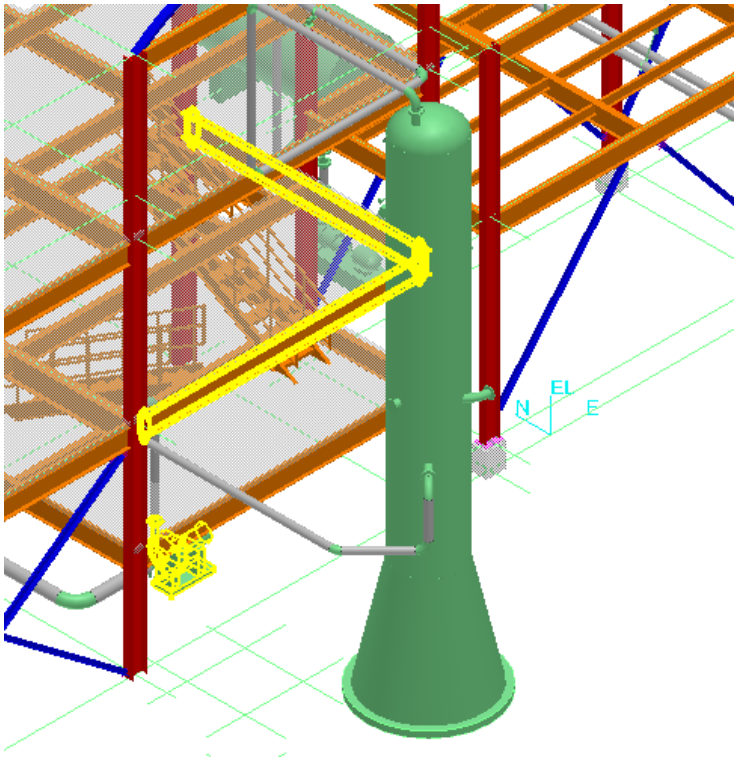


Figure 7: Clip by View Object

8. Click the **Accept** button on the **Clip by Object** ribbon.

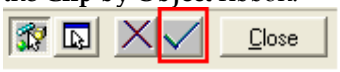


Figure 8: Accept Button on the Clip by Object Ribbon

Your view will look like Figure 9.

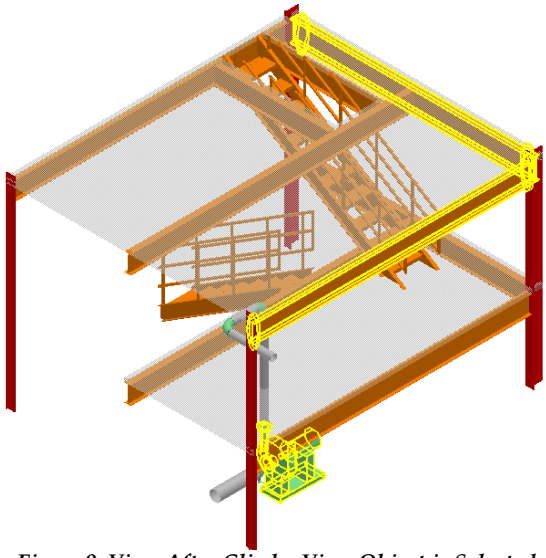


Figure 9: View After Clip by View Object is Selected

9. Graphically adjust the clipping volume by selecting the **Clip by Volume** command. The command will display a box showing the clipping boundaries. Select the top clipping boundary and move it up to see the third story bay.

Notes:

- Once you select the boundary, the command prompts you to locate a point. You can extend the boundary out by an approximate distance or an exact distance with PinPoint. You can set the clipping plane to a precise location by positioning the cursor over geometry.
- You can continue editing the other clipping boundaries and then terminate the command.
- You can also use the Clip by Volume command to define the clipping volume by four points. The first two points define a rectangle in horizontal plane. The second two points define the front and back clipping plane locations along the z-axis.

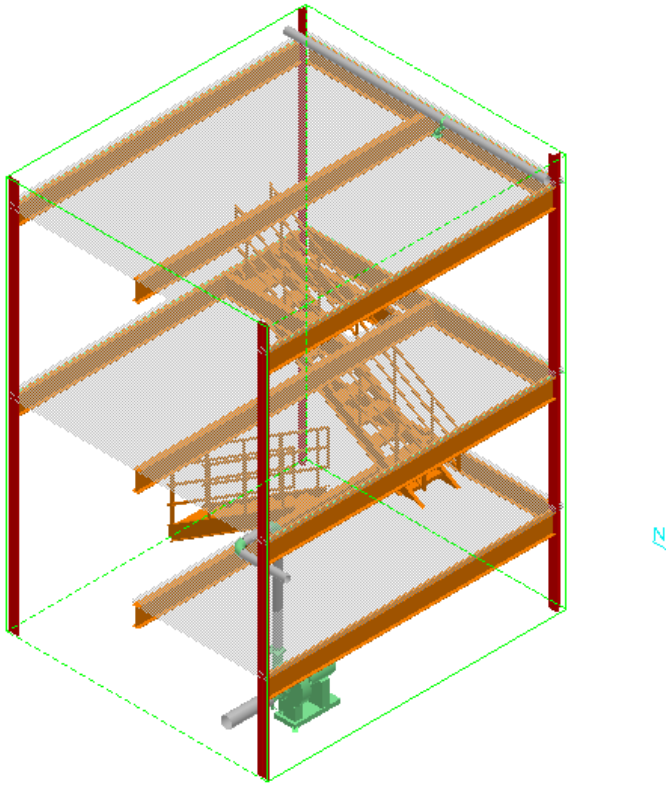


Figure 10: View After Clip by Volume is Selected and Top Plane Moved

10. Set the active view to Isometric view using the Common Views control. Select the **Active View Control** command on the common toolbar to modify the view parameters of the active view from a separate window.

Notes:

- The **Active View Control** command on the **Common** toolbar allows you to edit all the parameters of the view. It is most useful for graphically adjusting the clipping from a view where you can see all the geometry. You can experiment with other edits the Control can make, but it is generally better to use the common toolbar commands for those operations.
- The **Active View Control** window has a set of commands for manipulating the view within the Active View control. These commands **Top View**, **Side View**, **Front View, Pan, Zoom, Window Area,** and **Fit.**
- By default, the **Top View** button is selected when the command is activated.

Set the Active View Control view to the elevation view. Modify the top clipping boundary of the active view by dragging the dotted clipping plane above all graphics in the **Active View Control** window. Your active view will change to that shown in figure 12

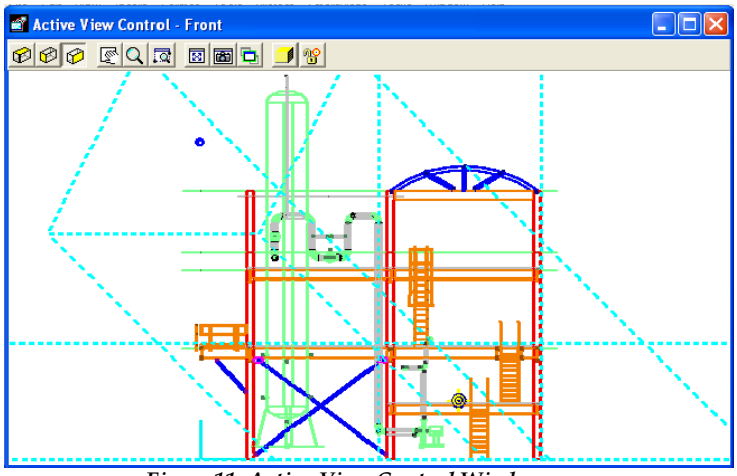


Figure 11: Active View Control Window

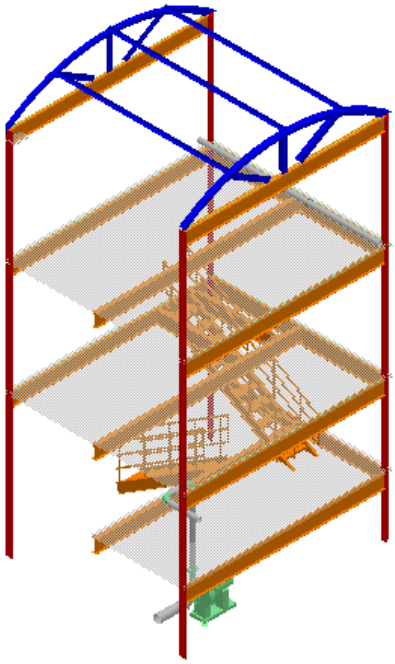


Figure 12: Active View

To clear the clipping, click the **Clear View Clipping** command on the **Common** toolbar.

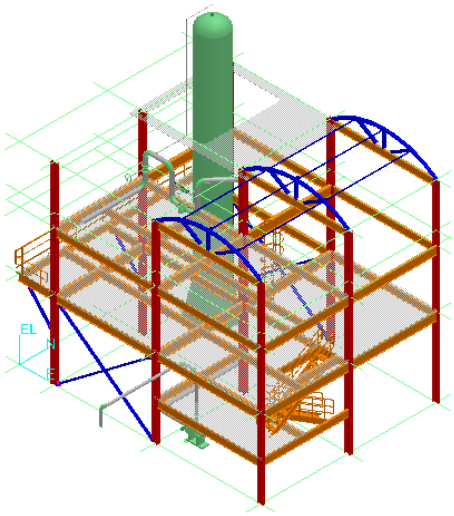


Figure 13: Clear View

11. To rotate the view about the vertical discharge of the pump, select the **Rotate View** command.

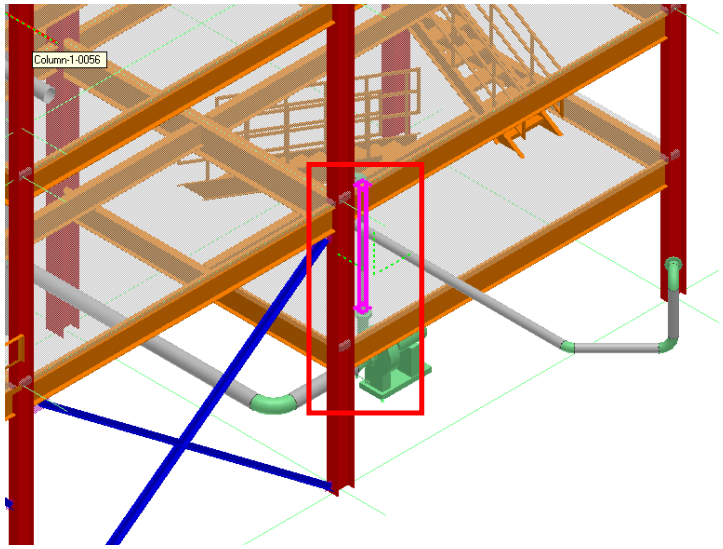


Figure 14: Rotate View

The rotate view command displays a coordinate system triad at the center of the active window. You have several options for rotating the view:

- Select an axis of the triad or any linear geometry in the model. Enter the degrees you want to rotate the view about the selected axis.
- Click over the selected axis and drag to dynamically rotate the view about that axis.
- Click somewhere else in the view away from the selected axis and drag to

dynamically rotate the view in three dimensions about the center of the view.

For this example, from the standard isometric view, select the vertical pipe connected to the pump discharge nozzle and key in 15 degrees in the field on the rotate ribbon. See Figure 15.

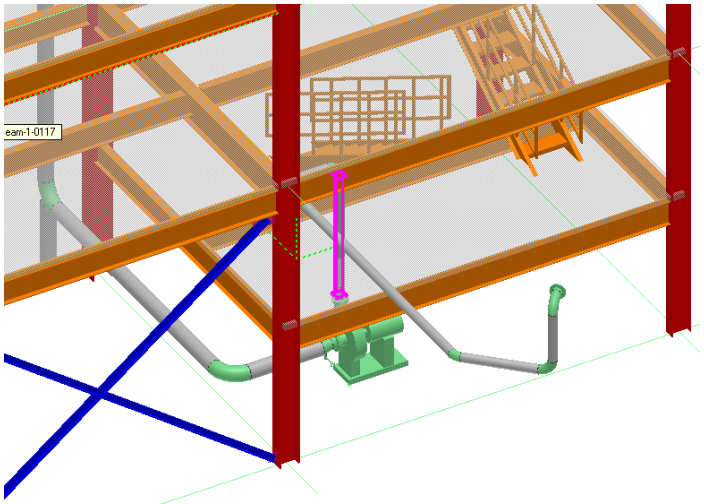


Figure 15: View After 15 Degree Rotation

Now, dynamically rotate the view by click over the vertical pipe and dragging the cursor. This gives you an easy way to look all around a specific object.

For more information related to manipulating a view, refer to the *Using the View Menu: An Overview* topic of the user guide *CommonUsersGuide.pdf.*

Session 5: Selecting Objects in a Model

Objective:

By the end of this session, you will be able to:

- Select objects individually, by fence, or by using filters.

Prerequisite Sessions:

- SP3D Overview
- Creating a Session File
- Defining a Workspace Using a System Filter
- Defining a Workspace Using a Volume Filter
- Manipulating Views

Overview:

You can select design objects for modification by using the **Select** command and the **Select by Filter** command.

The **Select** command is the first command on the vertical toolbar. The Select command ribbon bar contains a **Locate Filter** drop-down list, which limits the selection by object type such as equipment or pipe parts. Each task environment provides a unique list of filter options useful for that task. The Select command has the following selection options:

- Individual selection with the cursor in the graphic view or **Workspace Explorer**
- Multiple selections in the graphic view by fence with drag-drop feature of the cursor. You can choose to select all objects fully within the fence or all objects within and overlapping the fence boundaries.
- Multiple selections by "Shift-select" in the **Workspace Explorer**
- Single selection by keying in a name in the name field of the **Workspace Explorer**
- Multiple selections by keying in a name with wild card search criteria (*) in the name field of the **Workspace Explorer**

The **Tools > Select by Filter** command allows you to select objects by running a query on the objects in the workspace using one or more filters. You access the same filters through the **Select by Filter** command or the **Define Workspace** command.

Steps for Selecting Individual Objects Using the Select Command:

Select Pipeline **400-P** of Unit **U04** using the **Select** command. After selecting the pipeline, the view should look like Figure 1.

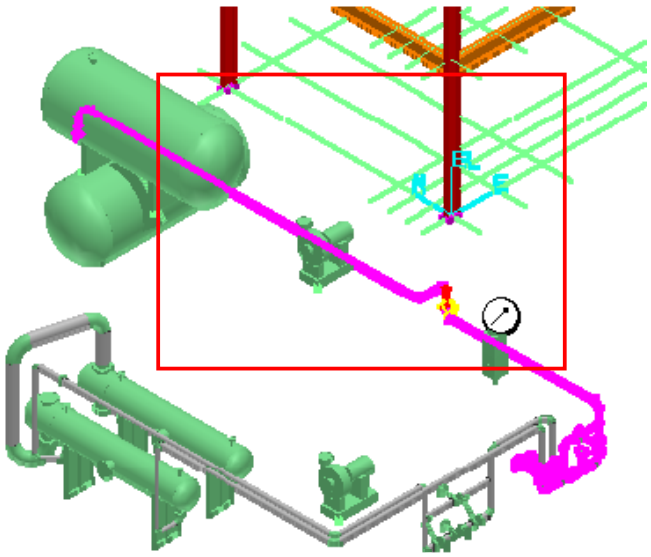


Figure 1: Selecting a Specified Object by Using the Select Command

Tip:

- You can set the colors used for highlighted and selected objects on the Colors tab of the Tools>Options dialog.

1. Define your workspace to display Unit **U04** and Coordinate System **U04 CS**.
2. Select the piping task environment by using the **Tasks > Piping** command.

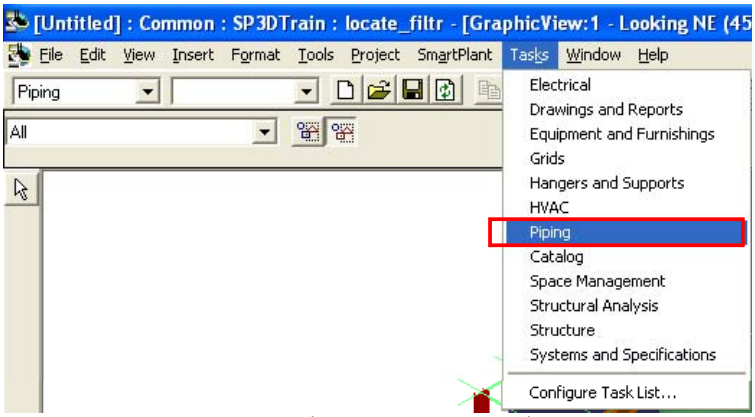


Figure 2: Tasks > Piping Command

3. Click the **Select** button on the vertical toolbar.

Figure 3: Select Button on the Vertical Toolbar

4. On the **Select** ribbon, select the filter **Pipelines** in the **Locate Filter** drop-down list to locate only the piping objects.

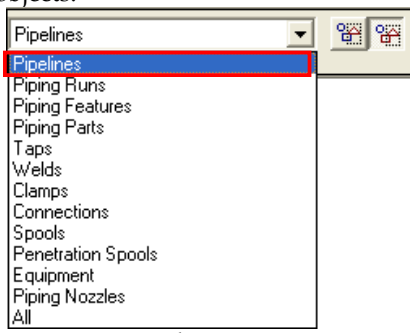


Figure 4: Locate Filter Drop-Down List

Tip:

- When you are working on a specific task, you can select objects of other disciplines by selecting the **All** option in the **Locate Filter** drop-down list. For example, if you are working in the Piping task environment, you can set the **Locate Filter** to **All**, and then select a structural member.

5. Point the cursor to Pipeline **400-P** until it is highlighted. Click the highlighted pipeline to select it in the view. You can select additional objects by holding down the CTRL key.

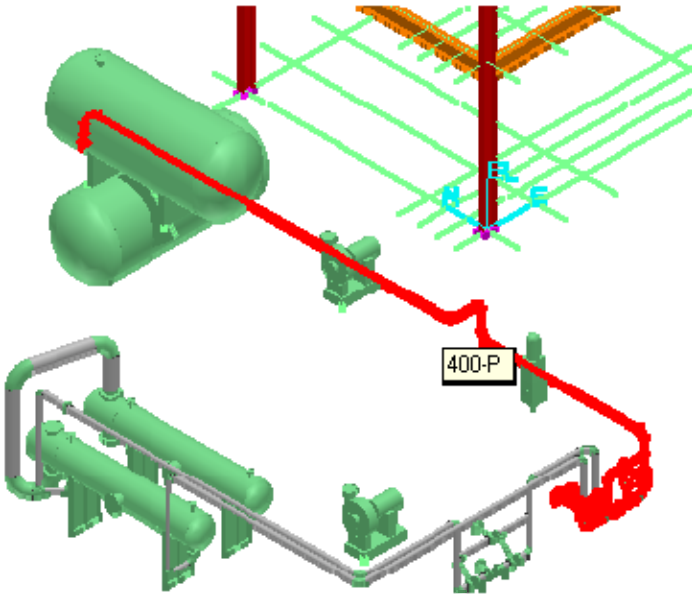


Figure 5: Highlighted Pipeline 400-P

Tips:

- You can change the option in the **Locate Filter** drop-down list after you select an object by using the **Edit > Locate Filter** command. The shortcut key combination for the **Locate Filter** command is CTRL+E. The option in the **Locate Filter** drop-down list is needed because the **Edit** ribbon for the first selected object replaces the **Select** command ribbon with the **Locate Filter** options.
- The select command locate behavior is adjusted by the Locate Zone and Dwell Time properties on the General tab of the Tools>Options command. The Locate Zone defines a tolerance in screen pixel dimensions about the cursor point. Objects are located if they are within this tolerance of the cursor point. The objects under the cursor point are located in order of their view-z position. The first object found at the cursor position is highlighted when you move your cursor over it. If you pause for the duration of the "Dwell Time", then the QuickPick tool will display showing all objects that are found under the current cursor position. You can then select your desired object by highlighting the options and picking on the QuickPick tool. (Refer to Figure 6.)

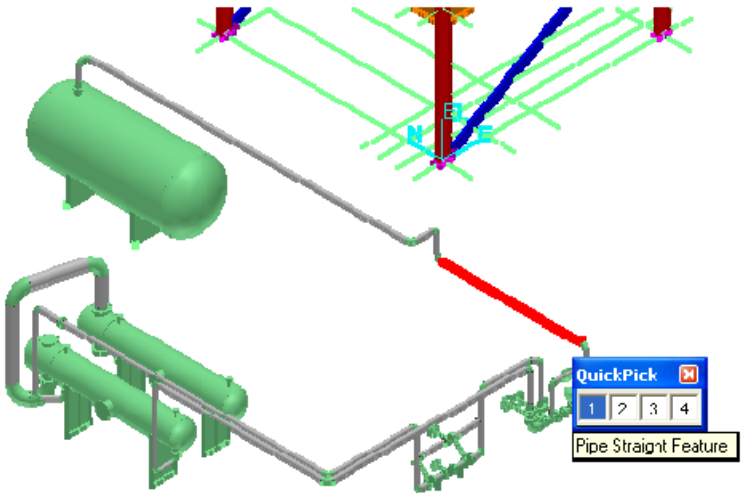


Figure 6: QuickPick Dialog Box

Steps for Selecting Objects by Using Inside Fence:

Select Pipeline **403-P** of Unit **U04** using the **Inside Fence** option of the **Select** command. After selecting the specified objects, the view should look like Figure 7.

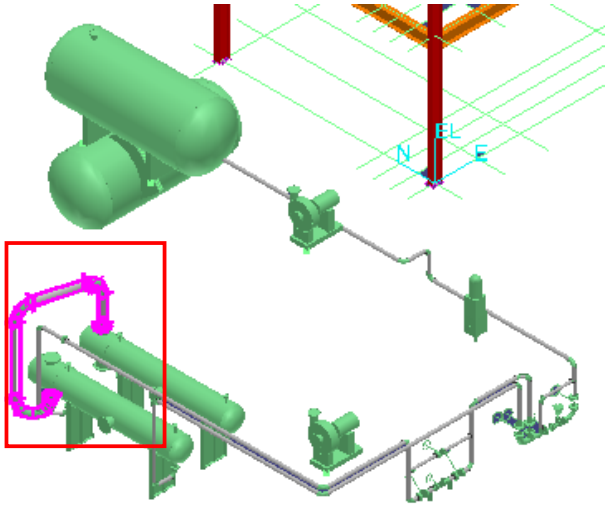


Figure 7: Selecting Specified Objects Using the Inside Fence Option of the Select Command

1. Define your workspace to display Unit **U04** and Coordinate System **U04 CS**.
2. Select the piping task environment by using the **Tasks > Piping** command.
3. Click the **Select** button on the vertical toolbar.

4. Select the filter **Pipelines** in the **Locate Filter** drop-down list to locate only the piping objects. Click the **Inside Fence** option on the **Locate Filter** ribbon. This option remains selected till the next use of the **Select** command or until you change the option.

Figure 8: Inside Fence Option on the Locate Filter Ribbon

5. Place a fence around Pipeline **403-P** by dragging the dashed fence lines around Pipeline **403-P**.

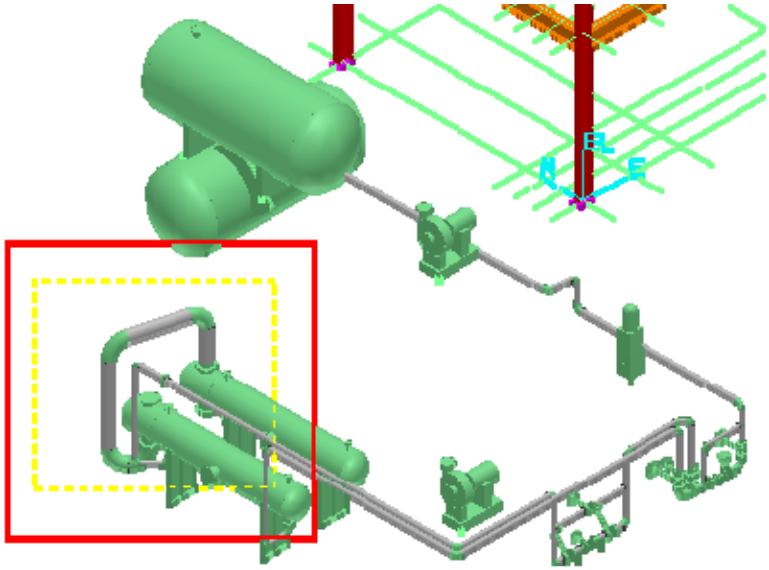


Figure 9: Inside Fence Created Around Pipeline 403-P

After creating the fence around the pipeline and releasing it, Pipeline **403-P** is selected. This will select only the objects that lie fully inside the fence.

Steps for Selecting Objects by Using Overlapping Fence:

Select Pipelines **403-P**, **402-P**, and **401-P** of Unit **U04** in your workspace using the **Overlapping fence** option of the **Select** command. After selecting the specified objects, the view should look like Figure 10.

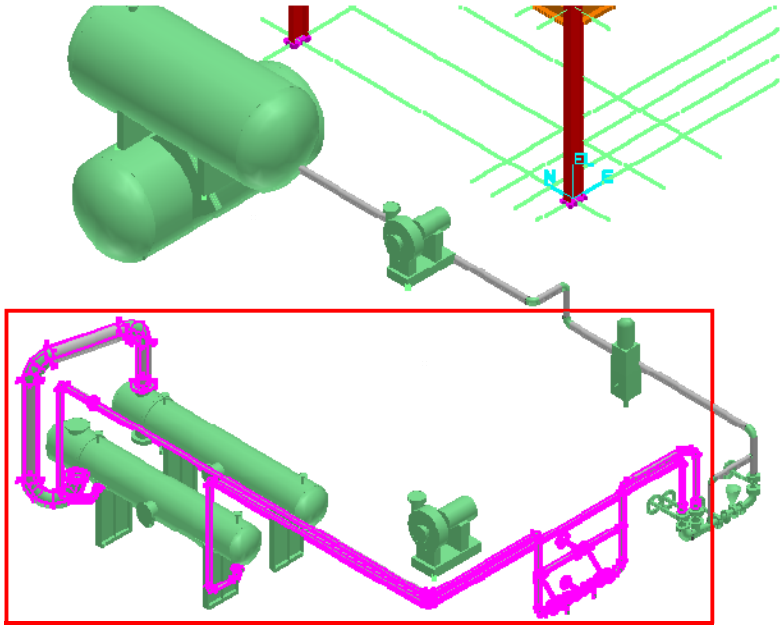


Figure 10: Selecting Specified Objects Using the Overlapping Fence Option of the Select Command

1. Define your workspace to display Unit **U04** and Coordinate System **U04 CS**.

2. Select the piping task environment by using the **Tasks > Piping** command.

3. Click the **Select** button on the vertical toolbar.

4. Select the filter **Pipelines** in the **Locate Filter** drop-down list to locate only the piping objects. Click the **Overlapping Fence** option on the **Locate Filter** ribbon. This option remains selected till the next use of the **Select** command or until you change the option.

Figure 11: Overlapping Fence Option on the Locate Filter Ribbon

5. Place a fence around Pipeline **403-P** by dragging the dashed fence lines around Pipeline **403-P**. The fence should cover some portion of Pipelines **402-P** and **401-P**, as indicated by the dotted lines in Figure 12.

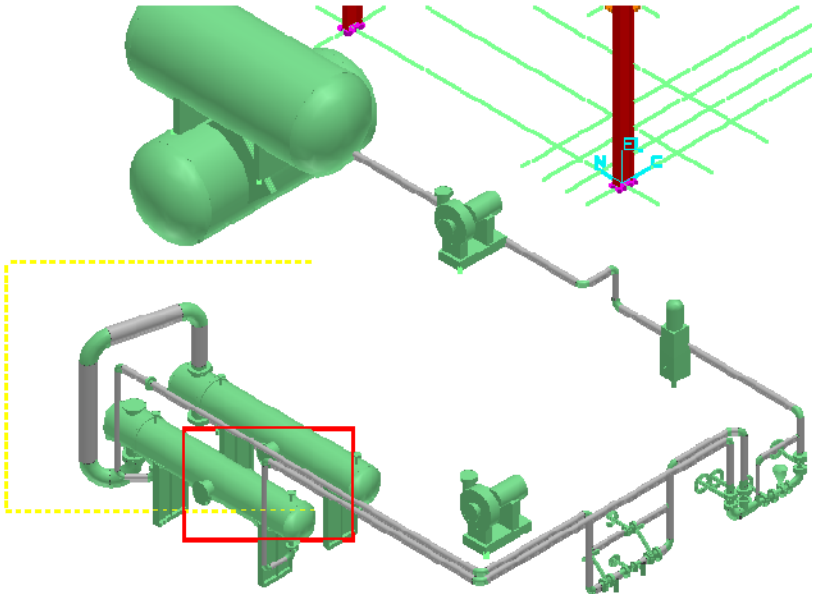


Figure 12: Overlapping Fence

6. After creating the fence and releasing it, Pipelines **403-P**, **402-P**, and **401-P** would be selected.

Steps for Selecting Objects by Key-in of Name:

Select Pipeline **400-P** of Unit **U04** by key-in of the name. After selection of the pipeline, the view should look like Figure 1.

1. Define your workspace to display Unit **U04** and Coordinate System **U04 CS**.
2. Select the any task environment.
3. Click the **Select** button on the vertical toolbar.
4. Select the filter **ALL** in the **Locate Filter** drop-down list.
5. Key-in **400-P** in the name field of the WorkSpace Explorer as shown in Figure 13 and Enter.

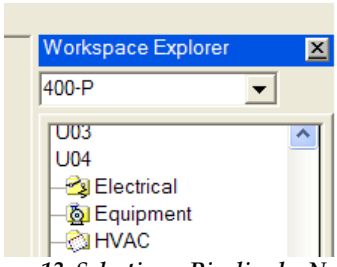


Figure 13: Selecting a Pipeline by Name

Tips:

- The select command Locate Filter also applies to selection by name.

- You can select multiple objects by name by entering "*" as a wildcard in the name search. For example, ***P** would select all objects with names ending in P.

Steps for Selecting Objects by Using Select by Filter:

Select all the structural objects in Unit **U01**, Area **A2**, of your workspace by using an object type filter. After selecting the specified objects, the view should look like Figure 14.

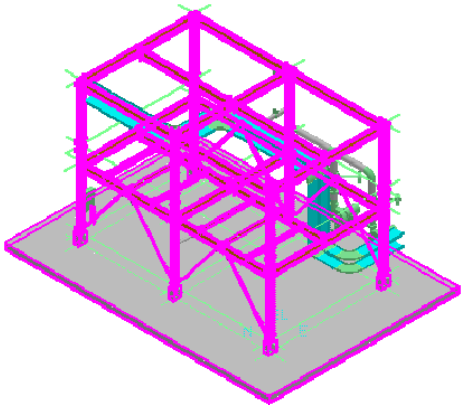


Figure 14: Selecting a Structural Object by Using an Object Type Filter

1. Define your workspace to display **A2**, **U01**.
2. Select the **Tools > Select by Filter** command.

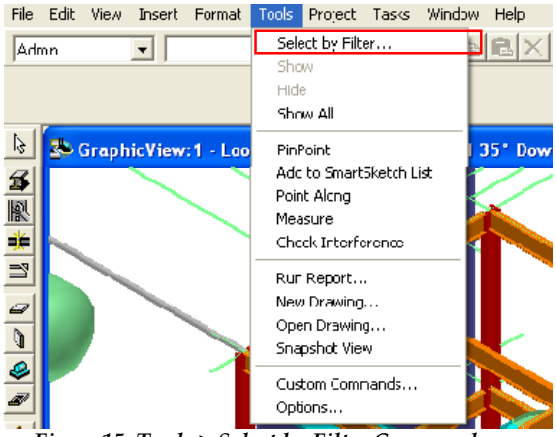


Figure 15: Tools > Select by Filter Command

The **Select by Filter** command opens the **Select Filter** dialog box where you can select one or more existing filters or create a new filter to use for the selection.

Tip:

- When the **Filter Selection** dialog box opens, a tree view lists the folders of the

available filters. When you double-click a folder, the view expands to show the next level of the folder organization. Double-click a filter to use it.

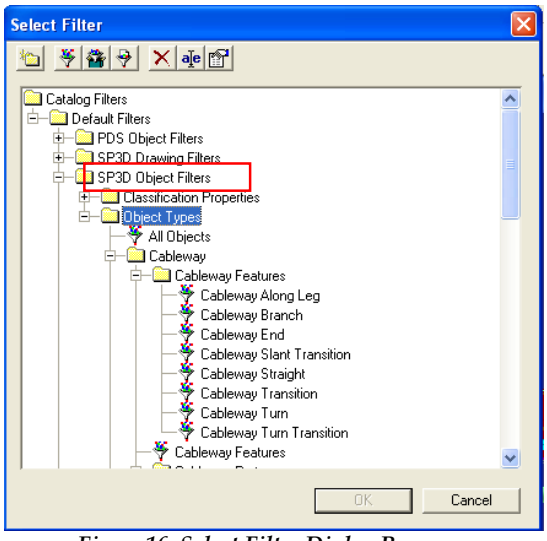


Figure 16: Select Filter Dialog Box

3. In the **Select Filter** dialog box, select **Catalog Filters**, expand **Default Filters > SP3D Object Filters > Object Types** and then, select the **Structure** filter.

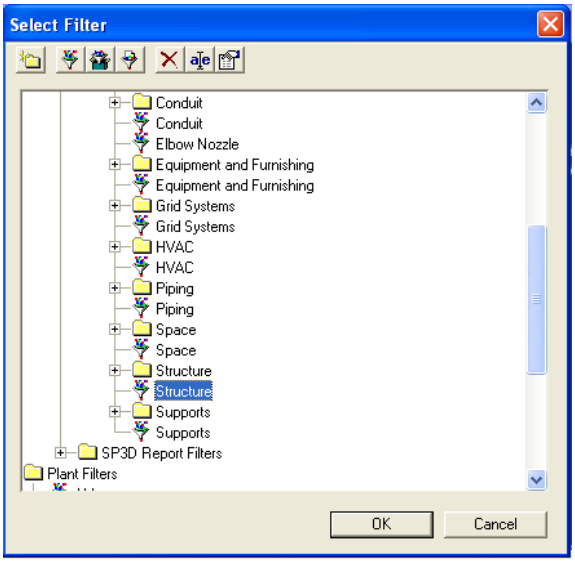


Figure 17: Structure Filter in the Select Filter Dialog Box

Tip:

- You can include more than one filter by pressing the CTRL key and clicking more filters. This will find objects that meet any of the filters selected (the union of all filters selected).

4. Click **OK** to select all structural objects in the current workspace.

For more information related to selecting objects by an object type filter, Locate Filter, and inside/overlapping fence, refer to the following topics of the user guide *CommonUsersGuide.pdf*:

- *Using Filters: An Overview*
- *Filter Properties Dialog Box*
- *Using the Tools Menu: An Overview*

Session 6: Applying Surface Style Rules

Objective:

By the end of this session, you will be able to:

- Apply surface style rules to set the color and surface transparency of the objects displayed in your workspace.

Prerequisite Sessions:

- SP3D Overview
- Creating a Session File
- Defining a Workspace
- Manipulating Views
- Selecting Objects

Overview:

The surface style rule lets you to use color to distinguish objects for your individual modeling or design purposes. For example, you can display objects that are on the **To Do List** in red. You can show objects that have the **approved** status in one color and that have the **rejected** status in another color. You can show a hot service pipe in one color and a cold service pipe in another color. You can elect to show piping insulation and slabs in transparent colors so you can see the objects behind them. You can also define style rules that show objects in their approximate actual physical color if you are reviewing the model with a customer.

A surface style rule is the combination of a filter that identifies a set of objects by specific criteria and an associated surface style. You can select any number of surface style rules to apply to the objects displayed in your session. Surface style rules work like a coin changer. An object is checked to see if it meets the filter criteria of the first surface style rule in the list of selected surface style rules. If it does, the associated surface style is applied to that object. If it does not, the next rule in the list is checked. If an object does not meet the criteria of any of the surface style rules, it is displayed in the system-defined default style.

Surface style rules are applied to objects when you:

- Place a new object.
- Edit an existing object.
- Refresh the workspace.
- Select a rule in the **Workspace** list, and then click **Apply** in the **Surface Style Rules** dialog box.

The surface style rules you select are stored in your session file and can be included in your templates.

Tips:

- Surface style rules can impact the display and update performance of SP3D

significantly. You should use only a few rules during normal design, but may use a number of rules to create visualizations for engineering and client reviews.

- To improve performance, use filters based on object type and properties, and limit the use of filters based on the System hierarchy.

Steps for Creating a Surface Style Rule

Create and apply a surface style rule to change the color of equipment objects to yellow in Unit **U04** of your workspace. After applying the surface style rule, the view should resemble the highlighted area in Figure 1.

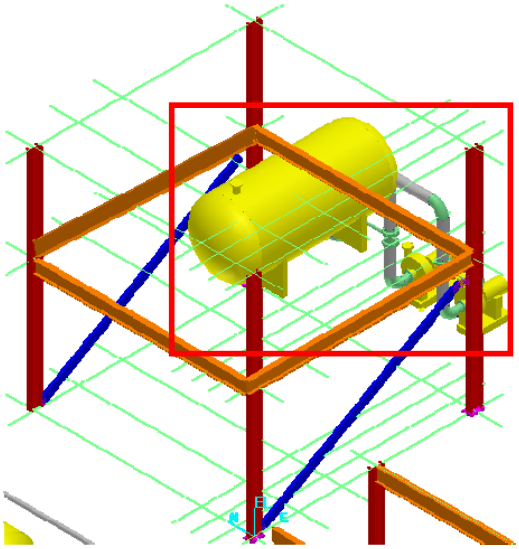


Figure 1: Output: After Applying Surface Style Rule to Equipment Objects

1. Define your workspace to display Unit **U04** and Coordinate System **U04 CS**.

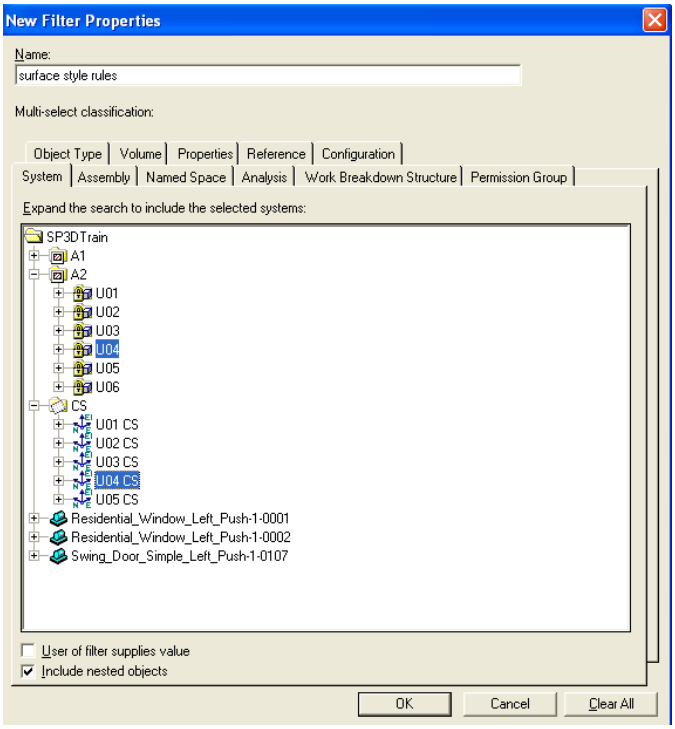


Figure 2: Defining a Workspace

2. Select the **Format > Surface Style Rules** command.

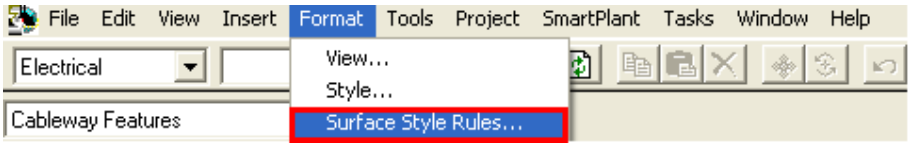


Figure 3: Surface Style Rule Command on Format Menu

3. The **Surface Style Rules** dialog box appears. It contains the surface style rules defined for the plant model. Click **New** to define a new rule.

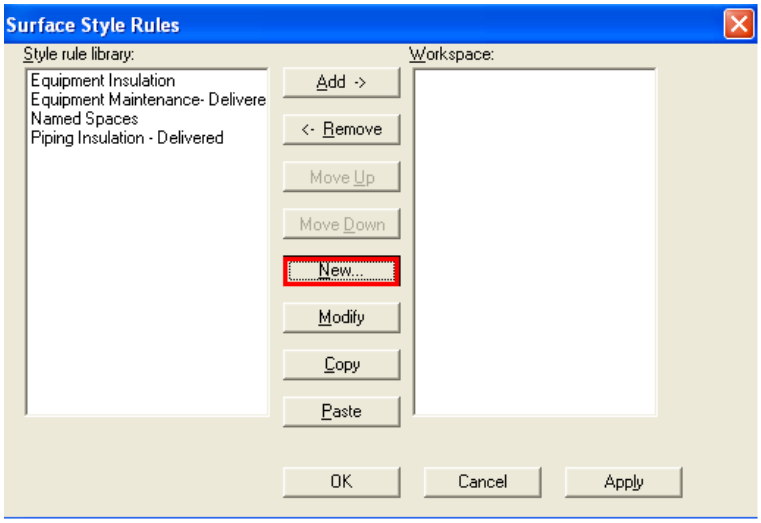


Figure 4: Surface Style Rules Dialog Box

4. The **Surface Style Rules Properties** dialog box appears. It contains fields to define the rule name, filter, the applied style, and the geometric aspects of the object to which the style is applied. Type **All Equipment objects** in the **Rule name** box as shown in Figure 5.

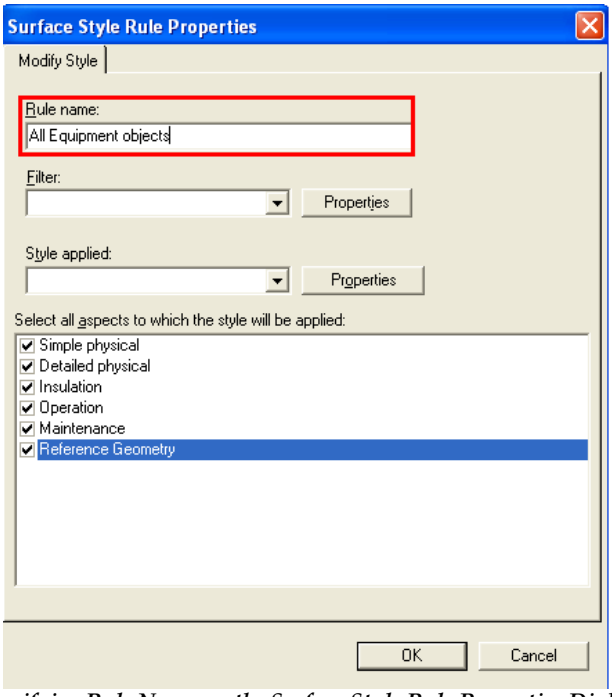


Figure 5: Specifying Rule Name on the Surface Style Rule Properties Dialog Box

- In the next step, you select a filter to identify the objects to which a surface

style will be applied. You can select a filter from the drop-down list or previously selected filters. displays the list of filters previously selected while creating surface style rules in the current session file.

- Select another existing filter using the **More…** option. This option displays the Select Filter dialog.
- Create a new filter using the **Create New Filter…** option. This option displays the **New Filter Properties** dialog box to define a new filter.
- displays the **Select Filter** dialog box to select a filter from the existing filters.
- The **Properties** button next to the **Filter** drop-down list displays the properties of the selected filter.

5. Click the **Filter** drop-down list and select the **More…** option to specify the filter as shown in Figure 6.

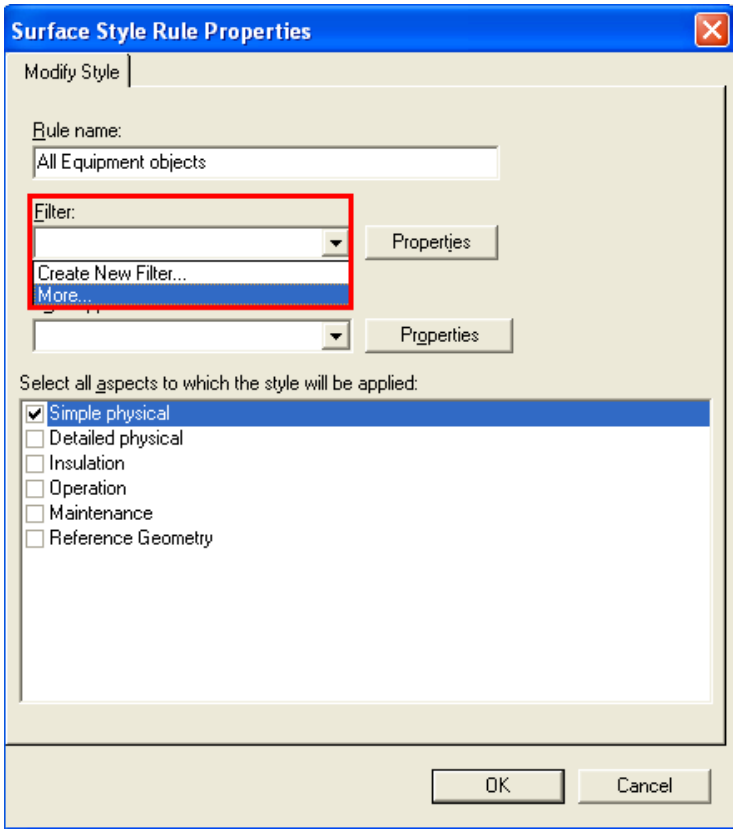


Figure 6: Selecting the Filter

6. Under **Catalog Filters**, expand **Default Filters > SP3D Object Filters > Object Types** and select the **Equipment and Furnishing** filter. Click **OK** to go back to the **Surface Style Rules Properties** dialog box.

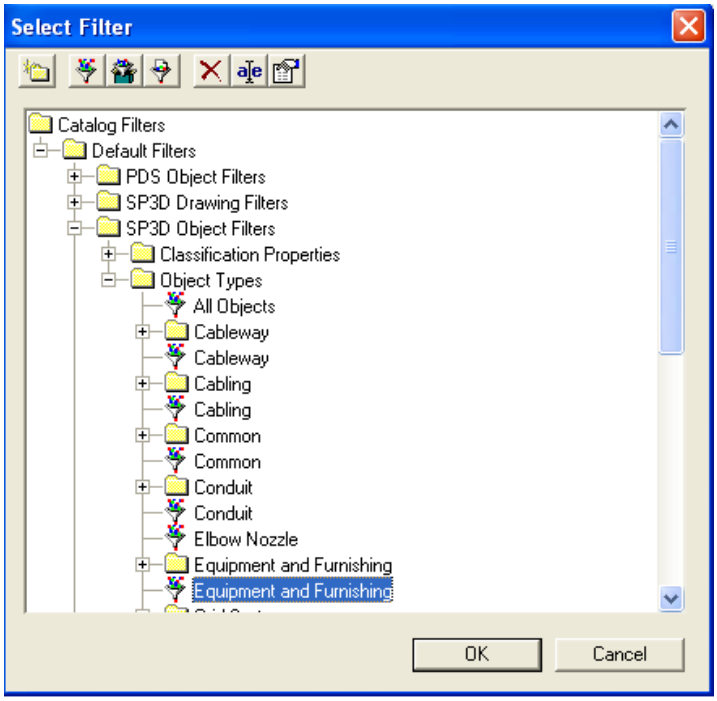


Figure 7: Selecting the Filter

Now, you will select the surface style that you want to apply to the objects selected by the filter.

- The **Style applied** drop-down list specifies the surface style. The list contains all surface styles available in the model database.
- The **Properties** button next to this drop-down list allows you to edit the selected style.

Tip:

- You can modify the existing styles by clicking the **Format > Style** command**.**
- While anyone can currently edit the surface styles (change the definition of colors), the administrator should be the only one who exercises this function. Access control will be added to the surface styles in a future release.

7. Select **Yellow** in the **Style applied** drop-down list of the **Surface Style Rules Properties** dialog box.

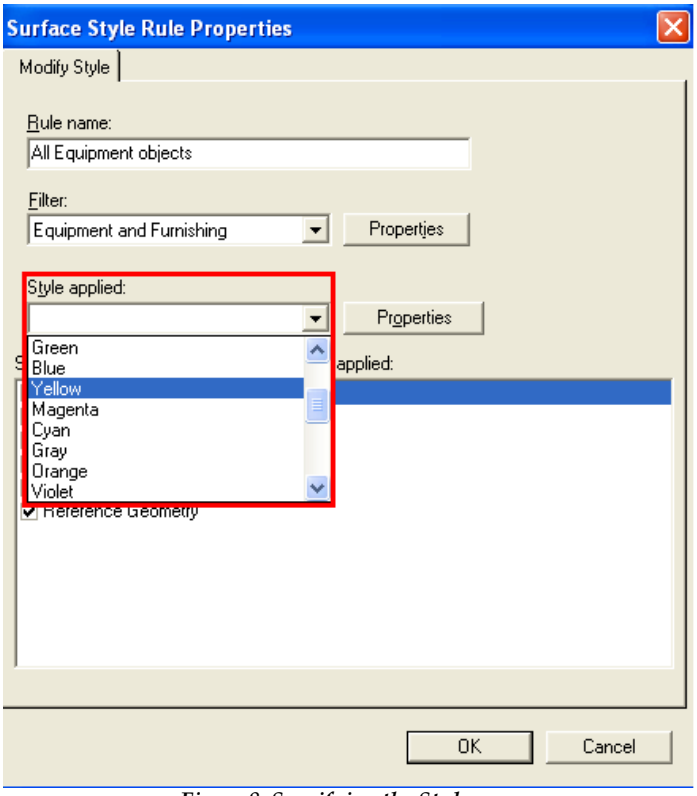


Figure 8: Specifying the Style

In the next step, you to select the aspects of the design objects to which the style rule applies. An aspect is an alternate geometry for a design object. An object can have one or more aspects (see list below). For Example, a surface style rule can be defined to show Piping insulation in a translucent surface style so you can see the pipe's simple physical aspect.

The aspects to which you can apply the selected surface style rule are:

- The **Simple Physical** aspect is the physical geometry of the object that is suitable for visual recognition and interference detection purposes.
- The **Detailed Physical** aspect is a more detailed model of the physical geometry. Currently, the design objects delivered by Intergraph do not use this aspect.
- The **Insulation** aspect is the insulation geometry.
- The **Operation** aspect is the space around an object required for its operation.
- The **Maintenance** aspect is the space around an object required to perform maintenance on the object.
- The **Reference Geometry** aspect is geometry used for design reference only and does not participate in interference checking.

8. In the **Select all aspects to which the style will be applied** list, clear all check boxes

except the **Simple Physical** check box. Click **OK**.

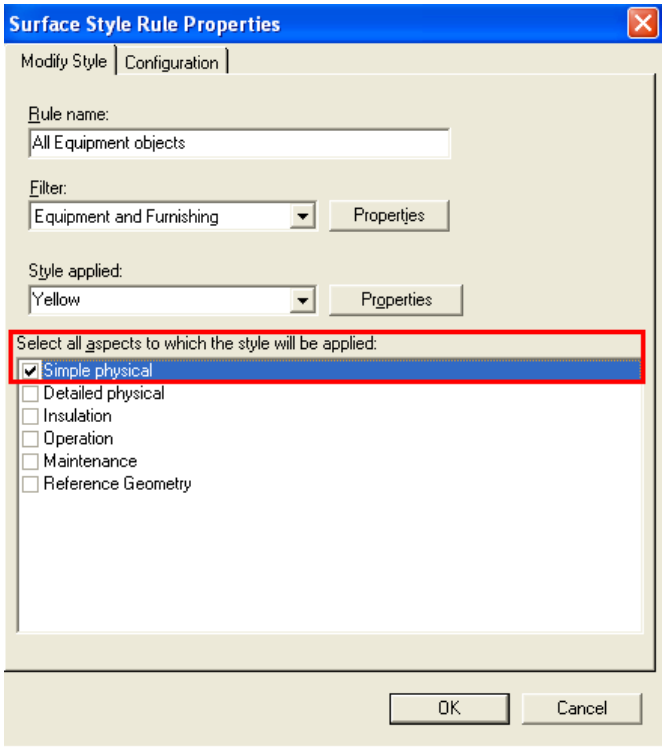


Figure 9: Specifying the Aspect

Notes:

- The system-default style is used to display all aspects of an object whose style you have not defined with a style rule.
- When you create a surface style rule, the rule is assigned to your active permission group.

9. To apply the rule, select the **All Equipment Objects** rule in the **Style Rule Library** and click **Add**.

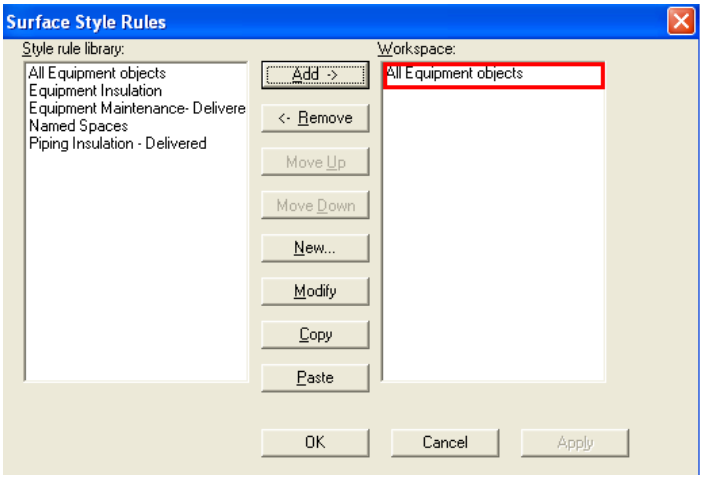


Figure 10: Added the Surface Style Rule to the Workspace

10. Click **OK**.

Tips:

- When an object is created or edited, it is checked against the filters of the surface style rules starting from the top of the list. When the object meets a filter, it is assigned that style and is not checked against the style rules further down in the list. For greatest efficiency, you should arrange the rules such that the rules appearing at the top of the list apply to most of the objects. Click **Move Up** and **Move Down** to change the order of the rules in the **Workspace** list.

For more information related to applying surface style rules, refer to the *Using Surface Style Rules: An Overview* topic in the user guide *CommonUsersGuide.pdf.*

Session 7: PinPoint Ribbon

Objective:

By the end of this session, you will be able to:

- Use **PinPoint** to enter precision points whenever commands prompt for a point.

Pre-Requisite Sessions:

- SP3D Overview
- Creating a Session File
- Defining a Workspace
- Manipulating Views

Overview:

The **PinPoint** ribbon (Figure 1) appears when you click the **PinPoint** button on the Common toolbar or the **PinPoint** command in the **Tools** menu. The PinPoint controls are active when any command prompts you to enter a point. The PinPoint options allow you to enter precision coordinates relative to the selected coordinate system and precision coordinate offsets relative to any point in the model.

You can change the position of the origin used for the coordinate readouts by clicking the **Reposition Target** option on the ribbon and then clicking a new position in the active view. The target position can be defined by entering the coordinates. It can also be defined at a precision point found on existing geometry by using the SmartSketch3D feedback. (Refer to the tutorial for SP3D Common Session 9: *Using SmartSketch Points*). As you move the cursor around, **PinPoint** displays the distances between the cursor and the target position. The target is your way of entering Delta positions.

You can lock one or more coordinate positions by entering the coordinates in the fields. You can also lock the coordinates at the current cursor position by using the F6, F7, and F8 function keys (Easting {x}, Northing {y}, and Elevation {z}, respectively). The second method is very useful in conjunction with the SmartSketch3D precision point feedback. You can position your cursor so that it finds, for example, the axis of a pipe and then enter F8 to lock just the elevation coordinate to the elevation of the pipe. As you move your cursor, only the Easting and Northing will change.

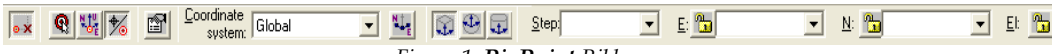


*Figure 1: **PinPoint** Ribbon*

Notes:

- The coordinate system selected on the **PinPoint** ribbon is called the active coordinate system. It is used by all commands that have functions that reference a coordinate system.
- You can choose to work with rectangular, spherical, or cylindrical coordinates on the PinPoint ribbon.
- You can define a temporary coordinate system with an origin and axis directions different from those of the global coordinate system or the available auxiliary coordinate systems you have defined in the Grids task.

The options available on the **PinPoint** ribbon are:

- **Display On/Off** - Displays or hides the **PinPoint** distance readout near the cursor in the active view. The text is displayed in the selected elements color defined on the Options dialog box, which is displayed using the **Tools > Options** command.

- **Set Target to Origin** - Moves the target to the origin of the active coordinate system

Note:

- If a different coordinate system is selected in the **Coordinate System** box, you must click **Set Target to Origin** to reset the target to the origin of the selected coordinate system.

- **Relative Tracking –** This mode moves the target to the last point you entered.

- **Coordinate system** - Sets the active coordinate system. You can select the global coordinate system from the drop-down list. Or, you can select a coordinate system in a graphic view or from the System hierarchy.

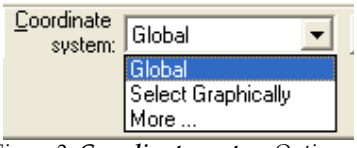


Figure 2: ***Coordinate system*** *Options*

The **Coordinate system** drop-down list shows the last seven coordinate systems that were selected in a session in addition to the three default options seen in Figure 3. These options are as follows:

- **Global -** Specifies the current **Global** coordinate system
- **Select Graphically –** Allows you to graphically select a displayed

coordinate system or to select any design object that has a local coordinate system such as equipment and nozzles

- **More** - Displays the **Select Coordinate System** dialog box (Figure 3) where you can select the coordinate systems by browsing to it in the system hierarchy. The **Workspace** option shows only the coordinate systems in the current workspace. The **Database** option shows all coordinate systems defined in the model database.

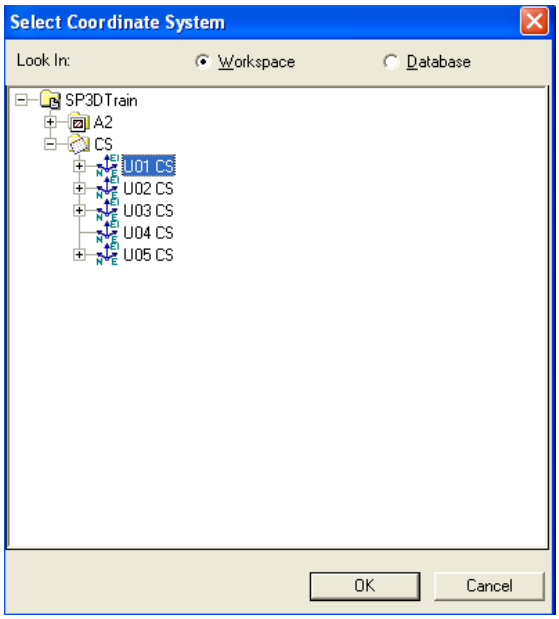


*Figure 3: **Select Coordinate System** Dialog Box*

- **Properties of Active Coordinate System** - Shows the properties of the active coordinate system selected in the **Coordinate system** drop-down list (Figure 4).

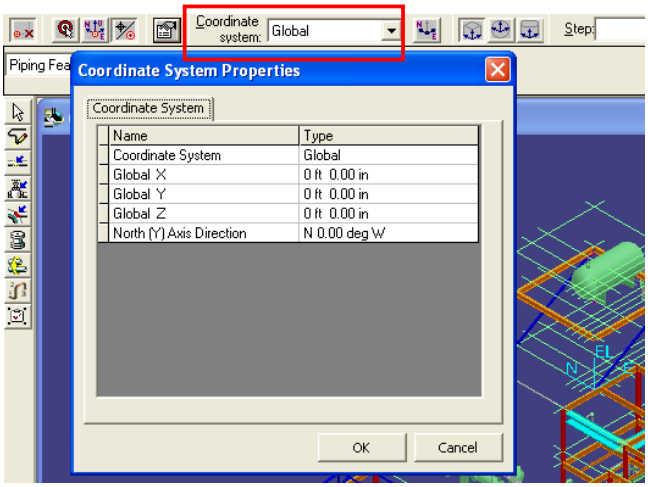


Figure 4: Coordinate System Properties

- **Define Coordinate System by 3 Points** - Defines a temporary coordinate system by three points. Point 1 defines the origin of the coordinate system. Point 2 defines the end of the local E(X)-axis. Point 3 defines the end of the local N(Y)-axis.

- **Rectangular Coordinates** – Sets the readout to the rectangular coordinates mode. The following controls appear with this coordinate mode:

  - **Step –** Forces the coordinates to snap to multiples of the displayed step value. To disable this behavior, you delete the value displayed in this field.
  - **E –** East or x-axis coordinate
  - **N –** North or y-axis coordinate
  - **EL –** Elevation or z-axis coordinate

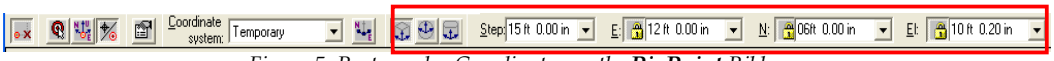


*Figure 5: Rectangular Coordinates on the **PinPoint** Ribbon*

- **Locking and Unlocking Values:** This toggle button locks or unlocks coordinates. The current state of the button shows whether or not the coordinate is locked.

  Tips:

  - The **F6, F7**, and **F8** function keys toggle the coordinate lock for the E, N, EL coordinates, respectively. When you toggle to lock, the current cursor coordinate is entered as the locked value and the keyboard focus is set to that coordinate on the pinpoint ribbon.
  - The function keys are the recommended way to put the keyboard focus in the desired coordinate field to enter a coordinate rather than clicking in the coordinate field. This gets you in a habit that works whether you want to use the current smartsketch point coordinate value or if you want to enter a different coordinate value.

- **Spherical Coordinates –** Sets the readout to the spherical coordinates mode. The following controls appear with this coordinate mode.
  - **Distance –** Radial distance coordinate (should be called radius)
  - **Horizontal -** Horizontal angle coordinate as measured clockwise from the N-axis in the horizontal plane to the radial axis

- **Vertical -** Vertical angle coordinate as measured up from the horizontal plane to the radial axis

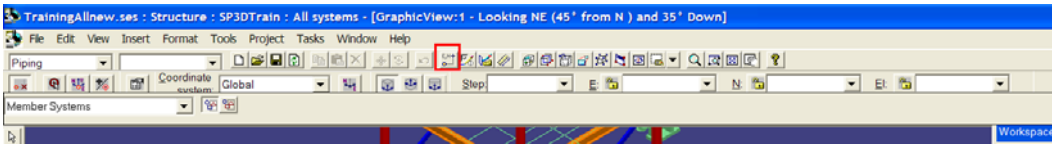
Tip:

- When you use spherical coordinates and lock the distance, you must also lock at least one of the angle boxes on the ribbon. You cannot unlock an angle while the absolute distance is locked and no other angle is locked.

- **Cylindrical Coordinates -** Specifies the cylindrical coordinates mode for the **PinPoint** command. The following corresponding settings appear with this coordinate mode:
  - **Radius –** Radial distance coordinate
  - **Theta -** Horizontal angle coordinate as measured clockwise from the N-axis in the horizontal plane to the radial axis
  - **Z –** Elevation or z-axis coordinates

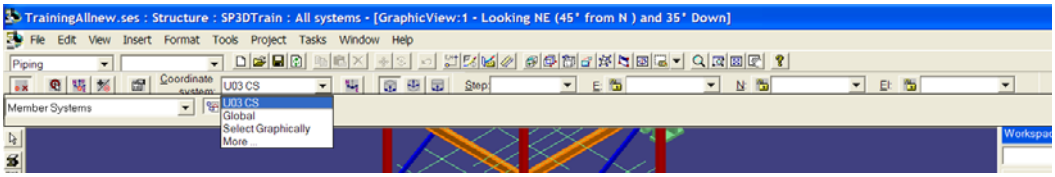
Practice Using Precision Point Tools:
When a command is in a smartstep prompting you to enter a point, you can enter the exact point you want with the aid of **PinPoint** in conjunction with **Smartsketch** precision point hot spots on graphic objects. The **PinPoint** tool allows you to view and enter coordinates relative to the coordinate system of your choice.

1. Define a workspace having all systems of the training plant. Activate the PinPoint ribbon by picking the PinPoint button on the horizontal tool bar.



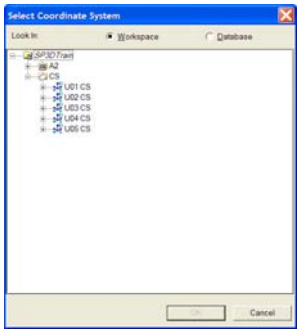
The PinPoint ribbon displays the coordinate system you selected for input/output of coordinates. This coordinate system is called the **Active Coordinate System**. All commands that need a coordinate system for orientation use this coordinate system.

2. Click on the **Coordinate System** combo to see the coordinate systems available for selection and then pick the "**More…**" option



The last few coordinate systems you selected will display in the list along with the options: **Global, Select Graphically, and More**…

All coordinates are stored in the database relative to the native **Global** coordinate system. You cannot edit this coordinate system. You can create other coordinate systems in the Grids Environment to use as convenient references for modeling different portions of the plant. These coordinate systems are available for selection in the Coordinate System combo or selection from the **More…** option.

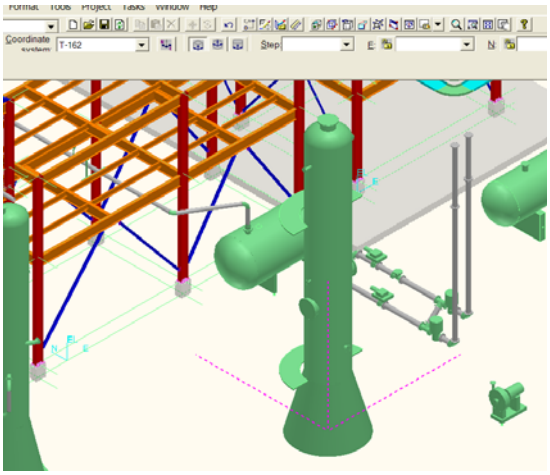


You can navigate the System hierarchy to locate a coordinate system currently included in your workspace or you can pick the **Database** option to allow you to browse the entire model database to select the coordinate system you want.

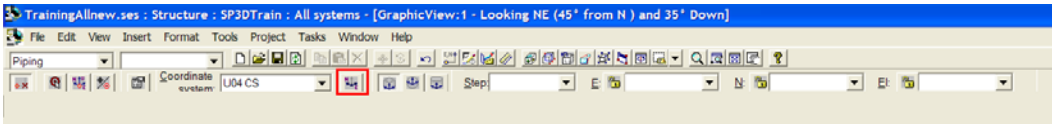
3. Select **U04 CS** with a double-click on the coordinate system name. Or, select and pick OK.

The "**Select Graphically**" option prompts you to select an object. You can select a coordinate system by pointing at the coordinate system graphic or you can select any object that has a local coordinate system, for example, equipment or a structure member.

4. Pick "**Select graphically**" and select a structure member and then select equipment T-162.



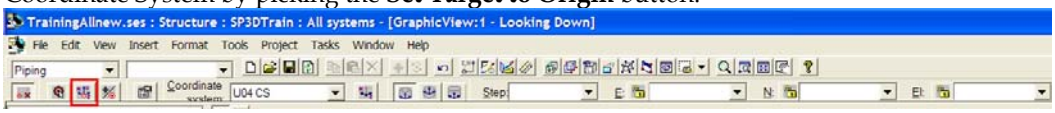
You can set the active coordinate system to a temporary definition using the **Define Coordinate System by Three Points** button.



You will be prompted to select the origin, east (x-axis) and north (y-axis) direction. Experiment with this on your own.

5. Pick the Coordinate System combo drop-down and pick **U04 CS** from the list.

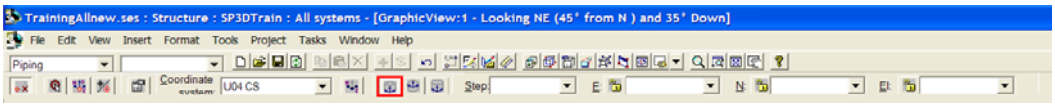
Once you have selected the Active Coordinate System, you must define the origin from which the displayed coordinates are measured. This temporary origin is called the **Target**. The Target allows you to view and enter coordinates relative to any point in the model. When you change the Active Coordinate System, the position of the Target in the model remains where it was. You can choose to set the Target to the origin of the Active Coordinate System by picking the **Set Target to Origin** button.



6. Pick the **Set Target to Origin** button.

You select the coordinates you see on the PinPoint ribbon by picking, **Rectangular, Spherical, or Cylindrical Coordinates** mode.

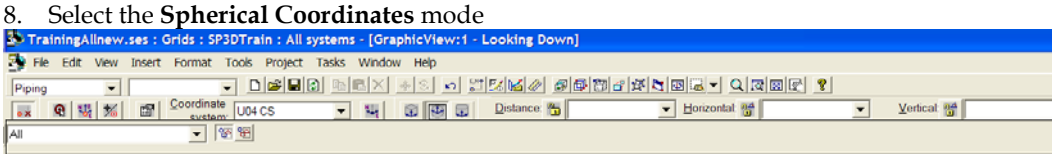
7. Select the **Rectangular Coordinates** mode



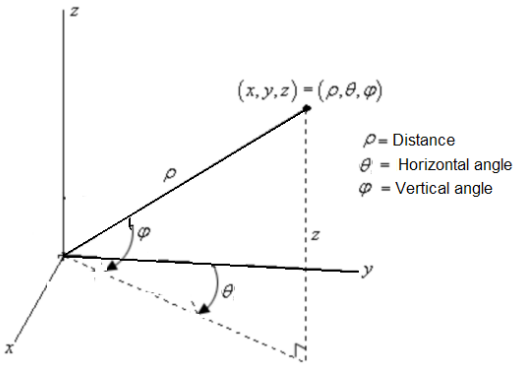
You see the rectangular coordinates; Easting (x), Northing (y), and Elevation (z) fields that display the coordinates of the current cursor position (measured from the Target) when you are in a command step prompting you to enter a point.

The **Step** field and **lock** buttons will be described later.

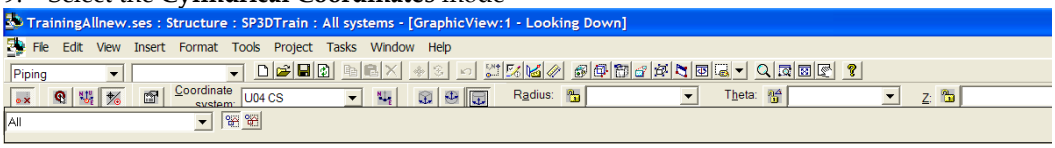
8. Select the **Spherical Coordinates** mode



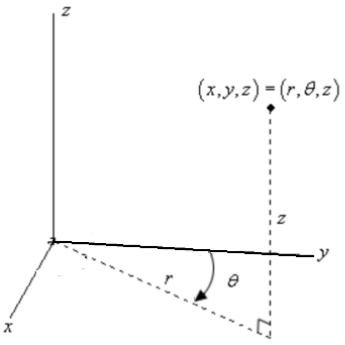
You see the spherical coordinates; **Distance** (should be called "radius"), **Horizontal** angle, and **Vertical** angle. The Distance is the length from the Target to the cursor point. The horizontal angle is measured from the north (y) axis to the projection of the line from the Target to the cursor point on the plan plane. The vertical angle is measured from the plan plane to the line from the Target to the cursor point.



9. Select the **Cylindrical Coordinates** mode

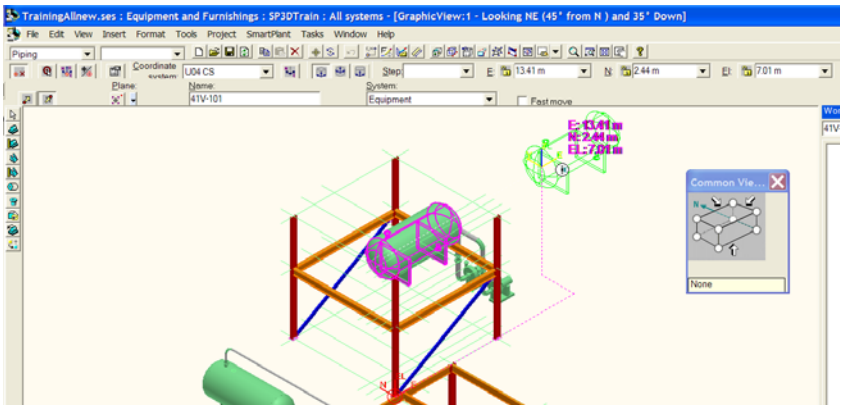


You see the cylindrical coordinates; **Radius**, **Theta**, and **Z**. The Radius is measured from Target to projection of the cursor point on the plan plane. Theta is measured from the north (y) axis to the projection on the plan plane of the line from the Target to the cursor point. Z is measured from the plan plane vertically to the cursor point.



10. Reselect the Rectangular Coordinates mode. In the Equipment Environment, select equipment 41V-101 and pick the **Move** command to enter into a command that prompts you to enter a point.

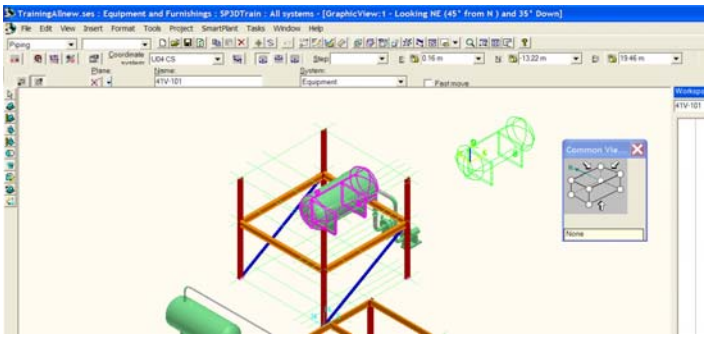
The Move command defaults the "from point" to the origin of the equipment and goes immediately into the "to point" step of the move.



Notice the lines going from the Target Location (sphere graphic) along the Active Coordinate System axes directions to the cursor point. You also see the coordinates displayed on the screen and in the fields on the ribbon bar.

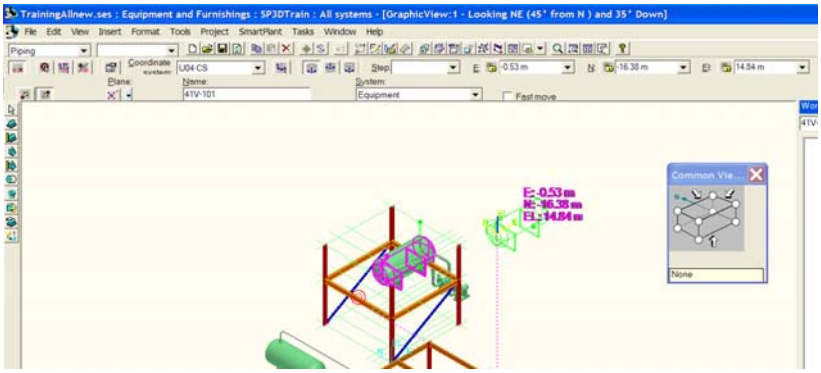
You can toggle the display of the pinpoint graphics in the window off and on by picking the **Display on/off** button (or **F9**).

11. Enter **F9** to toggle off the PinPoint graphics display in the window and then toggle the display back on.



You can set the Target location to get coordinates relative any point in the model. This is how you enter precision offsets from existing geometry.

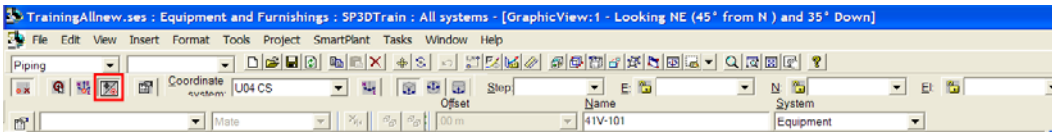
12. Pick the **Reset Target** button and select the midpoint of the indicated member by positioning your cursor near the evident midpoint. You will see a glyph display indicating a Divisor point. These keypoints will be discussed in detail in the "Using SmartSketch Points tutorial.



You can also quickly **Reset Target** by first positioning the cursor over the point you want and entering **F12**. This immediately resets the target to the current position of the cursor.

13. Click the "move from" button on the move ribbon to select a different reference point on the equipment. Position the cursor over the top nozzle of equipment 41V-101 and enter **F12** to set the Target to the nozzle point. Then, position the cursor over the nozzle and click to enter the nozzle point as the "move from" point.

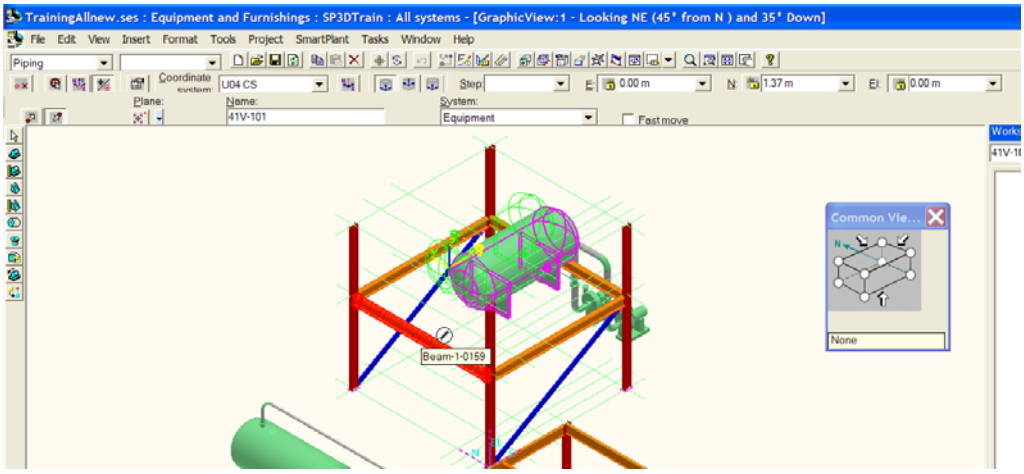
Optionally: Select the "**Relative Tracking**" mode on the PinPoint ribbon before you pick the "move from" button on the Move ribbon. This mode resets the Target automatically to the last point you enter.



You can enter coordinates by clicking in the desired coordinate field on the PinPoint ribbon or entering **F6, F7**, or **F8** to set the east, north, or elevation coordinates respectively. The function keys are the most efficient method. When the function key to fix a coordinate is pressed, the coordinate of the current cursor position is locked and the keyboard focus to set to that coordinate field. You can leave the coordinate at the indicated locked value or enter another coordinate.

The cursor point will be forced to the coordinates you enter or "lock". As soon as you enter the point with a click, all the coordinate locks will be automatically removed. You can also unlock the coordinates by clicking on the **Lock** toggle buttons or entering **F6, F7, or F8** when the coordinate is locked.

14. With equipment now in the "move to" step, click on the select lists for the east and elevation coordinates and select 0 to lock the movement to just along the north direction. Position the cursor over the midpoint of the indicated member and click to move the equipment so that the nozzle point is moved north middle of the pad.



15. Undo the move.

PinPoint works in a similar way using the coordinates of the Spherical and Cylindrical coordinate systems. You have seen how the PinPoint coordinate locks can be used in conjunction with smartsketch points to enter precision coordinates for points. The "Using SmartSketch Points" tutorial will show other useful scenarios.

For more information related to the **PinPoint** ribbon, refer to the *Using the Tools Menu: An Overview* topic of the user guide *CommonUsersGuide.pdf*.

Quiz:

1. Which types of coordinates can you define on the **PinPoint** ribbon?
2. Which of the following options can you use to change the target position:

- 
-
- 
- 

Session 8: Measurement Ribbon

Objective:

By the end of this session, you will be able to:

- Measure distance, radius, and angle.

Prerequisite Sessions:

- SP3D Overview
- Creating a Session File
- Defining a Workspace
- Manipulating Views

Overview:

The **Measurement** ribbon (refer to Figure 1) appears when you select the **Measure** command in the **Tools** menu or from the **Common Toolbar**. The measurements are temporary and are not stored in the model. You can specify the units of measurement by clicking the **Tools > Options** command and selecting the **Units of Measure** tab**.**

The **Tools > Measure** command can:

- Measure the distance between two points.
- Measure the minimum distance between two objects.
- Display the diameter and radius of holes and fillets.
- Determine the angle between two lines or surfaces.
- Sum repeated measurements and display the cumulative results on the ribbon.

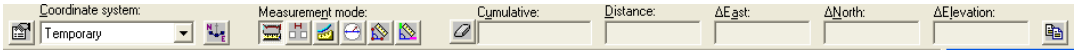


Figure 1: Measurement Ribbon

Options on the Measurement Ribbon:

The options on the **Measurement** ribbon are described below:

- **Coordinate System Properties**—Displays the properties of the active coordinate system

- **Coordinate system**—Sets the **Active Coordinate System**. The active coordinate system can also be changed on the **PinPoint** ribbon bar. Measurements are relative to the active coordinate system.

- **Define Coordinate System by 3 Points**—Defines a temporary coordinate system by three points. Point 1 defines the origin of the coordinate system, Point 2 defines the end of the local E axis, and Point 3 defines the end of the local N axis.

- **Measurement Modes:**
  - **Measure Distance Between 2 Points**—Measures the linear distance between two points.
  - **Measure Minimum Distance**—Measures the shortest distance between two objects.
  - **Measure Distance Along Element**—Measures the distance between two points along the axis of a selected linear element. Points are projected to the axis.
  - **Measure Radius and Diameter**— Measures the radius or diameter of cylindrical arcs or surfaces such as holes, fillets, or cylinders.
  - **Measure Angle Between 3 Points**—Measures the angle between three points. The second point is the vertex.
  - **Measure Angle Between Objects**—Measures the angle between planar surfaces and/or the axis of cylinders.
- **Clear**—Resets the value in the **Cumulative** box to zero.
- **Cumulative**—Sums repeated measurements until it is cleared or you switch between the angle and distance modes of measurement.
- **Copy Measurement**—Copies the measurement values from the ribbon to the clipboard.

The remaining controls on the ribbon depend on the angle or distance measurement mode selected:

- **Measuring the Distance**—While measuring linear distances, the following options appear on the **Measurement** ribbon.
  - **Distance**—Displays the distance between two points.
  - **Δ East**—Displays the distance along the E-axis.
  - **Δ North**—Displays the distance along the N-axis.
  - **Δ Elevation**—Displays the distance along the EL-axis.

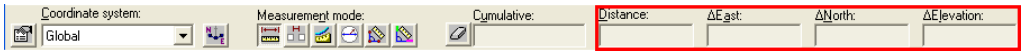


Figure 2: Fields for Distance Measurement

Examples showing the use of the linear distance measurement:

- **Measure Minimum Distance**: Use the **Measure Minimum Distance** command from the **Measurement** ribbon to measure the clearance between, for example, the pump and the column. Locate the pump and the column. The minimum distance between the displayed geometric aspects of the two objects and the distance along the active coordinate system axes will be displayed on the **Measurement** ribbon as well as on the graphic view, as shown in Figure 3.

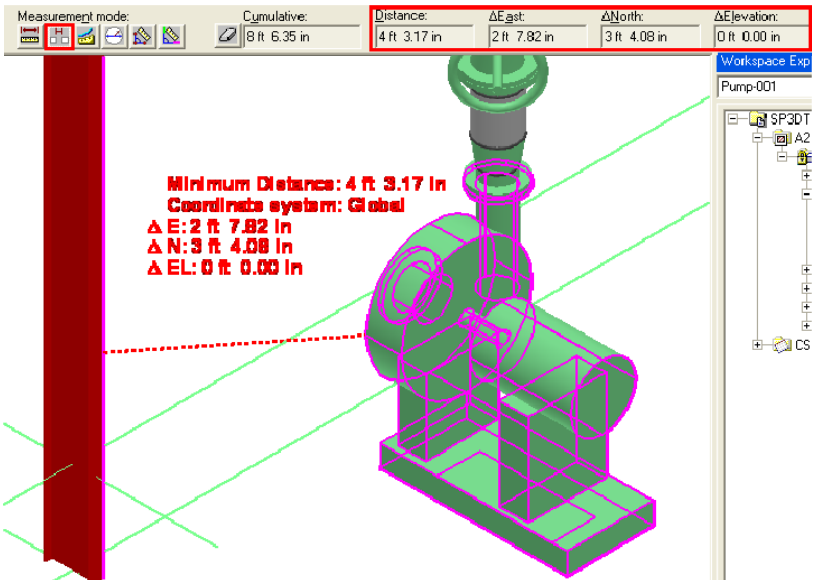


Figure 3:Minimum Distance Measured Between a Column and Pump

- **Measure Distance Between 2 Points:** Use the **Measure Distance Between 2 Points** command from the **Measurement** ribbon, to measure distance between, for example, the tank discharge centerlines. Locate the two nozzles of the tank. The absolute distance between the two points and the distance along the active coordinate system axes will be displayed on the **Measurement** ribbon as well as on the graphic view, as shown in figure 4.

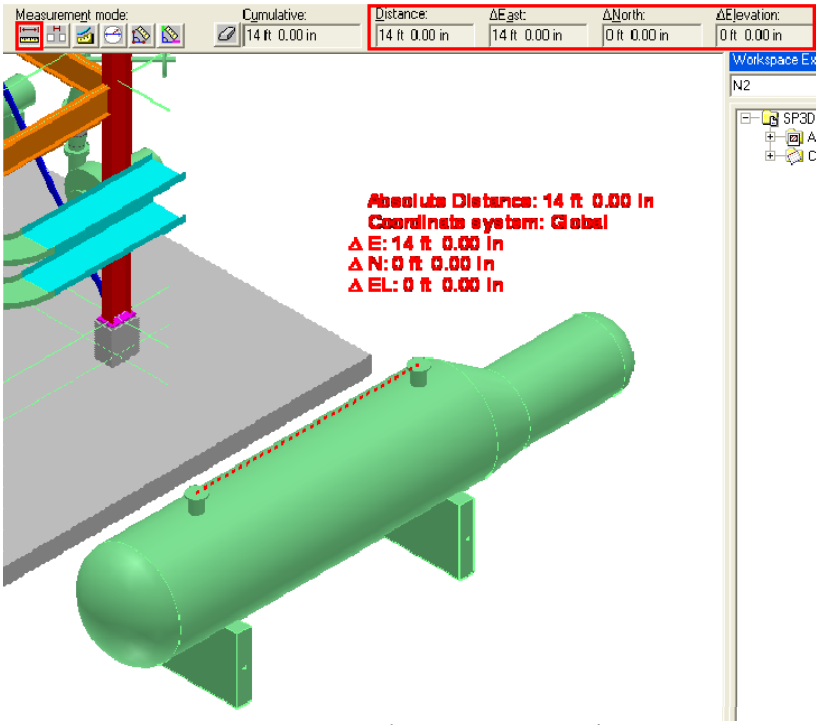


Figure 4: Distance Measured Between Two Nozzles

- **Measuring the Radius or Diameter:** While measuring radius or diameter, the following options appear on the **Measurement** ribbon.

  - **Radius—**Displays the radius of a hole or a fillet
  - **Diameter—**Displays the diameter of a hole

Figure 5: Fields for Radius or Diameter Measurement

- **Measuring Angles:** While measuring angles, the following options appear on the **Measurement** ribbon.

  - **Angle—**Displays the angle between specific points or surfaces in a model.
  - **Apparent angle—**Measures the angle of the lines projected on the active view plane. If lines intersect and are coplanar, then both the **Angle** and **Apparent angle** boxes display values. If the lines do not intersect and are not coplanar, then only the **Apparent angle** box displays a value.

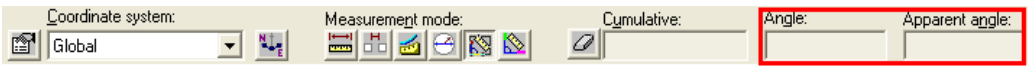


Figure 6: Fields for Angle Measurement

Examples showing the use of angle measurement:

- **Measure Angle Between Objects:** Use the **Measure Angle Between 2 Objects** command to measure the angle between, for example, a column and a vertical corner brace. Locate the column and the brace. If the infinite extension of lines (or planes) defined by the selected objects intersect, the **Angle** between them is displayed on the **Measurement** ribbon. If they do not intersect, only the **Apparent Angle** as found by the projection to the view plane is displayed.

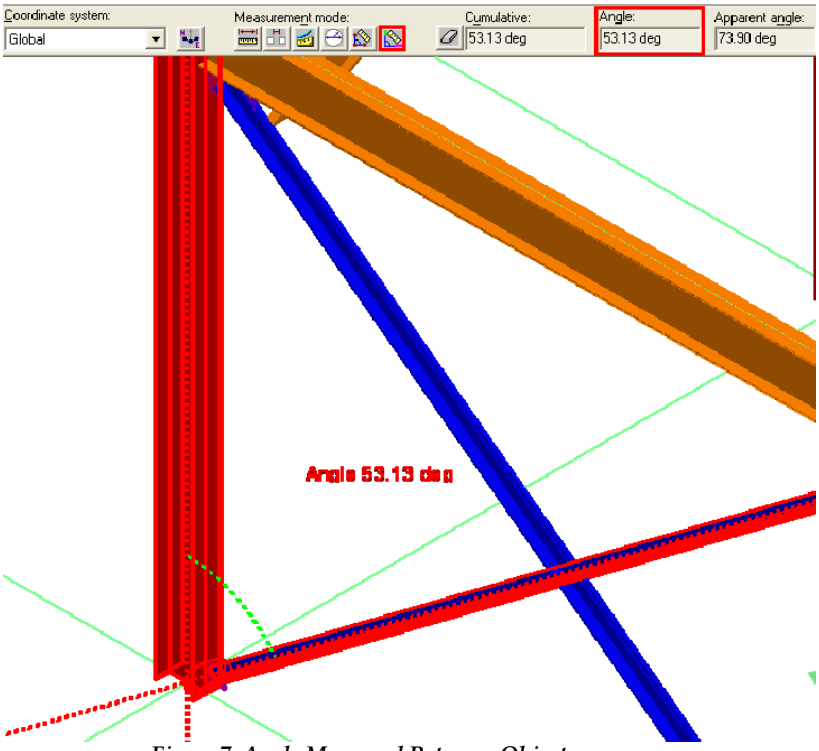


Figure 7: Angle Measured Between Objects

- **Measure Angle Between 3 Points:** Use the **Measure Angle Between 3 Points** command to measure a specific angle you can define by selecting three points. Locate the first point, second point (vertex), and third point. The **Angle** defined by the three points is displayed on the **Measurement** ribbon.

For more information related to Measurement ribbon, refer to the *Using the Tools Menu: An Overview* topic of the user guide *CommonUsersGuide.pdf*.

Quiz:

1. How do you define a temporary coordinate system?
2. What options change on the **Measurement** ribbon if you select different measurement

modes?

3. Which measurement options will you use to measure the shortest distance between two objects?

Session 9: Using Smartsketch Points

Objective:

By the end of this session, you will be able to:

- Use the SmartSketch options to define which precision points will be located.
- Use SmartSketch points in conjunction with PinPoint to enter precision points.
- Use the **Add to SmartSketch List** ribbon options to make locating smartsketch points in dense models easier.

Prerequisite Sessions:

- SP3D Overview
- Creating a Session File
- Defining a Workspace

Overview:

When any command is in a smartstep prompting you to select a point, the software activates the SmartSketch point location function. When your cursor is close to a SmartSketch point, the geometric objects providing the precision are highlighted and a small glyph near the cursor displays the specific type of precision point found. When you click to enter the point, the coordinates of that precision point, the type precision point, and the object(s) providing the point are supplied to the command asking for the point. Many commands will create associative point relationships that maintain the point at the precision point location when the geometric objects are modified.

Tip:

- Understanding when associative points are established is important to understand the modification behavior of the different design objects. The tutorials will tell you which commands create the associative point relationships. Better ways to show you that the associative point relationships exist should be provided in the future.

The basic types of SmartSketch points are:

- Key points on connection ports – the connection point on piping nozzle, conduit, HVAC, and foundation ports.
- Key points on geometry—end point, divisor point, point along linear object, and center of circle.
- Intersection points—intersection point of two linear objects or a point along the intersection of two planes.
- Linear relationship point—when command prompts for two points, the second point can locate the points that make the line between the first and second points

parallel or perpendicular to another linear object or an axis of the active coordinate system.

The SmartSketch points are only located relative to geometry of objects that have been added to the **SmartSketch List.** You can add objects to this list automatically when the cursor pauses over the object for your configured **Dwell time**. Or, you can choose the option to manually add objects to the **Smartsketch List** (CTRL+D and select). The option to manually add objects to the list is very useful if your model is so dense that it is hard to locate the SmartSketch point on a specific object of interest.

If you click the middle mouse button when a relationship indicator is displayed, the relationship will be locked. You can then move your cursor to locate other SmartSketch points. A second click on the middle mouse button will unlock the relationship. This feature of locking and unlocking the relationship is also helpful in a dense model to isolate the points you want.

You can control the behavior of the precision point location using the Add to SmartSketch command located in the Tools menu and in the horizontal toolbar. The command displays the ribbon shown in figure 1.

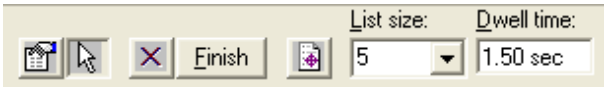


Figure 1: Add to SmartSketch List Ribbon

Options on the Add to SmartSketch List Ribbon:

The six options on the **Add to SmartSketch List** ribbon are described below:

1. **SmartSketch Options**—Displays the **SmartSketch Properties** dialog box, which enables you to set the SmartSketch points that you want the software to locate.

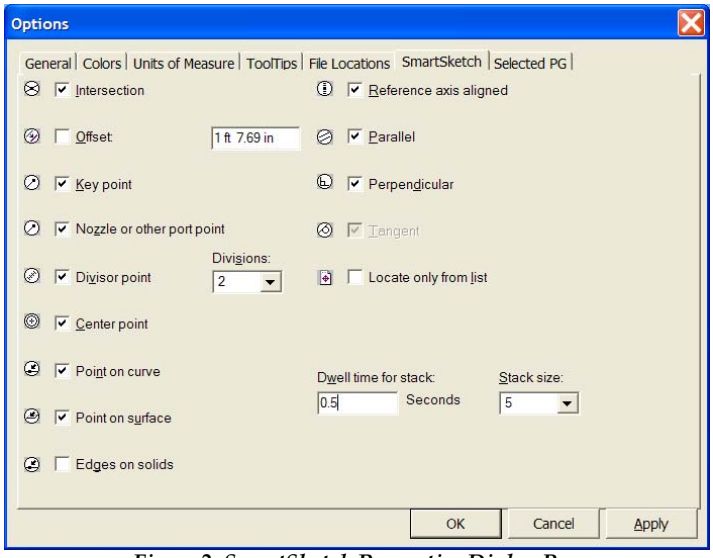


Figure 2: SmartSketch Properties Dialog Box

a. **Intersection**—Displays a relationship indicator ( ) when you move the cursor over points where two or more linear objects or parallel planes intersect.

b. **Offset**—Displays a relationship indicator ( ) when you move the cursor to a location that is the specified offset from an object or line you have added to the Smartsketch List. You need to enter an offset value in the **Offset** box (refer to Figure 3).

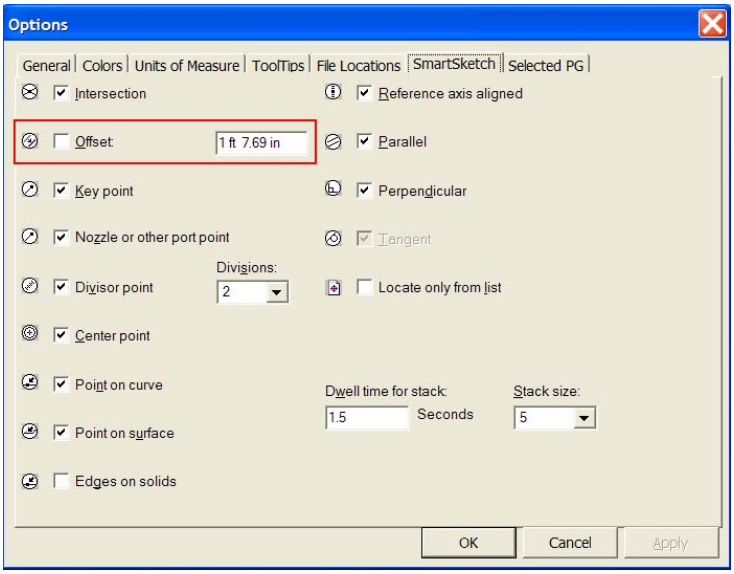


Figure 3: Offset Option

Tip:

- You need a projection plane to locate Offset points. You can set the projection plane using controls on the ribbon of the routing commands, Move command, and the 3D sketch component used in several application object modeling commands (for example, handrail and slab).

c. **Key point**—Displays a relationship indicator ( ) when you move the cursor over a point that the object has defined as important for design purposes. For example, a nozzle has a key point at the center of the port. A structure member has key points at the end of the member.

d. **Nozzle or other port point**— Displays a relationship indicator ( ) when you move the cursor over the graphics of a connection port such as a pipe nozzle. This keypoint is separated from the general geometric keypoints primarily so routing application users can elect to model without seeing the bounding box keypoints on the route geometry when they are not needed.

e. **Divisor point**—Displays a relationship indicator ( ) when you move the cursor to the points where a line is divided equally into the number of divisions that you set. A drop-down list enables you choose from 2 to 7 divisions for a line (refer to Figure 4).

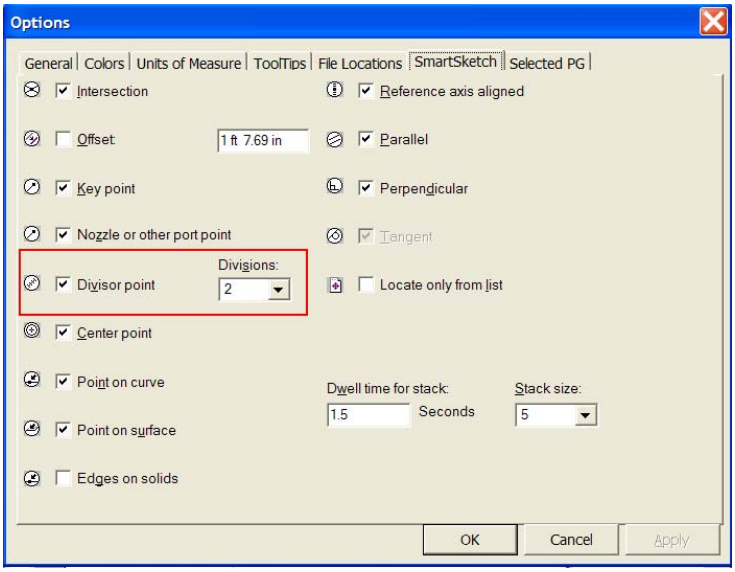


Figure 4: Divisor Point Option

f. **Center point**—Displays a relationship indicator ( ) when you move the cursor over the center of a circle or an arc.

g. **Point on curve**—Displays a relationship indicator ( ) when you move the cursor over any connection point on an object, such as a nozzle on a piece of equipment.

h. **Point on surface**—Displays a relationship indicator ( ) when you move

the cursor over a surface.

i. **Edges on solids**—Displays a relationship indicator () when you move the cursor over an edge on a solid object such as structure members, walls, and slabs.

j. **Reference axis aligned**—Displays a linear relationship indicator () when the line implied by the cursor point is aligned with the E-, N-, or EL-axis of the **Active Coordinate System.**

k. **Parallel**—Displays a linear relationship indicator () when the line implied by the cursor point is parallel to another linear object in the SmartSketch list. When this indicator appears, the software highlights the parallel objects. For example, if you add a section of a pipe that runs parallel to another pipe that has been added to the **SmartSketch List**, this indicator appears and both the objects are highlighted.

l. **Perpendicular**—Displays a linear relationship indicator () when the line implied by the cursor point is perpendicular to another linear object in the **SmartSketch list**. For instance, when you connect two pipes at a 90 degree angle, this indicator appears.

m. **Tangent**—Displays a relationship indicator () when you move the cursor such that the line implied by the cursor point is tangent to an object.

Tips:

- Most of the **Add to SmartSketch List** controls are also available on the **SmartSketch** tab of the **Options** dialog box.
- When a command is prompting you to locate a point, you can press F3 to toggle the relationship indicator for locating surfaces on and off. This makes locating key points much easier.

2.  **Select** - Selects objects to be added to the SmartSketch list.

3.  **Clears the SmartSketch List** - Removes all objects from the smartsketch locate list.

4.  **Locate on List Only** – When this toggle is on, objects must be added to the **SmartSketch List** manually. You can either use the Select smartstep (see item 2 above) or you can press Ctrl-D at any time a command is prompting for a point and pick. When the toggle is off, objects are added automatically when your cursor pauses over the object for the indicated **Dwell time**.

5. **Dwell time –** Specifies how long you must pause the cursor over an object before it is automatically added to SmartSketch list. A value around 0.5 seconds is usually best although.

6. **List size -** Specifies how many objects are added to the **SmartSketch List** before the software begins removing previously added objects. The value in this box must be an integer greater than 1. **List size** reflects is the same property as **Stack size** on the

SmartSketch Properties dialog box.

Using SmartSketch Points:

You can use SmartSketch points alone and in conjunction with PinPoint to define the precision points needed for your modeling. In this section you can practice several primary ways to use SmartSketch points in the context of routing a pipe. The same techniques can be used when entering points needed in the creation of other objects.

Define workspace as U02 and U03. We will route a pipe between the components near the two pumps to the nozzle at the top of the tower. Figure 5 shows the end result.

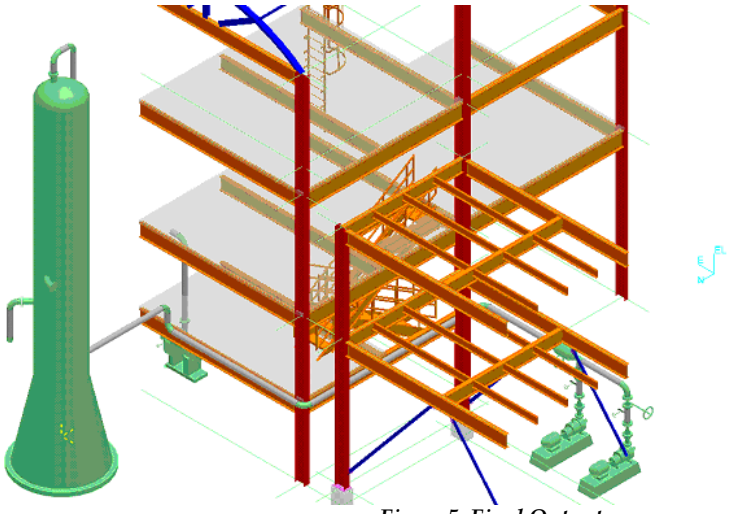


Figure 5: Final Output

1. Delete the pre-existing pipe before Clicking the **Route Pipe** command on the vertical toolbar.

Figure 6: Route Pipe Command on the Vertical Toolbar

2. Click **Tools > Add to SmartSketch List** command and select the **Locate From List Only** option on the **Add to SmartSketch List** ribbon.

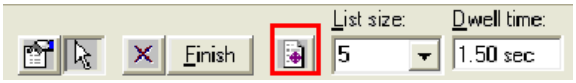


Figure 7:Locate From List Only option on the Add to SmartSketch List Ribbon

3. Locate the end feature of the component near the two pumps and click the **Finish** button on the **Add to SmartSketch List** ribbon.

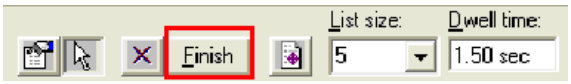


Figure 8: Finish Button on the Add to SmartSketch List Ribbon

This option is used in this example to make sure you can easily locate the SmartSketch points intended for the examples. In many design situations, you can easily work using the automatic addition of objects to the list.

4. Locate the piping component end feature keypoint.

Locating the keypoints on geometry is straightforward. You simply point, see the object highlight and the glyph, and then click.

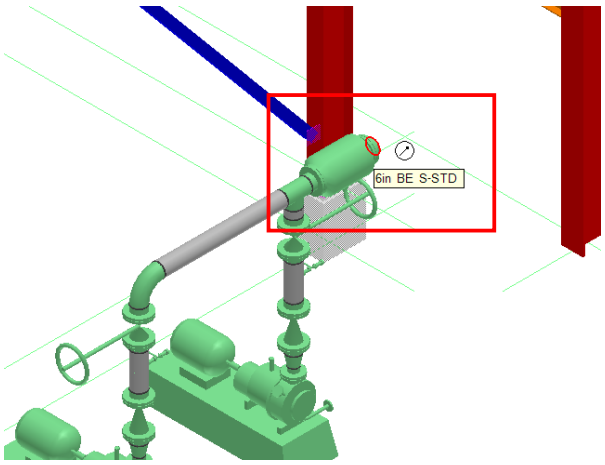


Figure 9: End Feature Keypoint

5. Set the route projection **Plane** to **E-W** as shown in the figure below. (the plane goes through the first route point).

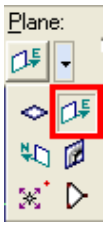


Figure 10: E-W Plane

The Route command automatically adds a fixed SmartSketch constraint to force the point to be on a line coming directly out of the component port (0 degree angle).

6. Click the **Add to SmartSketch List** command and add the surface of the column flange to the SmartSketch List and click the **Finish** button.

Tip:

- A quicker way to do this is to hold Ctrl+D down and select the surface.

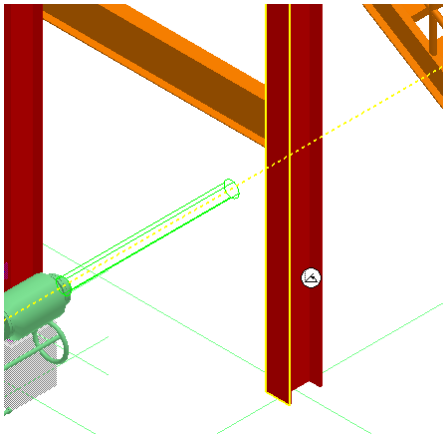


Figure 11:

7. On the **Route Pipe** ribbon set the **Offset** option to **Set Offset Reference** as shown in Figure 12.

Figure 12: Set Offset Reference Option

8. System displays the Set Offset Reference dialog box. The offset on the **Route Pipe** ribbon has additional offset calculation options. Select the option **Pipe Surface** on the **Measure from** section.

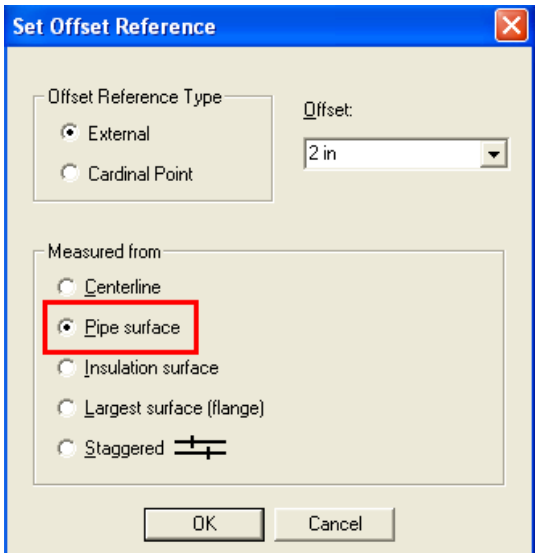


Figure 13: Set Offset Reference Dialog Box

The point you enter for the pipe is the center point for the pipe. The smartsketch offset distance is automatically calculated for you such that the surface of the pipe will be the offset distance from the line or plane you reference in the following step.

9. Locate the Offset from plane. In this situation, you have a projection plane and two relationships locked; the angle out of component and the offset from the column.

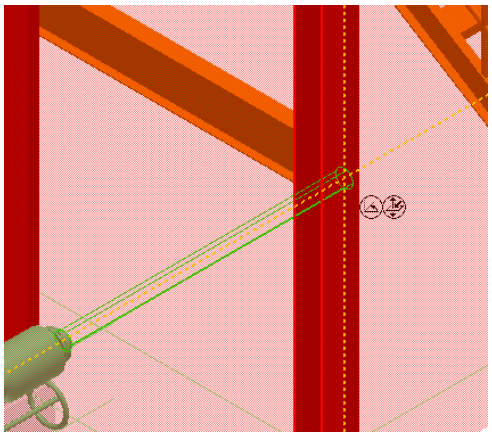


Figure 14: A Projection Plane and two Relationships Locked

Tip:

- The offset is often easier to locate a view perpendicular to the projection plane.

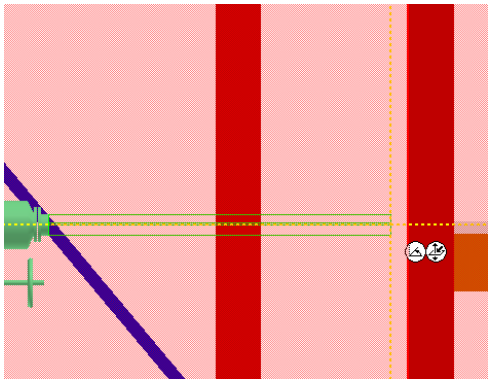


Figure 15: Perpendicular View

10. Set the route projection **Plane** to **N-S** as shown in the figure below.

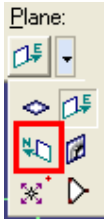


Figure 16: N-S Plane

11. Position the cursor so that you see the Align with North axis glyph and then middle mouse click to lock to the North axis direction (through the last point you gave). The pipe will now dynamically extend only along the north-south axis. The cursor point is projected to axis.

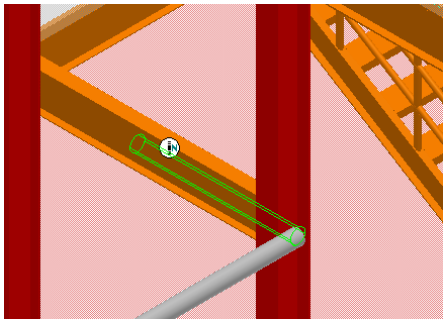


Figure 17:

12. Use the **Add to SmartSketch List** command or Ctrl+D to add the edge of the column near the tower to the SmartSketch list.

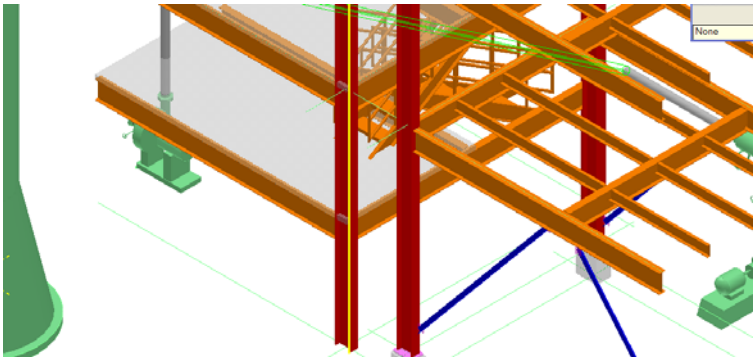


Figure 18: Edge of the Column Near the Tower Added to the SmartSketch List

13. Locate the offset from plane as done in step 7.

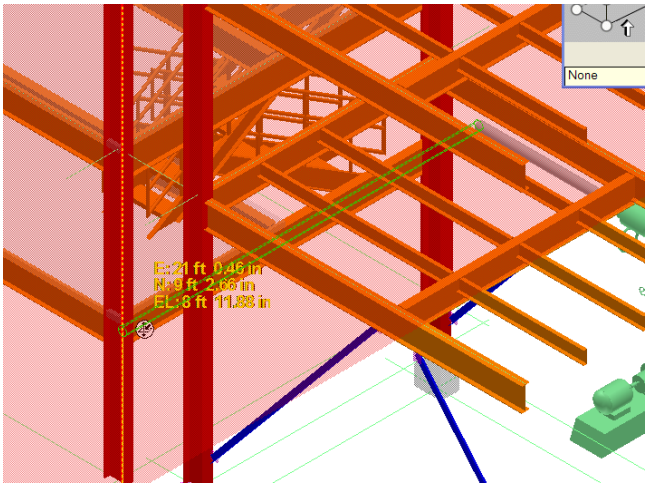


Figure 19:

14. Add the top nozzle on the vertical tank to the Smartsketch list and set the route projection plane to **No Projection Plane** option and lock SmartSketch to the E-W axis with middle mouse click.

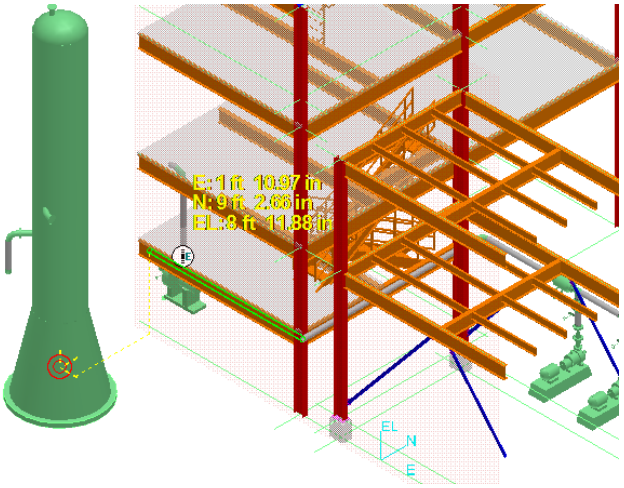


Figure 20: Locked SmartSketch to the E-W Axis With Middle Mouse Click

15. Position the cursor over the nozzle and click. This projects the point of the nozzle to the locked axis to establish the turn point.

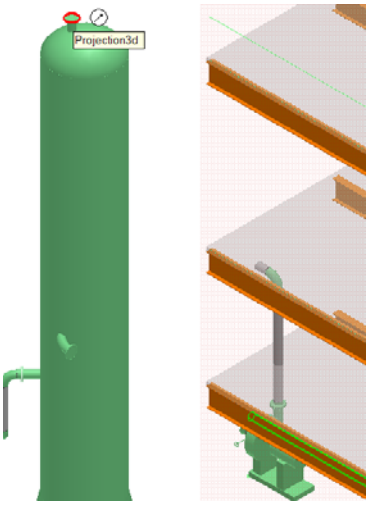


Figure 21:Selected Nozzle

16. Add the tank cylinder to the SmartSketch list.

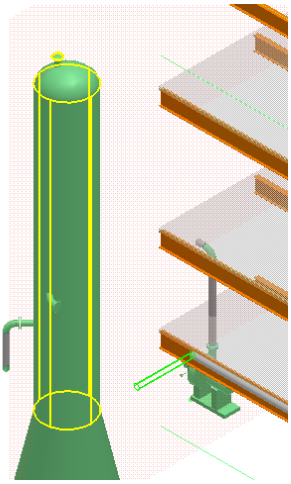


Figure 22:Selected Tank Cylinder

17. Set the route projection **Plane** to **E-W** and then lock smartsketch to be aligned with vertical axis.

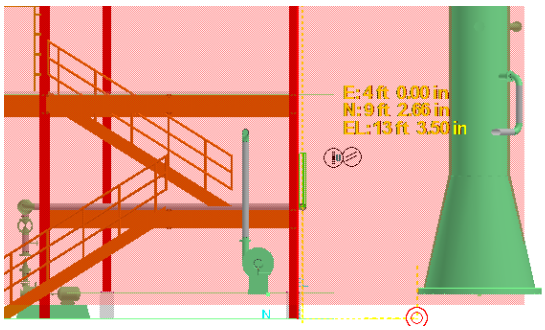


Figure 23: E-W Plane

18. Locate the bottom of the vertical vessel cylinder and click to place pipe.

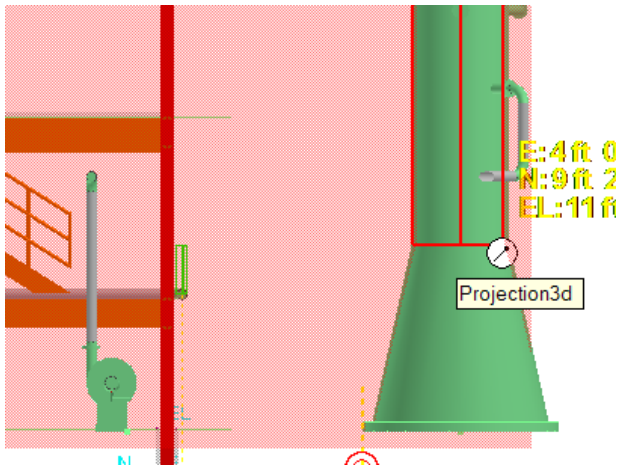


Figure 24:Selected Bottom of the Vertical Vessel Cylinder

19. Lock to N-S axis (route command also has a mode where it automatically locks to 90 degrees) and locate the offset smartsketch point. The offset is computed from the line created by the intersection of the projection plane with the cylinder.

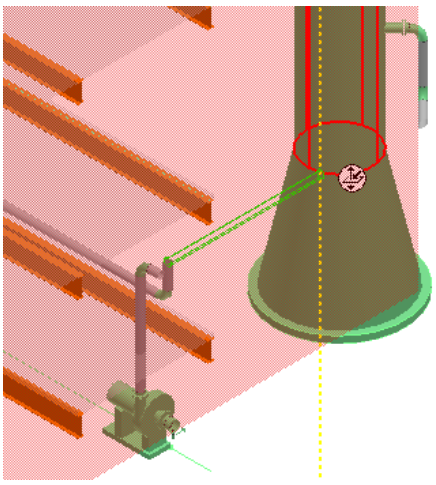
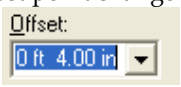


Figure 25: N-S axis

20. After locating the SmartSketch offset point change the offset from 0 ft 2.00 in to 0 ft 4.00in



21. Activate the PinPoint tool and set the target to the top nozzle of the vertical tank. Key-in **F8** and then key-in **1.5 ft**. This locks the next point you enter to be 1.5 ft above the nozzle.

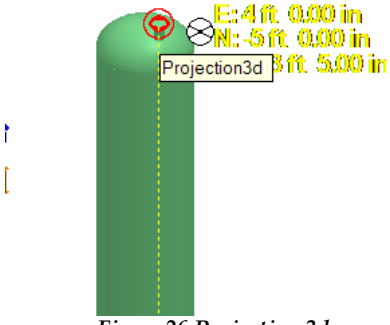


Figure 26:Projection 3d

22. Position the cursor so it finds the vertical axis location and click on the location. The projection plane does not have to be locked, but can be left locked.

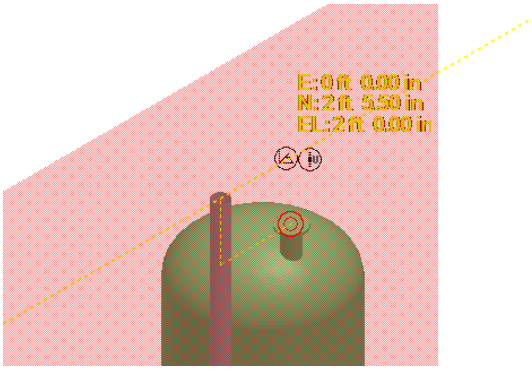


Figure 27: Vertical Axis Location

23. Lock to E-W axis and position cursor over the nozzle to locate the last turn point.

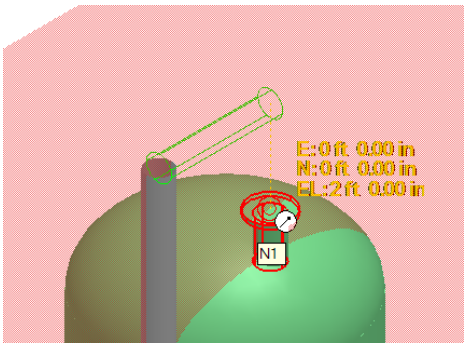


Figure 28: Last Turn Point

24. Finally, position cursor over the nozzle and click to connect to the nozzle.

Figure 30:Connected Nozzle

For more information related to Add to SmartSketch List ribbon, refer to the *Using the Tools Menu: An Overview* topic of the user guide *CommonUsersGuide.pdf*.

Quiz:

4. When does the software display **SmartSketch** points?
5. How do you select which **SmartSketch** point locate options are active?
6. When should you use the **Locate on List Only** option?

7. Where is this option located?
8. How do you lock a SmartSketch relationship before you enter the point?

Session 10: Assigning Objects to WBS Items

Objective:

By the end of this topic, you will be able to:

- Assign objects to a Work Breakdown Structure (WBS) item.
- Name some uses for the Work Breakdown Structure.

Prerequisite Sessions:

- SP3D Overview
- Creating a Session File
- Defining a Workspace
- Manipulating Views
- Selecting Objects in a Model
- Applying Surface Style Rules

Overview:

The **Work Breakdown Structure** (WBS) is basically a means to define additional groupings of design objects for whatever work purposes your company might need. It is well suited for dividing up the design for contracting or fabrication purposes.

The terminology for the first level of grouping object, the **Project** (and **As-Built**), was chosen to help Owner Operators manage the project work on existing facilities. EPCs can use the Project level grouping for major divisions of responsibility. An object can only belong to one project.

You can nest additional types of groupings called WBS Items under the project. An object must be claimed to the same project as the WBS Item to be assigned to the WBS Item. An object can belong to more than one WBS Item depending on the rules discussed below. Your company can customize the type/purpose of WBS Item by editing the Catalog. The default types delivered with the product are suitable for contracting activities in the different disciplines. The piping application uses the WBS Item groupings to manage the assignment of pipes to specific isometric drawings.

You can assign objects to WBS projects and WBS Items after they have been created. You can also have the objects assigned automatically as you create them. The currently active Project or WBS Item is shown in the drop-down list in the upper left corner of the task window next to the **Permission Group** box. If a project is identified, all new objects are assigned to that active project when they are created. If a WBS Item is identified, objects are first assigned to the parent project of the WBS Item and are then assigned to the WBS Item.

When you edit an object, it retains its current project and/or WBS Item relationship. You change the project parent of an object by selecting the object and running the **Claim** command. You can change the WBS Item relationship with the **Assign to WBS** command.

Steps for Manually Assigning Objects to a WBS Item:

Assign all the structural objects of Unit **U02** to a WBS Item, **Contract2**, of your active project.

1. Define your workspace to show Unit **U02**.
2. Click the **Tools > Select by Filter**… command. This command provides options to select the objects by using filters.

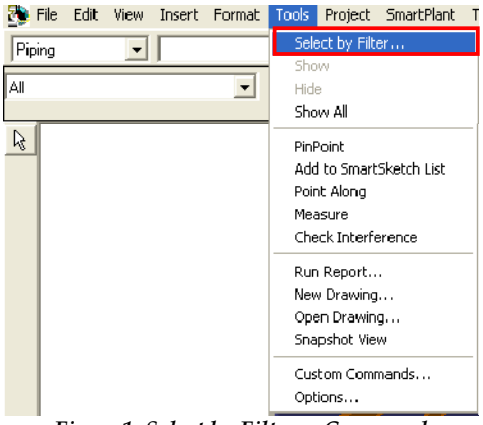


Figure 1: Select by Filter... Command

3. Under **Catalog filters** in the **Select Filter** dialog box, expand **Default Filters >SP3D Object Filters > Object Types** and then, select the **Structure** filter.

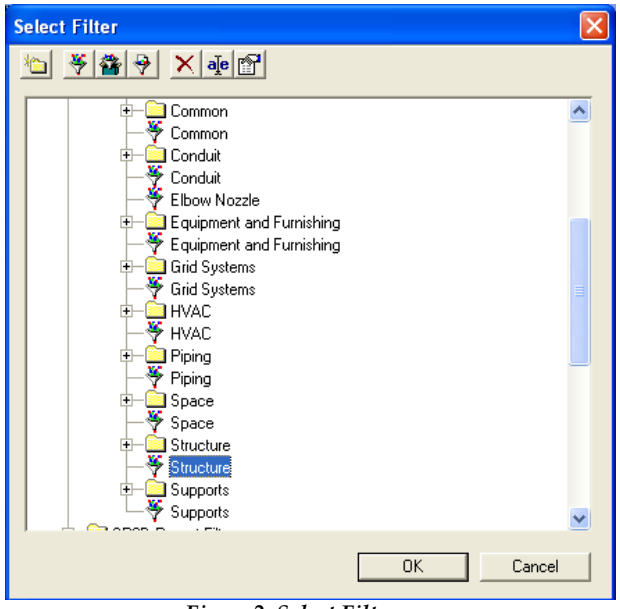


Figure 2: Select Filter

The system will highlight all structure objects in Unit **U02** of your workspace. (Refer to Figure 3.)

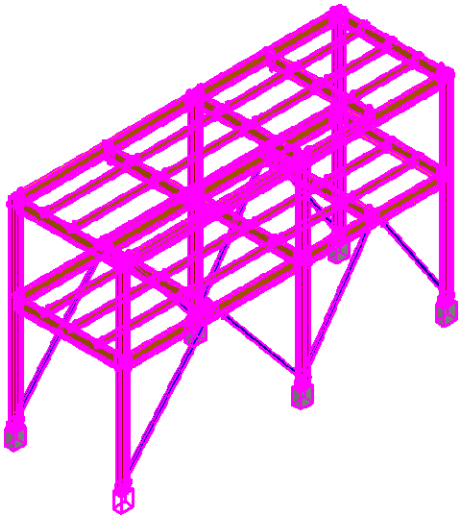


Figure 3: Structural Objects Selected Using The Structure Filter

4. From the active **WBS** drop-down list, select the **More…** option to select the active project in WBS.

Figure 4: WBS Drop-Down List: More… Option

5. In the **Select Active Project** dialog box, select **Project1**.

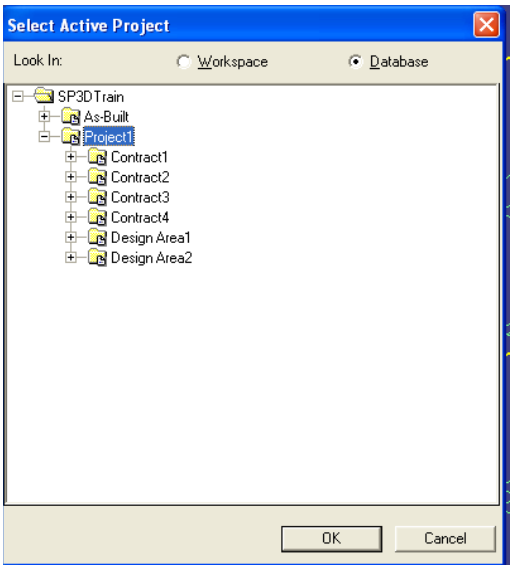


Figure 5: Select Active Project dialog box

Hint:

- If **Project1** is not available in the **Workspace** list, select the **Database** option at the top of the form. The Database option will display WBS Items that are found in the database and not just the items retrieved to the workspace based on the current Define Workspace filter.

6. Then click the **Project** menu and select the **Claim** command to associate all the selected objects with **Project1**.

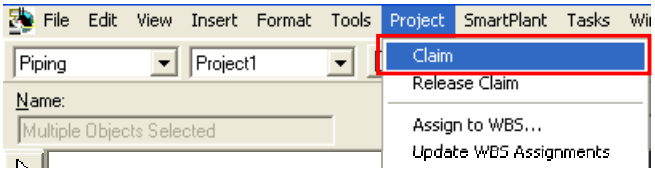


Figure 6: Project Menu: Claim Command

Notes:

- The **Claim** command creates a relationship between the selected objects and the active project. This relationship is communicated to the SmartPlant integrated environment if you are working in the Project/As-Built mode in SmartPlant Enterprise.
- An object can be related to only a single project. If you attempt to claim an object that belongs to another project, the claim will fail. You must select the **Release Claim** command and then claim the object to a different project. SmartPlant Enterprise requires this two-step change process.
- An object must be claimed to the project of the selected WBS Item before selecting the **Assign to WBS** command.
- You must have write access to the Project to claim an object to a project. This prevents people from creating objects within groupings they do not control. A person on a workshare site can claim objects to project even if the host owns the permission group the project belongs to. The person doing the claim at the remote location must be assigned "write" access to the project's permission group on the host. The same rules and behavior apply to assignment of objects to WBS Items.

7. Create a new WBS Item in Project1 by right-clicking Project1 and selecting the **Create WBS Item** command. Select the property options shown in Figure 7.

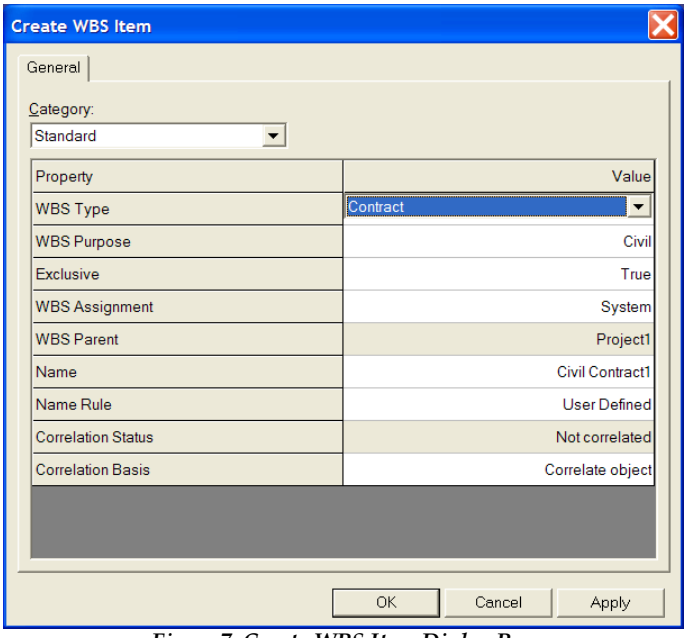


Figure 7: Create WBS Item Dialog Box

Notes:

- The available **Type** and **Purpose** property values are defined in the Catalog that the model is using.
- If the **Exclusive** property is set to true, it will prevent a given object from being assigned to another WBS Item of the same **Type** and **Purpose**. If the property is set to false, a given object can belong to multiple WBS Items of the same Type and Purpose.
- The **WBS Assignment** property controls the behavior of the **Assign to WBS** command. If you set this property as System and then assign a system to a WBS Item, the **Assign to WBS** command automatically assigns all objects nested under the system to the WBS Item as well. If the property is set to System, then you can ensure that ALL children of a given system will be assigned to the WBS Item without you having to do it manually. If you change this property to Assembly, the System children will not be automatically assigned to the WBS Item at the time of its creation. It will assign the children of assemblies automatically instead.
- The system assignment overrides any individual assignment that you may have previously made of objects to exclusive WBS Items.

8. Now click the **Project** menu and select the **Assign to WBS…** command, with all the structural objects highlighted. The **Assign to WBS…** command creates a relationship between the selected objects and a selected WBS Item.

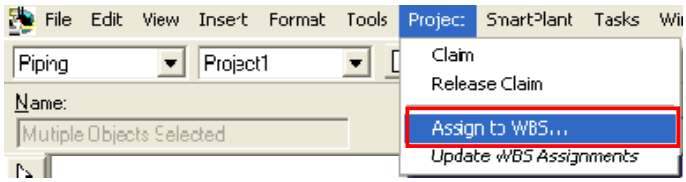


Figure 8: Project>Assign To WBS... Command

9. In the **Assign to WBS** dialog box, expand **Project 1** and select **Civil Contract1,** a WBS Item to which the structural objects need to be assigned. Then, click **OK** to create the relationships between the objects and the selected WBS Item.

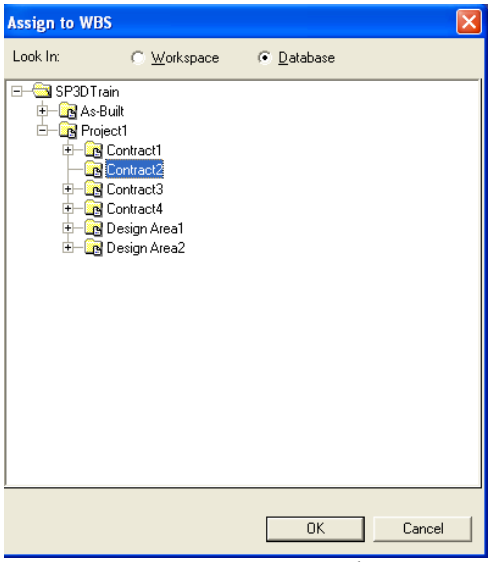


Figure 9: Assign To WBS Dialog Box

Tips:

- Later, if you want to verify whether the structural objects have been assigned to the WBS Item, **Civil Contract1**, you can review the properties of the objects in the **Relationship** tab of the **Member System Prismatic Properties** dialog box. (See the highlighted properties in Figure 10.)
- You can also select the WBS Item and click the **Select Nested** command on the shortcut menu. This will select all objects nested under the WBS Item.
- The objects related to WBS Items are not shown in the **WBS** tab of the **Workspace Explorer**.

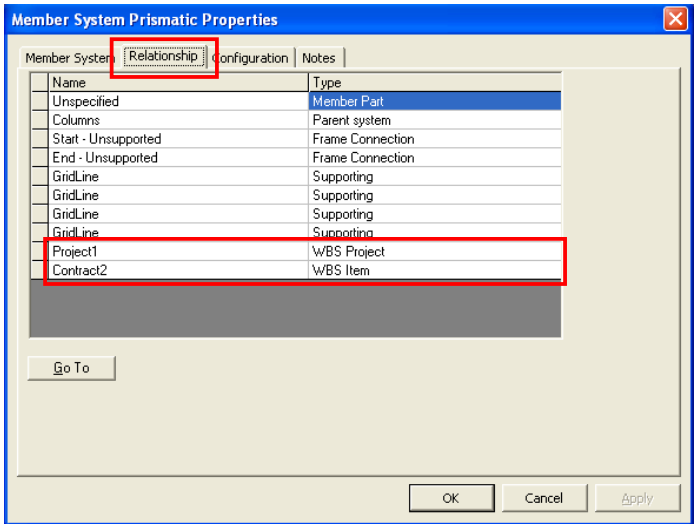


Figure 10: Member System Prismatic Properties Dialog Box

Steps for Automatically Assigning Objects to a WBS Item:

As mentioned in the Overview, you can also assign objects to a Project and WBS Item automatically as the objects are created.

1. From the active **WBS** drop-down list, select the **More…** option to select the active WBS Item.

2. Create objects.

As objects are created, they are automatically assigned to the active WBS project and/or WBS Item according to the following specific rules:

If a WBS project is explicitly chosen as the Active WBS, then all objects you create are automatically assigned to that project. The access control rules that control manual assignment apply for the automatic assignment.

If a WBS Item is explicitly chosen as the Active WBS, then only specific object types will be automatically claimed to the parent project of the WBS Item and then assigned to the WBS Item. Only lowest-level objects that are included in deliverables and have identity are automatically assigned (sometimes termed "parts"). In other words, design objects like piping features or the grouping objects like Pipelines are not automatically assigned, but piping parts are assigned. You must manually Claim any System object to a Project and manually assign it to a WBS Item.

For more information related to WBS projects and items, refer to the topics *Using the Project Menu: An Overview* and *Managing WBS Items and Projects: An Overview* in the user guide *CommonUsersGuide.pdf*.

Session 11: To Do List

Objective:

By the end of this session, you will be able to:

- Use the To Do List to view and correct model data inconsistencies.

Prerequisite Sessions:

- SP3D Overview
- Creating a Session File
- Defining a Workspace
- Applying Surface Style Rules

Overview:

The To Do List shows you issues or inconsistencies that need to be resolved in your model. It is critical to review the To Do List often and eliminate the problems. There are three basic types of issues, **Out of Date, In Error**, and **Warning.**

- **Out of Date**: On object is Out of Date when it has not yet recalculated after one of its related input objects has been modified or deleted. Figure 1 shows the **LadderA1-1-0101** is **Out of Date.**

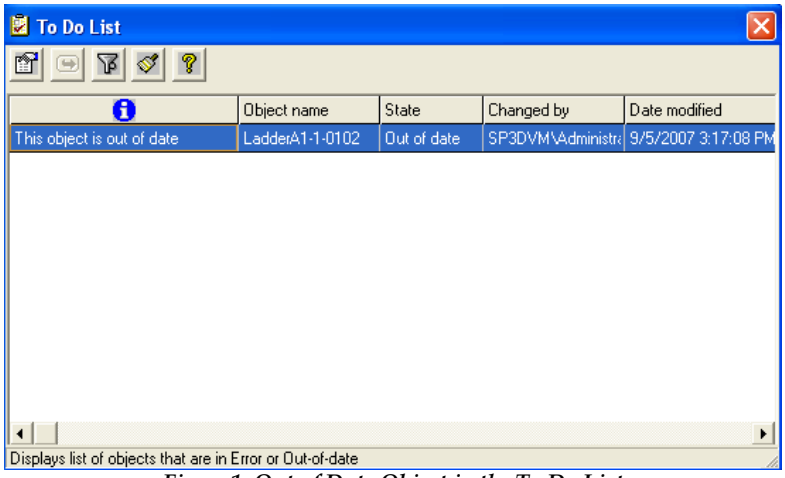


Figure 1: Out of Date Object in the To Do List

An object will not be recalculated if the user who changed the input object does not have write access to the dependant object, the object is not in working status, or if you have elected to delay the recalculation of all dependent objects in other permission groups (see Tools>Options > Selected PG tab). The **Synchronize Model with Catalog** command can add objects to the To Do List as Out of Date if reference data needed by the object is changed.

- **In Error:** There are many reasons why an object may be in an error state. The To Do List will describe the reason. These reasons fall into three classes:

  1) Changes in catalog data that a model object is dependant on (See Synchronize Model with Catalog) results in a calculation error.
  2) Loss of a required input object. When an object with the information on which your design object is dependent is deleted, your object will go on the To Do List. Changes between objects that have relationships (design integrity - although notification is dependent on specific relationship types - i.e., not all objects with relationship provide change notification within the To Do List). For example, if the bottom surface reference (a slab) of a **LadderA1-1-0102** is deleted, then the ladder would be in **In Error** state as shown in Figure 2

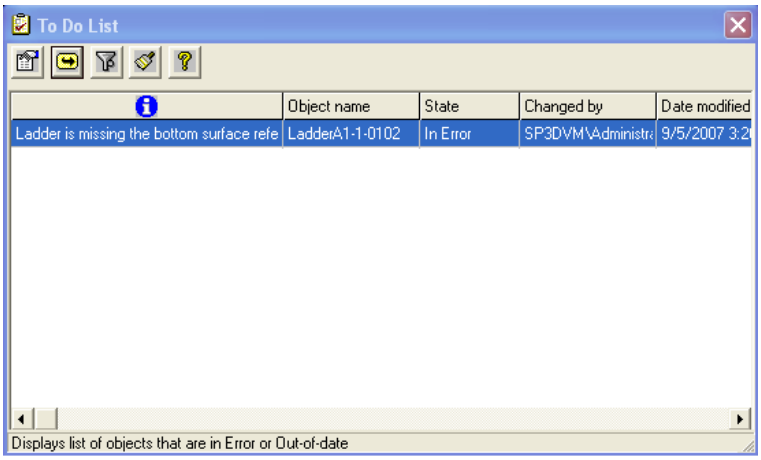


Figure 2: Object in In Error State in the To Do List

  3) System-required data integrity rules are violated when the object is recalculated due to a change to its related input objects.

- **Warning:** A few data integrity rules that may be broken by an object are not serious enough to require repair before generating design deliverables. These are marked as warnings.

  Tip:

  - The **Help** button on the **To Do List** dialog box provides more information about a specific problem and recommends action that you can take to repair your design.

To Do List:

To open the **To Do List**, click the **View > To Do List** command.

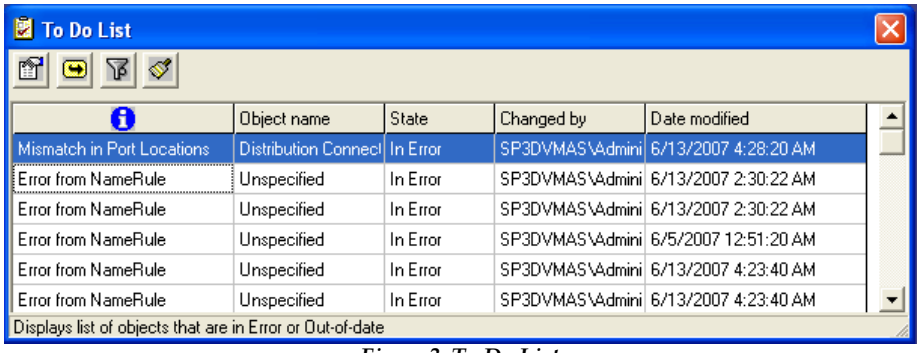


Figure 3: To Do List

Options in the To Do List Dialog Box

- **Properties –** Defines the type of To Do List items to be displayed and the information that should appear in the **To Do List**
- **Update –** Causes an object to recalculate. You use this option to get your **Out-of-date** objects to recalculate based on their updated inputs. This works for all objects regardless of what task environment you are currently in.

  Tips:

  - When you update an object in the **Out of date** state, the object is recalculated using its current input. In addition, all objects that depend on your Out-of-date object (and for which you have write access) are updated.
  - You can select more than one **Out of date** object at a time by using the CTRL or SHIFT key while selecting the objects.

- **Filter To Do List by Select Set –** This option lets you to list only the To Do List items related to the currently selected objects in the workspace. The To Do List items will continue to be restricted after you select this option even if the select set is changed.
- **Clear Filter –** This option removes the filter applied using the **Filter To Do List by Select Set** command. All To Do List items that match the currently defined criteria of the To Do List properties dialog box are displayed.

Fixing Objects In Error:

You can modify an object on the To Do List by selecting the object in the list and then editing the object using the Edit ribbon of the corresponding task. To see the Edit ribbon of an object, you must be in the task environment where that object was created. After you resolve the inconsistency in an object, the object is automatically removed from the To Do List.

Hint:

- **In Error** objects are usually edited one at a time. However, you can also select more than one object if an **Edit** option supports multiple objects.

Managing the To Do List

It is critical to review the To Do List often and eliminate the problems. Ideally, you should have no To Do List records related to objects that you document on design deliverables. You should generally follow process where you do not approve an object if it has a To Do List record.

One way to help you notice To Do List issues while you design is to define a surface style rule that sets the color of items with To Do List records to a special color for easy identification. Your administrator can define this style rule tailored for your company for you to use. Select and add this style rule to your session.

When your catalog administrator has run **Synchronize Model with Catalog**, such that the actual update of the model is delayed, then many of your design objects may be marked as out of date with the current catalog data. You should update and review the impact of these changes before attempting to model additional objects relative to the Out of Date objects.

Changes to piping specifications are currently the most difficult catalog changes to deal with. When a piping specification is changed in the catalog, all piping runs that use that specification and hence all features of these runs must update, and so go on the To do List. This happens even if the change to the piping specification will not cause an actual change to the resulting parts and if the piping features/parts are approved. You have no choice but to update the features and review the impact. This is an important area that will be improved in future releases.

For more information related to the To Do List, refer to the topic *Correcting Errors: An Overview* in the user guide *CommonUsersGuide.pdf*.

Quiz:

1. Give examples of reasons why an inconsistency arises in the objects within the SP3D database.
2. What are the different types of inconsistencies that can occur in the database?
3. Which option will you use to assign the object content in the To Do List?
4. Which menu will you use to open the To Do List?
5. Which option updates Out-of-Date objects?

Session 12: Interference Checking

Objective:

By the end of this session, you will be able to:

- Describe the interference checking features of SP3D.

Prerequisite Sessions:

- SP3D Overview
- Creating a Session File
- Defining a Workspace
- Manipulating Views

Overview:

SP3D interference detection runs concurrently in background, like Microsoft Word's "Spelling Checker" feature. Thus, interferences are identified and can be resolved quickly as they are created. In contrast, current design tools run the interference check after design is complete and this involves the laborious and iterative approach of 1) design work, 2) managing multiple designers to a common design status, 2) executing a check, 3) evaluating the results, 4) assigning problems to designers to be fixed, 5) correcting the problems, 6) iterating to ensure all problems have been resolved.

Interference between two objects is detected when their geometry intersects. Design objects can have multiple geometries, called aspects, which represent different geometric characteristics of the objects, such as the physical shape, insulation, maintenance, and operational spaces. You can configure the **Interference Checking** process to detect interferences between the different types of geometry. You can also configure the amount of clearance required between the physical aspects of the different object types. The interference checking process can optionally check for interference with the geometry from a referenced PDS project and referenced MicroStation files.

When interference is detected, an interference record is created. The **Type** property of the record indicates the severity of the interference as defined by your administrator for the geometric aspects that interfere. While several geometric aspects of two objects may intersect, only a single interference record indicating the highest severity level between two objects is created. The interference record is automatically removed when you modify the geometry so that there is no interference. You can approve interferences that you do not want to eliminate.

Two types of interference checking processes help you integrate interference detection into your daily design process:

- Server-based interference checking – also known as Database Detect
- Interactive interference checking – also known as Local Detect

Server-Based Interference Checking (Database Detect):

The server-based interference checking (Database Detect) process runs directly on the model database. This process can be run on any computer on which the Database Interference Detection service is installed. After you start the service, SP3D automatically checks all existing objects that have not been checked and all new or modified objects for interference.
You never have to be concerned about when to run Interference Check. It is running all the time and the software tracks the time when individual objects were checked for interference. The process can be stopped and restarted later. The system knows the objects that need to be checked.

Note:

- When an interference record is created, a relationship is established to the graphic objects involved in the interference. Currently, the design objects are edited from a database perspective by this operation. When an object is so edited, designers can no longer undo any edit they have previously made to these same design objects. To mitigate this problem, your administrator can set a delay time between when an object is created or edited and when the object is checked for interference. This delay should be longer than the time it typically takes an edited object to be bumped from the Undo buffer (you only save Undo for *n* operations). This time delay on the server checking will also explain why you should not expect an interference you have just recently created (as seen by the local detection) to show up immediately as a Database Detected interference.

While your site administrator manages the **Database Interference Detection** process, you can review the configuration and status of the detection by using the **Interference Detection Properties** dialog box.

In a workshare configuration, the Server-based interference checking is run at the Host location. The interferences found by the process are assigned to permission groups owned by the host and then replicated to the workshare sites and can be reviewed just like any other data added to the model. Unlike other data, however, Designers at the remote locations can edit the status of Interference objects if they have write permission to the Interference's permission group on the host.

Steps for Reviewing Database Interference Detection Settings:

1. Click the **Tools > Check Interference** command**.**

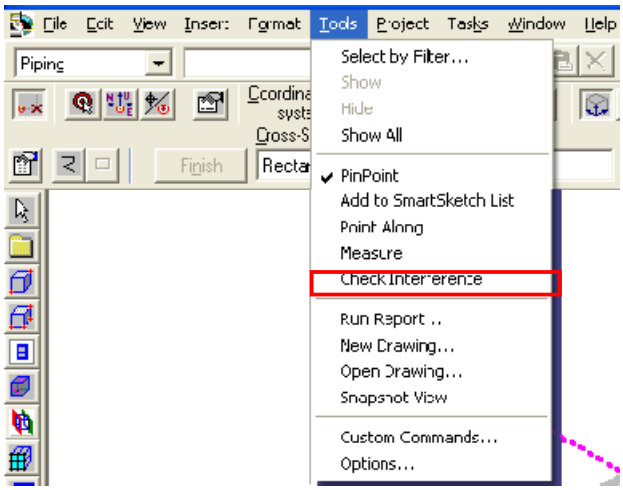


Figure 1: Tools > Check Interference Command

2. This will display the **Interference Checking** ribbon. Click the **Settings** option on the **Interference Checking** ribbon (Refer to Figure 2).

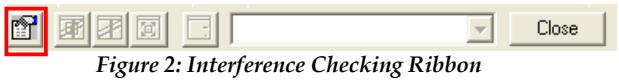


Figure 2: Interference Checking Ribbon

Note:

- The **Properties** option on **Interference Checking** ribbon will open the **Interference Checking Settings** dialog box. The Database Detect tab shows the status of the Server Interference Detection (Refer to Figure 3).

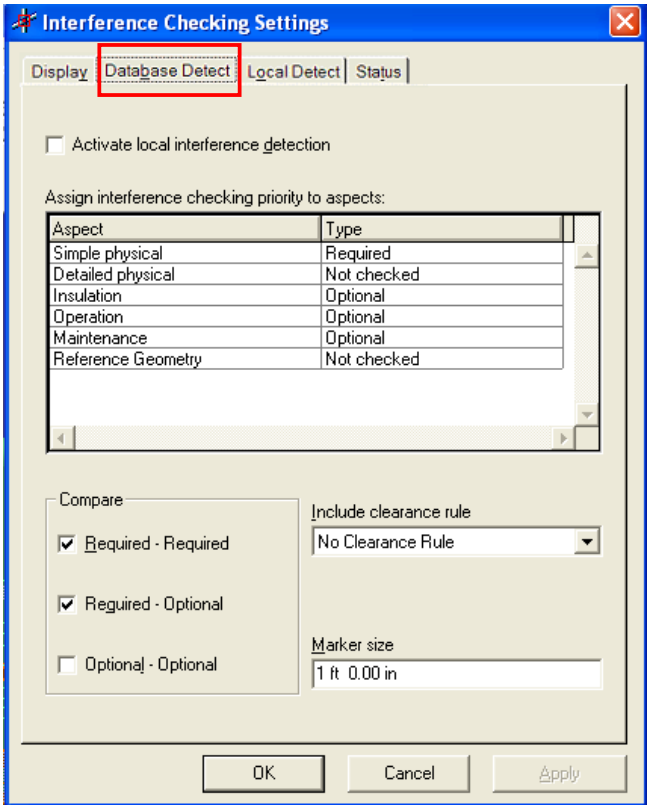


Figure 3: Interference Checking Settings Dialog Box

- The **Status** tab on the **Interference Checking Settings** dialog box will show you the status of database interference and the time when the interference started.

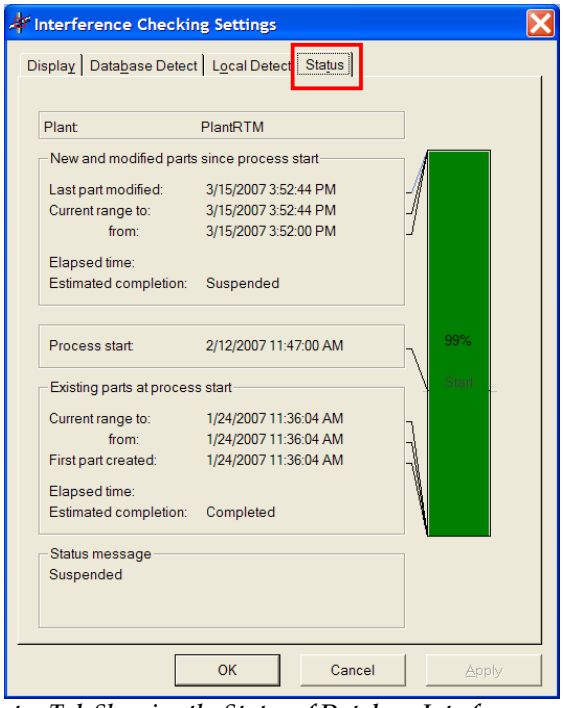


Figure 4: Status Tab Showing the Status of Database Interference

Interference records relating to any object that you elect to bring into your workspace by using the **Define Workspace** command are also automatically loaded and available for you to review in your design session. You can view any interference detected on the server after you have run the **Define Workspace** command by refreshing the workspace.

Database Detect marks the interference in the model by placing spherical interference markers at the location of the interference. For example, in Figure 5, the service has placed a spherical interference marker at the interference between a floor slab and a pipe.

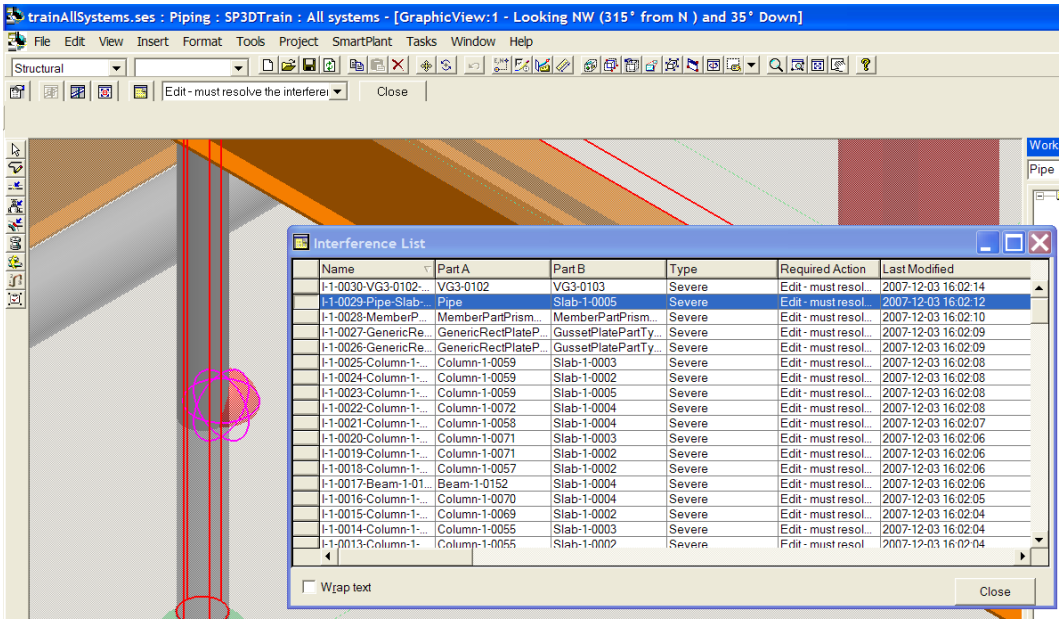


Figure 5: Spherical Interference Marker at the Interference Between a Floor Slab and a Pipe

Note:

- You can graphically select the interference marker without having the Interference List view displayed and review the properties of the interference record by using the **Edit > Properties** command.

Interactive Interference Checking (Local Detect):

Interactive interference checking (Local Detect) is a process that runs only in the current session on your SP3D client computer and detects interferences only when you create and edit objects. It does not check the existing objects. Local detect provides real-time help in the form of graphical feedback about interferences. You can see these interferences immediately after placing or editing the objects responsible for the interferences in a model. This process checks for interferences between the objects displayed in the active workspace. So, hidden objects or objects that are not retrieved to the workspace from the server are not checked.

The **Local Detect** process records any interference that it finds with a cube interference marker. For example, in Figure 6, the **Local Detect** process has placed a cube interference marker at the intersection of a pipe and the floor as it found an interference at that location.

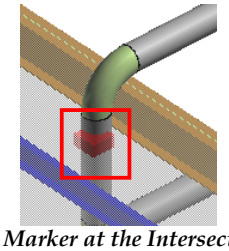


Figure 6: Cube Interference Marker at the Intersection of a Pipe and the Floor

Local Detect interference is not persistent. Therefore, if you close and open or refresh your session file, all the local interferences are removed from memory. However, if Database Detect is running on the model, any local interference in your model will be found by the database detection process and be seen at some future time when you refresh your workspace.

Steps for Starting and Stopping the Local Detect Process:

1. Click the **Tools > Check Interference** command**.**

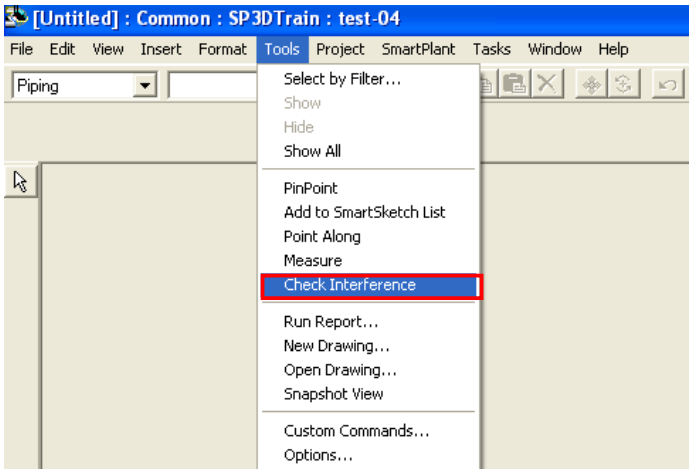


Figure 7: Tools > Check Interference Command

2. The **Interference checking** ribbon will appear. Click the **Settings** option on the **Interference Checking** ribbon. This will open the **Interference Checking Settings** dialog box. Click the **Local Detect** tab on this dialog box (Refer to Figure 8).

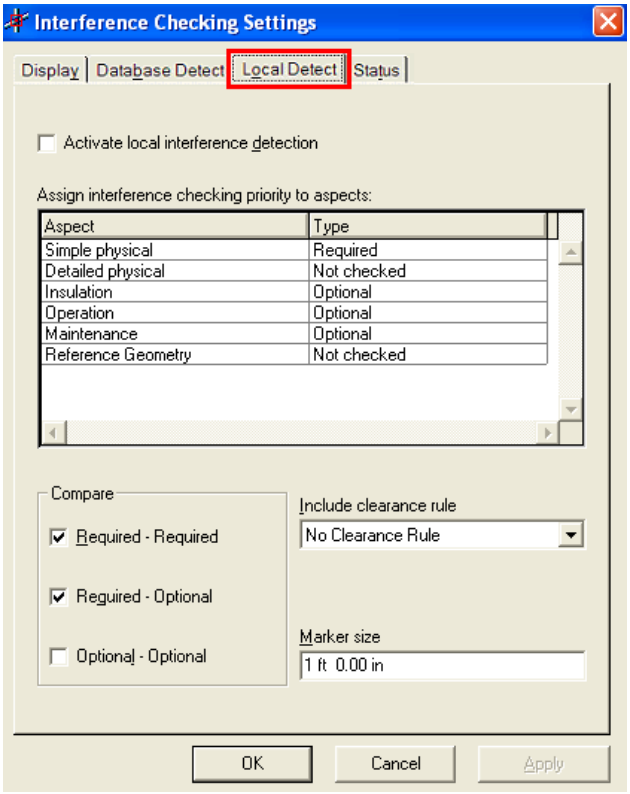


Figure 8: Interference Checking Settings Dialog Box

3. Select the **Activate local interference detection** check box and click **OK** in the **Interference Checking Settings** dialog box to start the **Local Detect** process in your session (Refer to Figure 9).

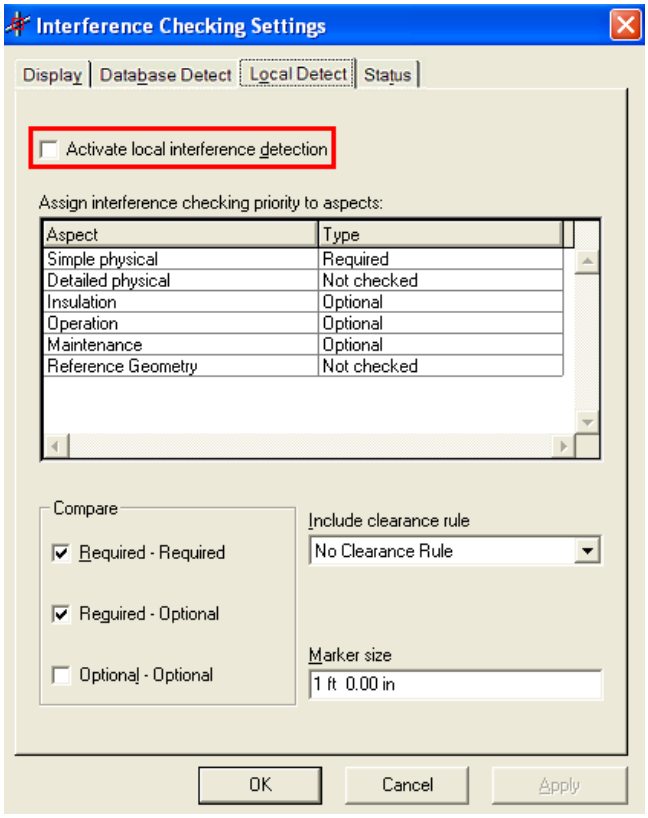


Figure 9: Interference Checking Settings Dialog Box

Tip:

- It generally best to use the same settings as your administrator for the database detection. The settings shown in figure 9 are typical.

4. To stop the Local Detect process, clear the **Activate local interference detection** check box in the **Interference Checking Settings** dialog box and then, click **OK**.

Reviewing Interferences:

When you click the **Tools > Checking Interference** command, the **Interference Checking** ribbon appears (Refer to Figure 10).

Figure 10: Interference Checking Ribbon

The various options available on this ribbon allow you to review and update the interferences if you have the right permission to the permission group the interference is assigned to. For example, if you are located at a satellite location in a workshare configuration, you can edit the Interference status for Interferences if you have been assigned write access to the appropriate permission groups on the Host. Your administrator will define rules for which permission group an Interference is automatically assigned to when it is detected. You can also change

the permission group of an Interference just as you can any other object in the system.

- **Settings:** The **Settings** option on the ribbon opens the **Interference Checking Settings** dialog box. This dialog box allows you to review the **Database Detect Interference** checking properties as well as the **Local Detect Interference** checking properties. It also allows you to start and stop the Local Detect process.

- **Hide Interferences:** This allows you to hide the interference markers. If no interferences are selected, then all interferences will be hidden. If one or more interferences are selected, then only the selected interferences will be hidden. When interference is hidden, it is also not shown on the List View.

- **Show Interferences:** This allows you to show (display) all the interferences related to design objects in your workspace. This will not show other objects you have hidden with the Tools>Hide command.

- **Fit Interference:** Fits the active view to both objects involved in the selected interference. If you have more than one interference selected, this fits to all objects related to the selected interferences. You can fit to a single object involved in the interference by selecting the objects in the Interference list view (picking on the field showing the name of the object) and then selecting the **Fit** command on the common horizontal toolbar.

- **List View:** This lists all server interferences related to objects that are in the current workspace and all current locally detected interferences. You can sort interferences by clicking on the columns. You can edit the displayed Required Action and Notes for Database Detect interferences on this form. All other fields are read-only. See Figure 5.

- The **Required Action:** When **Database Detect** interference is selected, it shows the action currently defined for the interference. You can select a different action from the drop-down list. The field is disabled when **Local Detect** interference is selected.

You can graphically select an interference you see and then edit the **Required Action** on the **Interference Checking** ribbon. Or, if the **Interference** ribbon is not displayed, edit the interference using the **Properties** dialog box.

You can select **Interference** on the **Interference List View**; pick the **Fit Interference** command to view the interference graphically. You can then either edit the **Required Action** on the **Interference Checking** ribbon or directly on the **Interference List View**.

You can add notes about the interference on the **Interference List View** or on the **Properties** dialog box for the interference.

Recommended Workflow for Managing Interferences:

The continuous interference detection processes allow you to integrate the information about interferences into your daily design activities rather than waiting for batch checking.

Enable local detect when you design. This will help you detect and solve problems immediately.

Observe the graphic interference markers that appear on your design objects and regularly activate the **Interference List View** and look at the **Database Detected** interferences related to your area of responsibility. There are several ways interferences can be assigned to you by the server process. For example, the name of the interference can indicate responsibility when it is calculated by a custom name rule. Optionally, the interferences can be assigned to different permission groups by rule to indicate responsibility. Your administrator configures how this is accomplished.

In addition to your administrator-established method, you can see all interferences related to design objects you have created by defining a workspace that includes only objects created by you. The **Interference List View** would then only have interferences that involve your objects. When you pick the interference on the **Interference List View**, both objects involved in the interference are highlighted even if you have not included one of the objects in your workspace definition. The software automatically retrieves the graphics for the other object from the server.

For more information related to interference checking, refer to the *Checking Interference: An Overview topic* of the user guide *IFCGuide.pdf*.

Quiz:

9. Which interference detection process generates records that are not persistent?
10. How do you identify the interferences that are generated by the Local Detect and Database Detect processes?
11. How do you start and stop the Local Detect process in your workspace?
12. How can you review the Database Detect Settings in your local machine?

Session 13: Space Management

Objective:

By the end of this session, you will be able to:

- Place space objects (folders and volumes).

Prerequisite Sessions:

- SP3D Overview
- Creating a Session File
- Defining a Workspace
- Manipulating Views
- Selecting Objects in a Model
- PinPoint Ribbon

Overview:

Space Management task allows you to create volumes to indicate a fire zone, an access way, or any other conceptual space that you want to define. You can also create volumes specifically to define drawing views. You can use the volumes to define the spatial criteria for retrieving other objects on the **Volume** tab of the **Filter Properties** of the **Define Workspace** command. Only a volume identified as an interference volume will participate in interference detection.

Since the volumes are not actually constructed, but just used to help manage the design, they are not included in the system (Plant Breakdown Structure) hierarchy. Volumes have their own separate, hierarchy displayed on the **Space** tab of the **Workspace Explorer**. You can create space folders in this hierarchy to group your volumes using the **Create Space Folder** command on the vertical toolbar. You retrieve the volumes to the workspace for review and editing by selecting them on the **Named Space** tab of the **Filter Properties**.

Creating Spaces:

There are several methods to define the geometry of a volume. Each is a different command on the Task toolbar and will be described below. A volume can be of four basic categories: an area, a zone, interference, or a drawing. The specific types of volumes nested under these categories can be customized in the catalog. Each custom type has its own set of properties. The default types currently delivered with SP3D are probably of limited usefulness to your company so customization is recommended.

The area and zone categories have no enforced functional difference. However, it is recommended that zones be used to identify volumes that may overlap. Areas should be used to identify volumes that do not overlap for the same type of area.

The interference type volume is the only volume that is checked for interference. You can associate an interference type of volume with one design object. When the design object moves, the associated volume will move.

The drawing type is used for the "Volume Drawing" automatic drawing generation mechanism. This mechanism is no longer recommended for new customers. However, the special drawing volume creation commands remain available to support customers who have used them in the past. New customers can use all rectangular volumes, regardless of type, to define drawing views in the manual Composed Drawing or the automatic Drawing by Query workflows. You will learn more about drawing views and volumes in the SP3D Drawings section.

In this session, we will describe all the methods of creating volumes and cover three of the most commonly used methods in step-by-step examples. The methods for placing spaces are:

- **Place volume by two points**: Places a rectangular volume by the diagonal corners of the volume.
- **Place volume by four points:** Places a rectangular volume by four points. The first two points define the x-axis of the local coordinate system for the volume. The third point defines the local y-axis plane and the depth. The fourth point defines the local-z height dimension.
- **Place volume by window:** Places a rectangular volume using the view boundaries (including depth) of a window.
- **Place volume by select set:** Places a rectangular volume aligned with the active coordinate system and fitted to the maximum extent of all objects in the select set.
- **Place volume by plane and offset:** Places a rectangular volume constructed relative to a plane, offsets to the front and rear of the plane, and three points defining the rectangular shape that is projected from the front plane to the rear plane.
- **Place volume by Grids:** Places multiple rectangular volumes as bounded by a set of grid and elevation planes.
- **Place volume by Bound Spaces:** Places a volume that is the combination of multiple existing volumes. The resulting volume does not affect the location of the original volumes in the space hierarchy. Volumes created by combining spaces cannot be used to define drawing views.
- **Merge volumes:** Places a volume that is the combination of multiple existing volumes. The original volumes are moved to be nested under the merged volume in the space hierarchy and cannot be used in the definition of other volumes. Merged volumes cannot be used to define drawing views.
- **Place volume along a path:** Places a volume by sweeping a cross section along a path. If you use a rectangular cross section and keep the path on a single plane, you can use the resulting volume to define a drawing view. The drawing view will look toward the plane of the path.
- **Place volume using primitive shapes:** Places a volume by using parametric shapes

defined in the Catalog.

Notes:

- Some commands can create relationships to the inputs used to define the volumes. These commands have two toggle buttons on the create ribbon to control this associative behavior. See the buttons to the right on the Volume by Two Points ribbon in Figure 1. If you establish relationships when you place the volumes, you can't move the volume but it will automatically adjust in size and position when the input objects are edited. If these buttons are not displayed, then the volumes are created by the command without relationships to other objects in the model.
- The **Associate Volume to Object** command is used to create a relationship between an interference volume and another graphic object. If the graphic object is moved, the interference volume is moved as well as long as you have not used associative points when you created the volume.

Steps for Placing Volume by Two Points:

Place a rectangular **Hazardous Zone** by using the **Place Volume by Two Points** command and referencing objects in Unit **U01.** The view of the model after placing the rectangular volume should look like Figure 1.

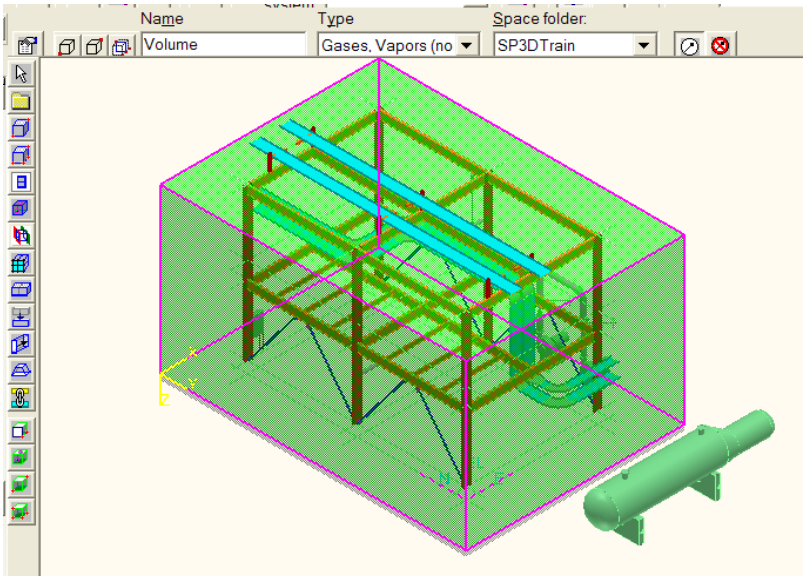


Figure 1: Output: Volume Placed by Two Points

Before you start with the procedure, define your workspace to show unit **U01** and **U01 CS** on the system tab and switch to the **Space Management** task environment.

Tips:

- If you don't select a folder from the Named Space tab, no existing volumes will be retrieved from the database. You will be able to see the volumes

you create in the examples below, but these newly created volumes will disappear when you **Refresh** your workspace. In this example, existing volumes are being left out of the workspace so they don't obscure the view.

- By default, the volume appears opaque when it is placed. To display the volume with translucent surfaces, you can apply surface style rules. A surface style rule setting all volumes to translucent green was used for Figure 1 and the rest of the figures in this tutorial. See Common session 6: Applying Surface Style Rules tutorial.

3. Activate the **PinPoint** ribbon by clicking the command button on the horizontal common toolbar.

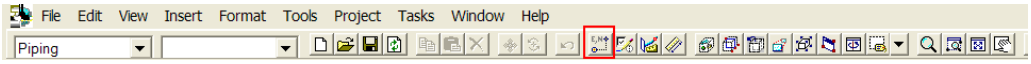


Figure 2: Activate PinPoint

4. Click the **Rectangular Coordinates** option on the **PinPoint** ribbon (Refer to Figure 3) and set the active coordinate system to **U01 CS**.
5.

Figure 3: Portion of PinPoint Ribbon

6. Click the **Place Volume by Two Points** button on the vertical toolbar.

Figure 4: Place Volume by Two Points Button

7. Click the **More...** option in the **Type** drop-down list on the **Place Volume by Two Point** ribbon to specify the type of volume to be placed from the SP3D catalog. Select the indicated type of zone.

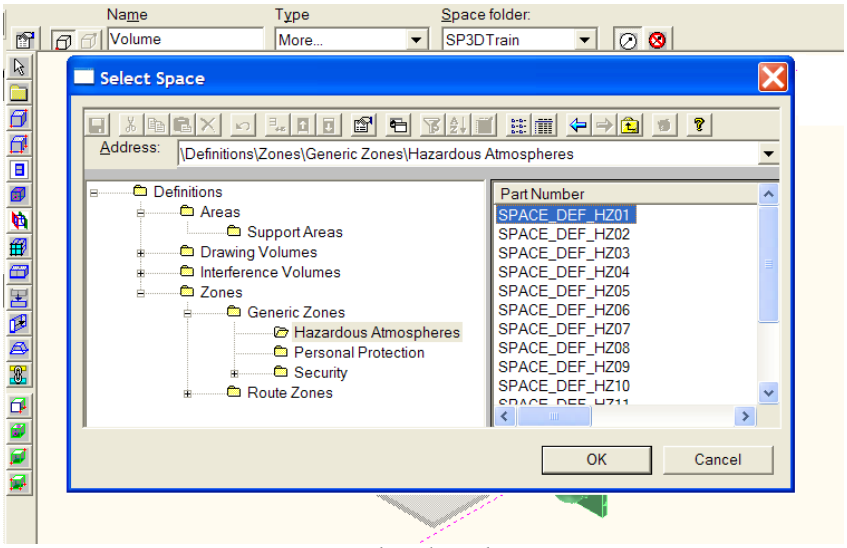


Figure 5: Select the Volume Type

Tips:

a. While you are in the "Enter First Point" smartstep (or any smartstep), you can edit the properties displayed on the create ribbon. After you finish editing the properties, the command remains in that smartstep waiting of the input.

b. Your administrator can customize the available types of zones so they are useful for your reporting and design purposes.

c. The name of the volume can be created automatically by naming rule. Open the properties dialog on the Volume by Two Points Ribbon and select the "Descriptive" name rule. This name rule will be selected by default the next time you use the command. The name is generated at the time the volume is created.

8. Now select the **SP3D Train** folder in the **Space folder** drop-down list to assign the new volume to that folder in the space management hierarchy.

   Tips:

   - If no space folders have been created, the default folder in SP3D is **SP3DTrain**.
   - Assigning volumes to a space folder helps organize the volumes you create.

9. Toggle the associative points off (see highlighted button) and then locate the corner of the slab with the smartsketch key point and click to define the first point of the rectangular volume (Refer to Figure 6).

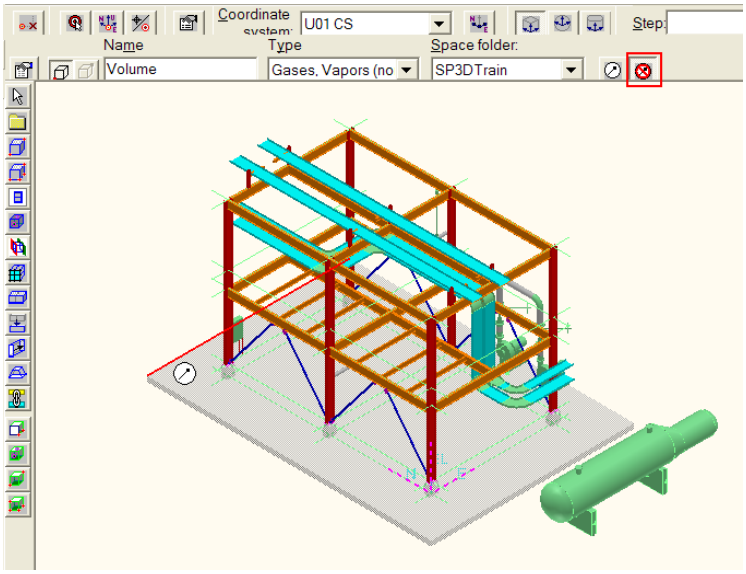


Figure 6: Locate First Point

Tips:

- You will be able to select and move the volume after you create it when the associative point toggle is off. However, the volume size will not change if the objects you selected for reference are modified. Generally, it is helpful to have associative points. The option is disabled in this example to illustrate the behavior.

10. Establish the second point that has the Easting and Northing of the diagonal slab corner and the Elevation of the structure. A click defines the point and immediately commits the volume to the database.

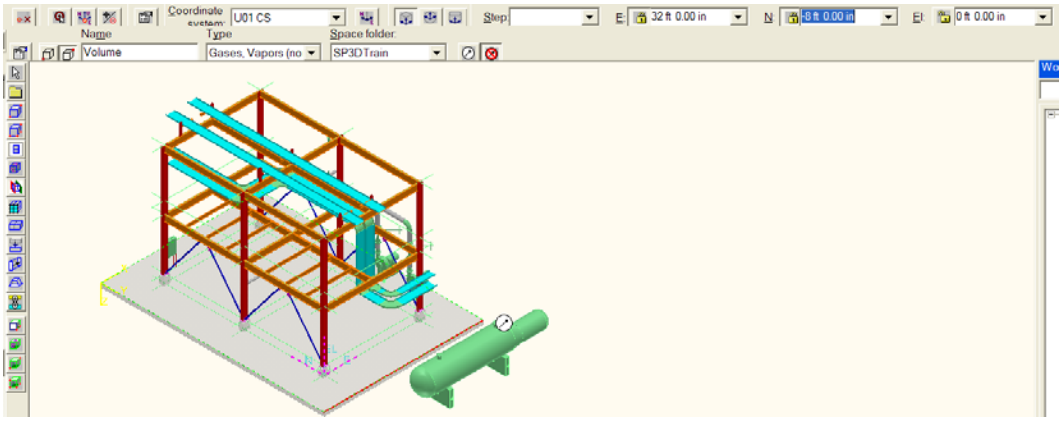


Figure 7: Lock Easting and Northing with F6 and F7

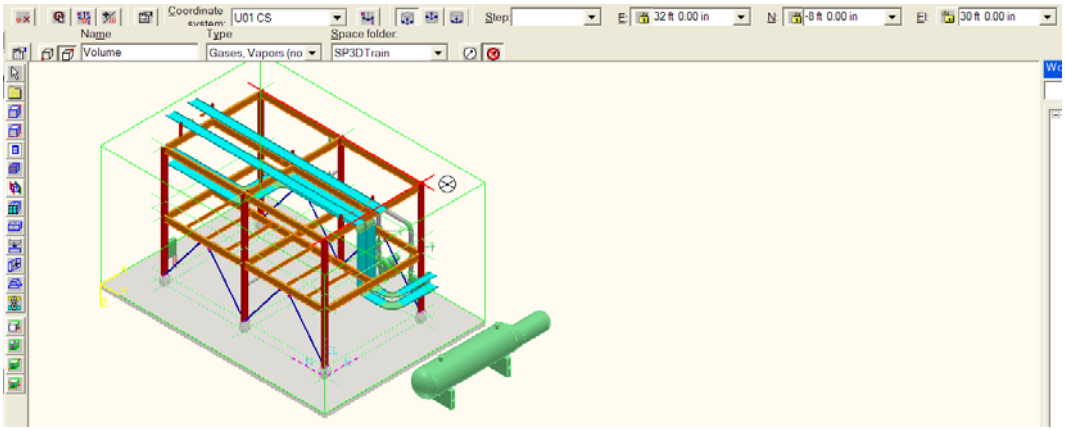


Figure 8: Locate Grid Line to Define the Elevation

Tips:

- Position the cursor over the diagonal corner of the slab and lock pinpoint Easting and Northing by entering F6 and F7 while the cursor is over the indicated smartsketch point.
- Locate any smartsketch point at the top of steel to set the elevation.
- The rectangle is aligned along the axes of the Active Coordinate System. You can get different alignment in global space by selecting the appropriate Active Coordinate System.

11. After the volume is created, **Move** it to any location. This illustrates that the smartsketch points you used during placement do not control the position. **Undo** the move.

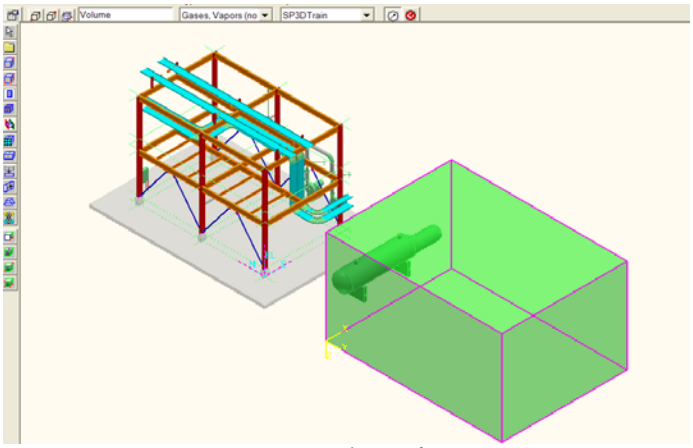


Figure 9: Volume after Move

12. **Refresh** the workspace and see that the volume disappears. Again, this happens because the workspace of this example was defined to show only objects under the System hierarchy.

Steps for Placing Volume by Four Points:

Place a volume by using the **Place Volume by Four Points** command using the same workspace as in the **Place Volume by Two Points** example. The view of the model after placing the volume should look like Figure 10.

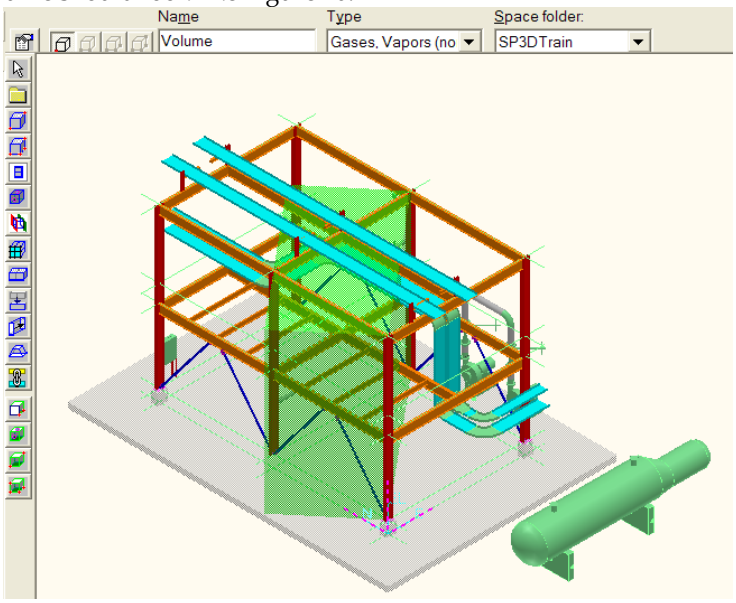


Figure 10: Output: Volume Placed by Four Points

Tip:

- You typically use the Volume by Four Points command when you want to place a volume that is not aligned with an existing coordinate system. If the volume can be aligned with a coordinate system, then it is best to use the Volume by Two points command. The Training Plant does not have a good situation where the Volume by Four Points command would be useful. ***The tutorial exercise just illustrates the orientation behavior of the command.***

1. Click the **Place Volume by Four Points** button on the vertical toolbar.

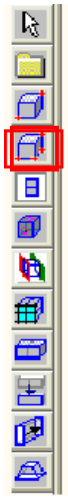


Figure 12: Place Volume by Four Points Button

2. Now specify the name of the volume, **Volume_2**, in the **Name** box on the **Place Volume by Four Point** ribbon. Entering a name on the ribbon will change the Name Rule option to User Defined.

3. Click the **More...** option in the **Type** drop-down list on the **Place Volume by Four Point** ribbon to specify the type of volume to be placed. If not currently selected by default, browse to and select the volume type shown in Figure 5. Then, click **OK**.

4. Retain the default folder **SP3D Train** as the folder to which the new volume has to be assigned.

5. Enter four points as illustrated in Figures 13 to 16.

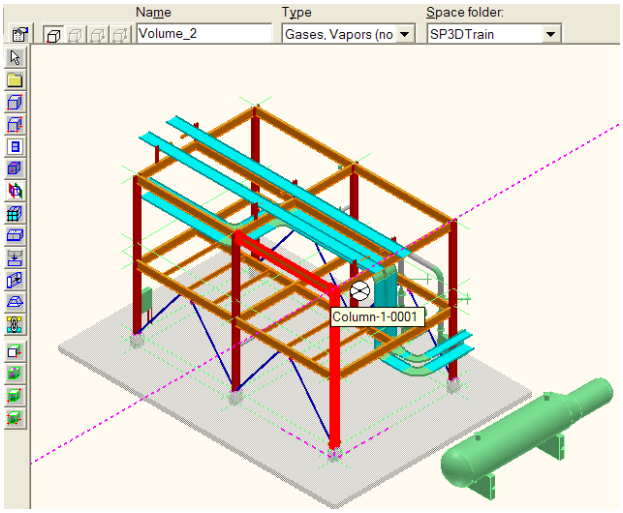


Figure 13: Enter First Point – Origin of the Volume

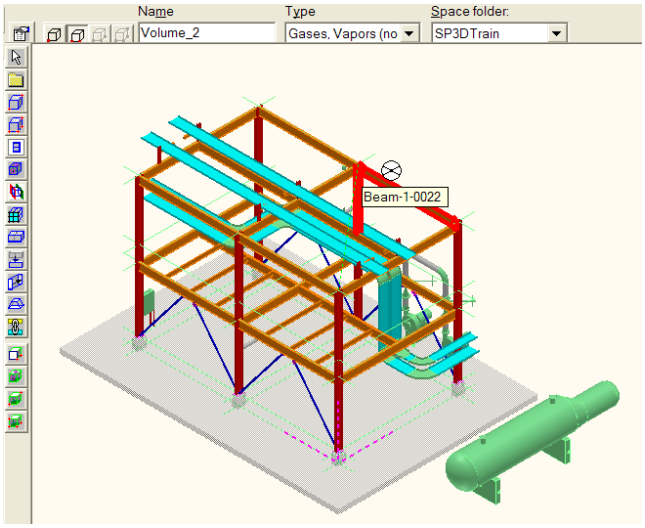


Figure 14: Enter Second Point – Local X axis Direction

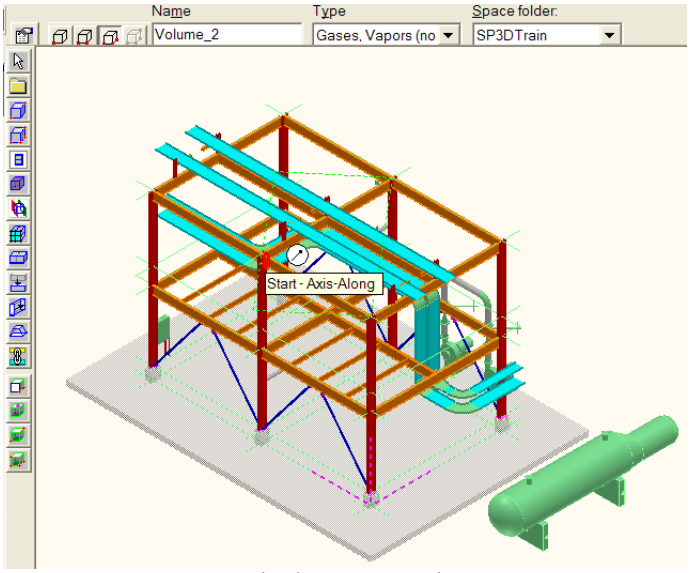


Figure 15: Enter Third Point – Local Y axis Direction

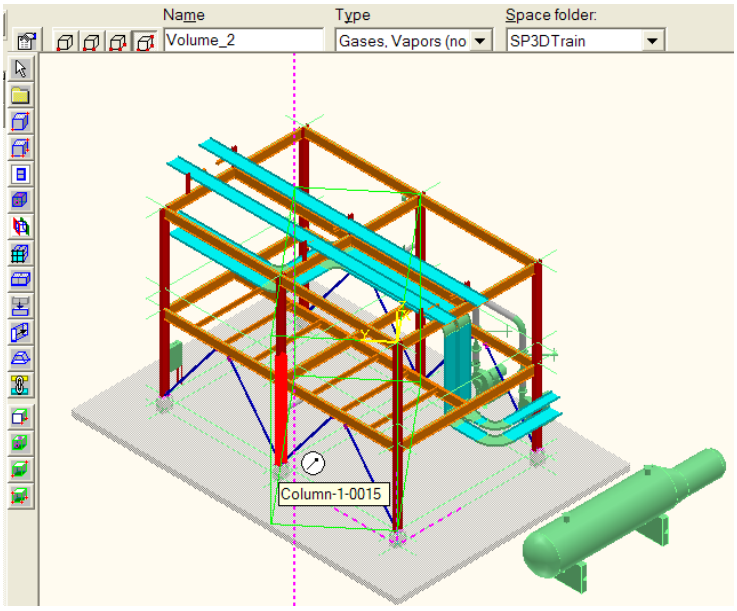


Figure 16: Enter Third Point – Local Z Height

6. Review the volume that is created (see Figure 10) and then **Undo** or **Delete** to remove this practice volume.

Steps for Placing a Volume Along a Path:

Place an interference volume for an access path in the portion of the plant occupied by units **U01** and **U04** by using the **Place Volume Along a Path** command. The view of the model after placing the volume should look like the highlighted area in Figure 17.

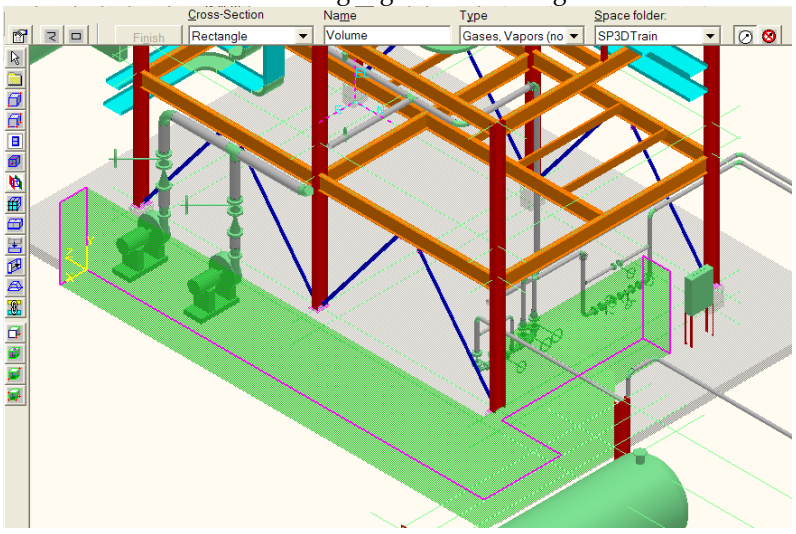


Figure 17: Output: Placed by Using Volume Along a Path

Before you start with the procedure, define your workspace to show units **U01, U04** and coordinate systems **U01CS and U04CS. S**witch to the **Space Management** task environment.

1. Click the **Place Volume Along Path** button on the vertical toolbar.

Figure 18: Place Volume Along Path Button

2. The Path ribbon is automatically displayed. Sketch the access way path shown in Figure 19.

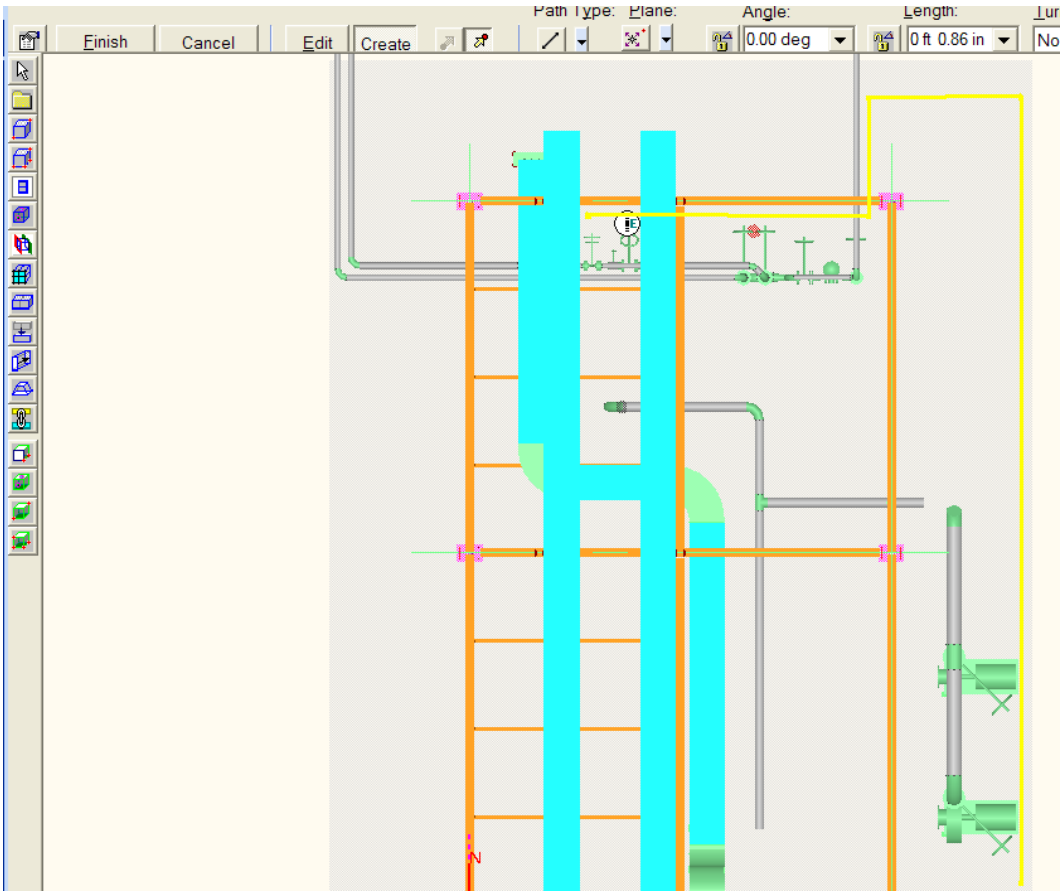


Figure 19: Defining Path of Volume

Tips:

- The exact coordinates of the path are not important for the exercise. The Imagine the path as defining the inside edge of the access way.
- Make sure the surface of the slab is located when selecting the points.
- Start near the pumps.

3. Click Finish on the Path ribbon to complete the path. The shape last used with the command in the current session will be displayed relative to the path by default. See Figure 20.

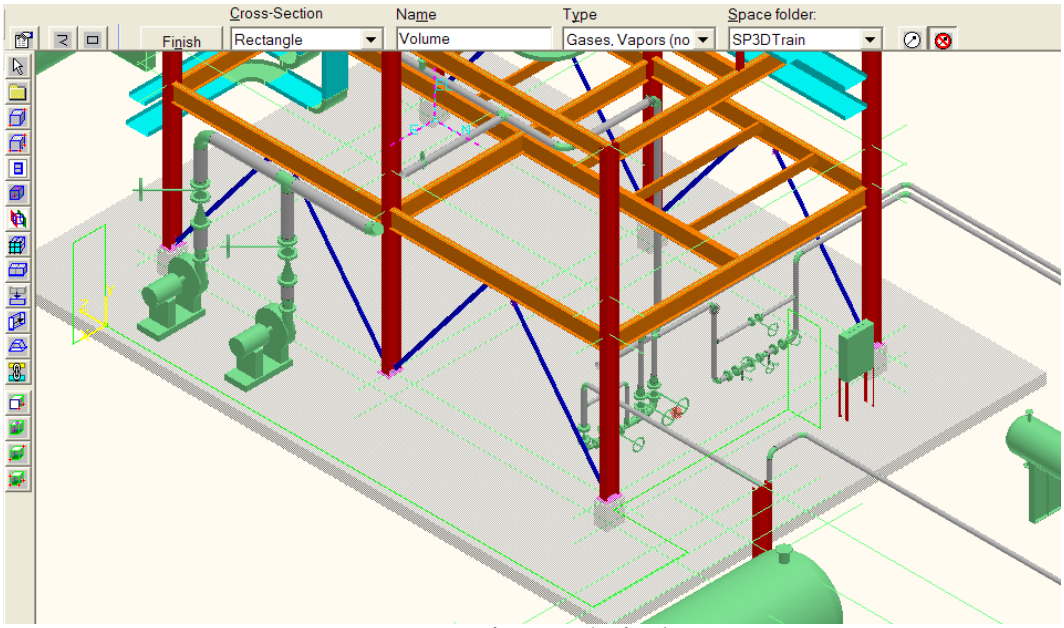


Figure 20: Defining Path of Volume

4. Select the Properties button on the command ribbon and pick the Cross-section tab. Enter the properties as shown.

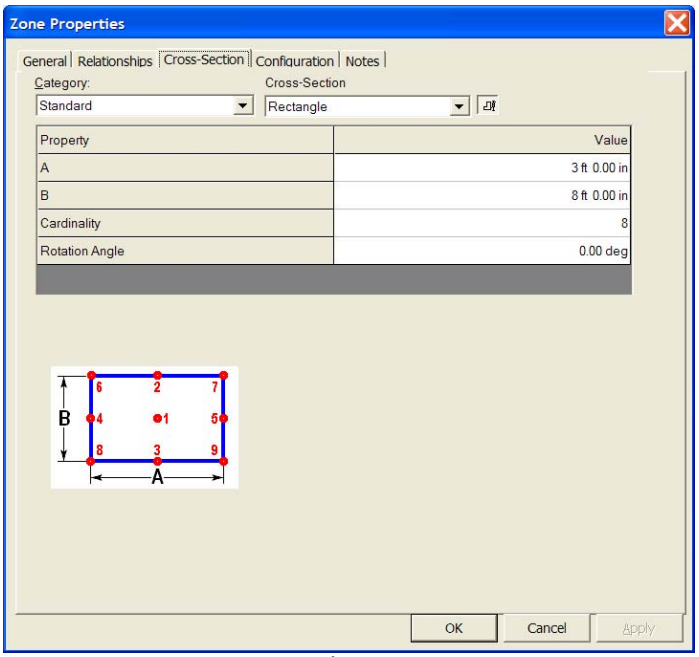


Figure 21: Define Cross Section

Tips:

- The cross section is oriented on the path at the indicated **Cardinality** point. The orientation of the sketch is looking from the first point to the second point of the

path.

- If the path does not orient correctly the first time. Restart the command after having executed the desired cross section and cardinal point once. This sets the defaults correctly for the next execution of the command.

5. Now specify the name of the volume **Volume_3** in the **Name** box on the **Place Volume Along Path** ribbon.

6. Click the **More...** option in the **Type** drop-down list on the **Place Volume Along Path** ribbon to specify the type of volume to be placed as indicated in Figure 22.

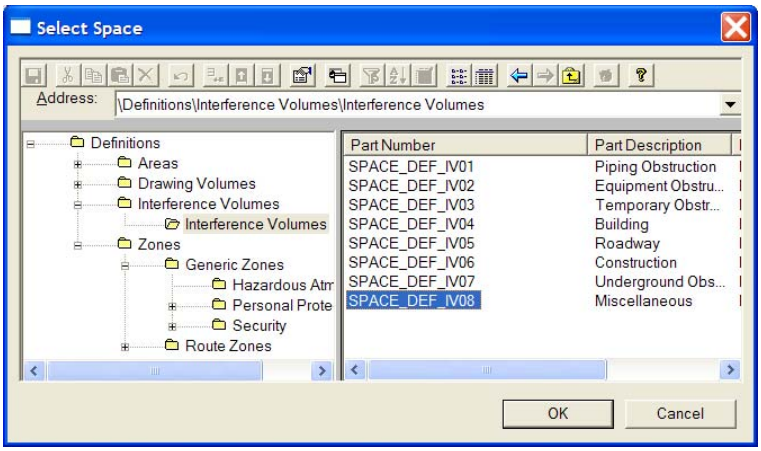


Figure 22: Select Type as Interference Volume

7. Leave the space folder to the defaulted **SP3DTrain**.
8. Click the **Finish** button on the **Place Volume Along Path** ribbon to place the volume. See Figure 17.

For more information related to space management, refer to the *Space Management: An Overview* and *Creating Space Objects: An Overview* topics in the user guide *SpaceMgmtUsersGuide.pdf.*

Session 14: Inserting Reference Files

Objective:

By the end of this session, you will be able to:

- Insert a reference file in a model.

Prerequisite Sessions:

- SP3D Overview
- Creating a Session File
- Defining a Workspace by Simple Filter
- Defining a Workspace by Volume Filter
- Manipulating Views
- Selecting Objects in a Model

Overview:

You can reference geometry files from other formats in SP3D. To insert a reference file into the model, the file must reside on a directory shared on the network. The default share location is the Symbol share. In a global workshare configuration, your administrator updates the Symbol share at each remote location to match that on the Host when changes are made to files on the Symbol share. You can still attach a reference file that is in a share other than the Symbol share. If you do this, however, you will not have access to the reference files at remote workshare locations that can't access the share you used. The software will warn you if you choose a share other than the Symbol share. You must share the file on your computer or on the network. Your administrator will tell you what shares you may use. MicroStation V7 or Microstation V8 files with the .dgn extension or 2D AutoCAD files with the .dxf and .dwg extensions are currently supported.

MicroStation design files use the concept of master units: sub units: positional units (MU: SU: PU) to express length dimension. All distances in SP3D are stored in meters. When you insert a reference file in SP3D, the MU: SU: PU working units are used in the conversion of units into meters. If the MU: SU: PU units are not defined within the MicroStation file, an error appears. The recommended working units in a reference file are 1:1000:80 for the metric system (m: mm) and 1:12:2032 (ft: in) for the English system.

Steps for Inserting a Reference File:

Insert a MicroStation V7-format .dgn file in a model.

10. Define your workspace using Unit **U04, U04 CS** and **Reference** from the **Reference** tab. Name the filter **U04 U04CS Ref.**

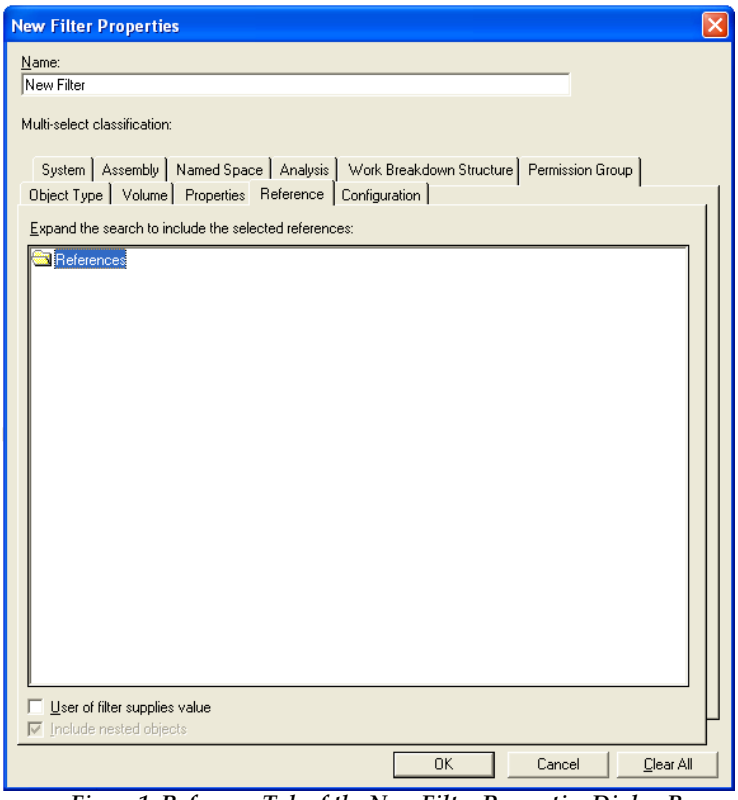


Figure 1: Reference Tab of the New Filter Properties Dialog Box

Tips:

- The Reference node on the reference tab will only appear if a reference file had been inserted at anytime in the past into the current model. The reference file can be later deleted and the reference node will still be shown. This is because the reference node object is not created until a reference file is first inserted.
- If you do not see a Reference node on the Reference tab of the filter properties dialog, wait and redefine your filter as described above **after** you have inserted your first reference file.

11. Select **Options…** from the **Tools** pull-down menu. Under the **General** tab check the **Reference** box and click **Ok.**

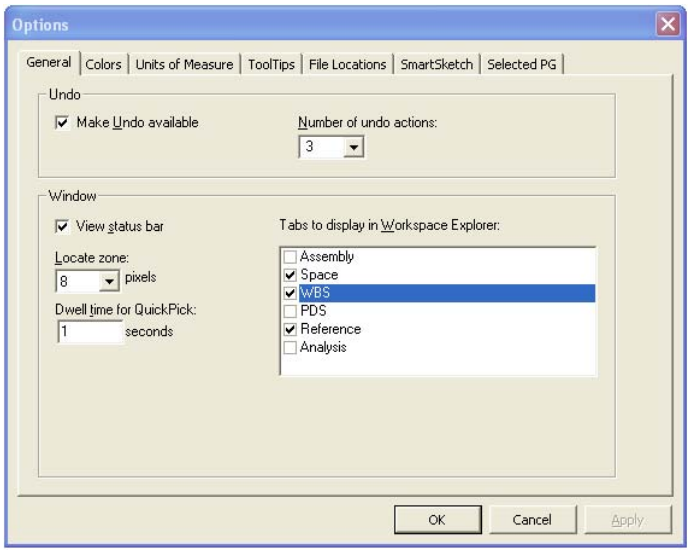


Figure 2: General Tab of the Options Dialog Box

12. To access the **Reference** tab, save the session, close and reopen it or switch from the **Common** task to the **Catalog** task and back to the **Common** task to see the new **Reference** tab.

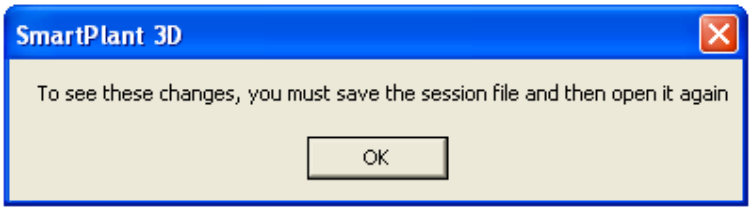


Figure 3: Message Box Prompting You to Save the Session

13. Click the **Insert > File** command.

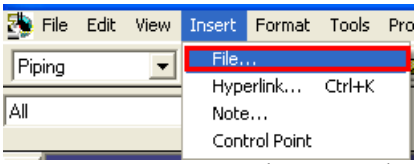


Figure 4: Insert > File Command

14. The **Insert File** dialog box appears. Browse to select the file using a UNC path connection (or enter UNC path file name). It is important to use UNC path so that the model server will have proper access to the reference file.

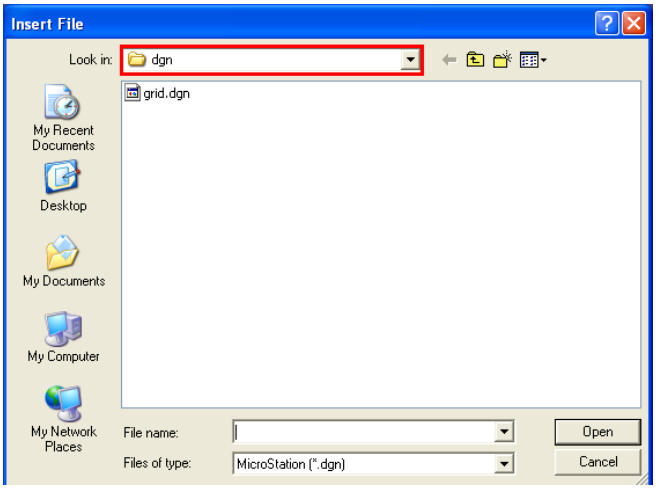


Figure 5: Insert File Dialog Box

Tip:

- You can select multiple files to be inserted by holding down the **CTRL** or **SHIFT** key while selecting the files.

15. Click **Open** to insert the selected reference file.

After you insert the file, an additional tab, **Reference**, is added to the Workspace Explorer to display the hierarchy of the referenced data. The **Reference** tab also starts appearing in the **Filter Properties** dialog box. You can define your workspace to include the reference files.

Tips:

- The location and orientation of the inserted files will match the currently active coordinate system after initial attachment you can move and rotate the attachment by selecting the attachment and using the move and rotate commands. You can view the attachments on the **Reference** tab of the **Workspace Explorer.** You add this tab by selecting the **Reference** tab option on **Tools > Options** (General page). After selecting the **Reference** tab option, as a message will remind you, you must close the workspace and open it again to see the **Reference** tab.
- When references are attached, the **Reference** tab starts appearing in the **Filter Properties** dialog box. You can define your workspace to include the reference files.
- To detach a reference file from an SP3D model, select the file in the Workspace Explorer, delete it, and then refresh the workspace.
- When troubleshooting reference files, ensure that the file or folder is shared with proper permissions. Also, make sure that the working units of the reference file are supported. The file must have units that the software can read and convert.

For more information related to inserting reference files, refer to the topic *Using the Insert Menu: An Overview* in the user guide *CommonUsersGuide.pdf.*

Session 15: Placing Control Points

Objective:

By the end of this session, you will be able to:

- Place control points.

Pre-Requisite Sessions:

- SP3D Overview
- Creating a Session File
- Defining a Workspace
- Manipulating Views
- Selecting Objects in a Model
- Using PinPoint

Overview:

A Control Point defines a 3D coordinate and is associated to another object. Its geometry is a sphere with user-defined diameter in the Reference Aspect. It is not included in interference detection. The control point is automatically assigned to the same permission group as the parent object. If you delete the parent object, the control point is automatically deleted. Any SP3D object can be the parent including the System hierarchy objects. It has properties and associated notes. The Type and Sub Type properties indicate the meaning or intended use of the control point.

The Control Point is currently used for:

- Labels on drawings – Drawing styles can detect the control point of any desired type and subtype and output the note associated with the control point as a label with the leader pointing to the location of the control point.
- Points for automatic dimensioning on drawings – Piping Isometric and general Orthographic drawings place dimensions automatically as defined by the dimensioning style rules of the drawing. Orthographic drawings can dimension to control points of any type and subtype. Isometric drawings can dimension to control points that are associated to straight pipe and have a note of the type, **Fabrication**. The text content of the note will be output to the Isometric drawing.
- Division point in piping – The spooling and automatic WBS Item creation commands in piping use a control point of a specific type and subtype. The details of what types to use are covered in the piping tutorials.
- Recording points of design interest with notes – Any design note you might want to record that refers to a coordinate in the model can be added using control points.

Steps for Placing Control Points:

Place a control point on an Electrical Device, an equipment component, of Unit **U01**. The view of the model after placing the control point should look like Figure 1.

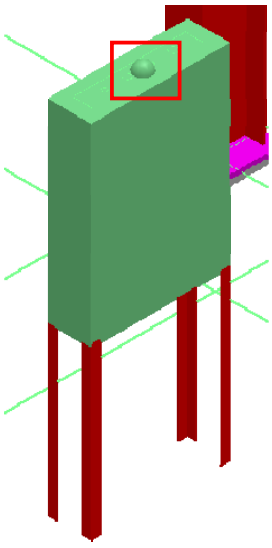


Figure 1:Final Output: Placed Control Points in a Model

Define your workspace to show Unit **U01**.

16. Click the **Insert> Control Point** command.

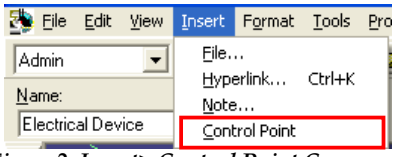


Figure 2: Insert> Control Point Command

17. Select **Electrical Device** as the parent object of the control point – the object on which the control point has to be placed.

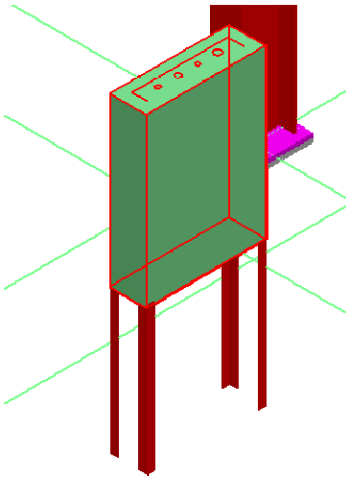


Figure 3: Selected Electrical Device

18. As the control point is being placed on an equipment component, classify the control point as **Mechanical Equipment** in the **Subtype** drop-down list of the **Control Point** ribbon.

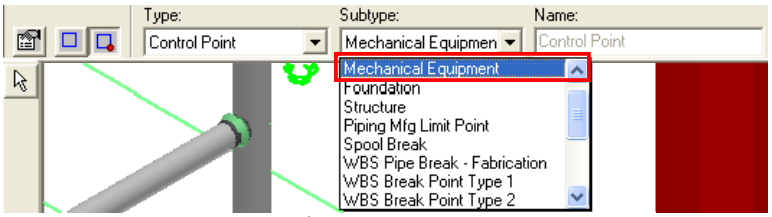


Figure 4: Subtype Drop-Down List

Tips:

- The Types are: Control Point, Key Point, and Insertion Point. Only the Control Point type has fixed software meaning for piping isometrics and WBS group creation when related to piping objects. The automatic dimensioning and labeling of orthographic drawings can be configured to use any type of control point. Your administrator will give you instruction for which types to use to drive automation of your custom drawing types.
- You can use PinPoint to locate the control point at a specific coordinate or at some specific offsets from a location on the parent object.

19. Click the cable tray port of the **Electrical Device** to place the control point as shown in the highlighted area in Figure 5.

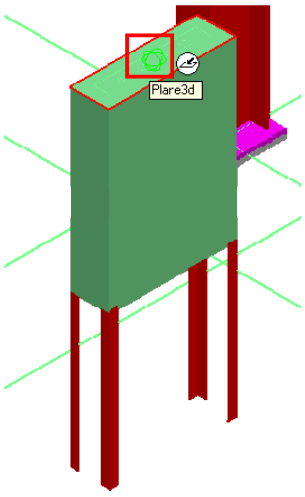


Figure 5: Cable Tray Port of the Electrical Device

Tips:

- After placing control points in a model, you can view them graphically by setting your view to show the Reference Aspect (see **Format View** command).
- You can select the control point graphically when the reference aspect is displayed. If the control point is inside the volume of the object or difficult to graphically select, you can select it in the Workspace Explorer. First, select the parent object of the control point. The Workspace Explorer automatically scrolls so you can see the selected object. The control point is nested under its parent object in the Workspace Explorer. You can then easily see and pick the control point from the Workspace Explorer rather than graphically.
- You can see the control points related to an object by looking at the Relationships tab of the object.

20. With the control point selected, select the **Properties** command to open the **Control Point Properties** dialog box.

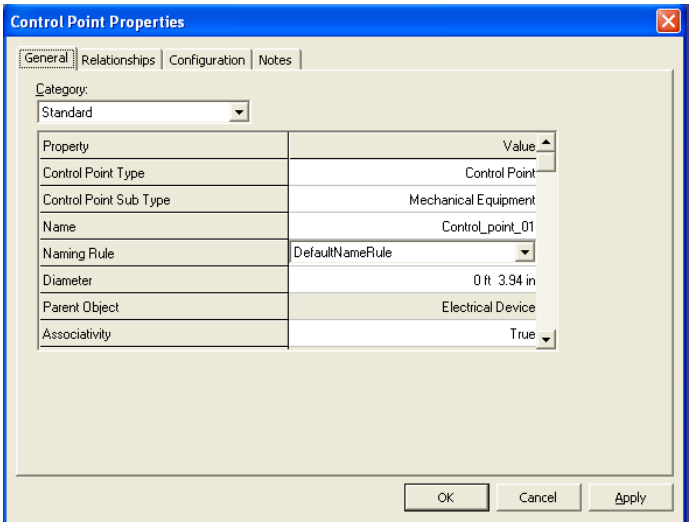


Figure 6: Control Point Properties Dialog Box

Tips:

- You can change the size of the control point sphere by changing the diameter property.
- When the Associativity property is set to true, the control point will move when you move the parent object. You cannot move the control point relative to the parent object. When set to false, the control point will not move when you move the parent object. You can also select and move the control point relative to the parent object. Once you toggle the associativity of the control point to False, you can't toggle it back to True.

For more information related to placing control points, refer to the topic *Using the Insert Menu: An Overview* in the user guide *CommonUsersGuide.pdf.*