SmartPlant 3D
Practice Labs for Common

Process, Power & Marine

INTERGRAPH

Table of Contents

SP3D Common Practice Lab: Creating a Session File

Creating a Session File with a View of a Model:

Create a session file with a view of a portion of the model and save the file on your desktop.

Steps:

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**.

3. Click **Save**.

SP3D Common Practice Lab: Defining a Workspace by a System Filter

Before going through the remaining SP3D Common sessions, define your workspace to include all objects in the SP3Dtrain model database:

1. Start **SP3D** software by using the command **Start > Programs > Intergraph SmartPlant 3D > SmartPlant 3D**.

2. In the **New** dialog box, select the **EnglishUnits** or **MetricUnits** template and then, click **OK**.

3. Click the **File** menu and select the **Define Workspace** command.

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

5. In the **Select Filter** dialog box under **Plant Filters**, select **All** and click **OK**.

6. Click **OK** in the **Define Workspace** dialog box.

7. Select the **View > Fit** command.

Now, you will hide some existing modeled objects from the workspace before starting the session. Use the filter mechanism to select the existing modeled objects.

8. Select the **Tools > Select by Filter** command to open the **Select Filter** dialog box.

9. Select and expand the **For Instructors Only** folder.

10. Select the **Common Tutorial Session - Select and Hide** filter and click **OK**.

11. Click the **Tools > Hide** command to hide the selected objects.

Defining a Workspace by Creating 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.
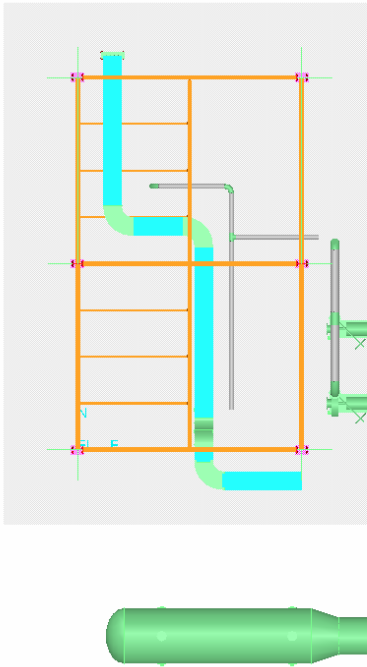
Figure 1: Output: After Defining Workspace for Unit U01

Steps:

1. Click the **File** menu and select the **Define Workspace…** command. The **Define Workspace** dialog box appears.

2. In the **Filter** drop-down list of the **Define Workspace** dialog box, select the **Create New Filter…** option. The **New Filter Properties** dialog box is displayed.

3. 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**.

4. Click **OK** to complete the filter definition process.

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

6. Click the **Fit** button on the **Common** toolbar to fit all objects retrieved by the simple filter **Unit 1** in the active view.

SP3D Common Practice Lab: Defining a Workspace by Volume Filter

Defining a Workspace by 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.
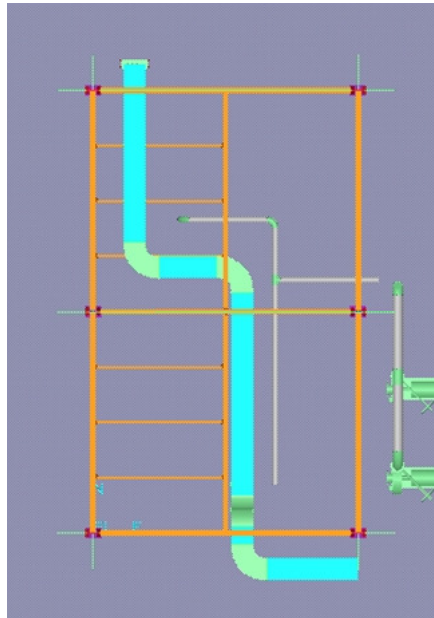
Figure 1: Final Output: Objects in the Defined Workspace by Volume Filter

Steps:

1. Define the workspace 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.

2. 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**.

3. To retrieve only objects in the selected systems with geometry in **Volume U01**, click the **Volume** tab and select **Volume U01**.

4. Click **OK**.

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

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

7. Click the **Fit** button on the Common toolbar to view all the objects retrieved by the Volume filter.

SP3D Common Practice Lab: Manipulating Views

Manipulating Views in the Active Window:

Use the view manipulation commands available on the **Common** toolbar for changing views, clipping views, and rotating views.

Steps:

1. Click the **File** menu and select the **Define Workspace**… command. The **Define Workspace** dialog box appears.

2. Select the **More**… option in the **Filter** drop-down list of the **Define Workspace** dialog box. The **Select Filter** dialog box is displayed.

3. In your training plant, select **U03** from **Plant Filters > Training Filters** in the **Select Filter** dialog box and click **OK**.

4. Click **OK** in the **Define Workspace** dialog box.

5. Select the **View > Fit** command.

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

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

8. Change the view to **Isometric** view and magnify the area highlighted in Figure 1 by clicking the **Zoom Area** button on the **Common** toolbar. Choose an area that just includes the vertical tank and the pump.
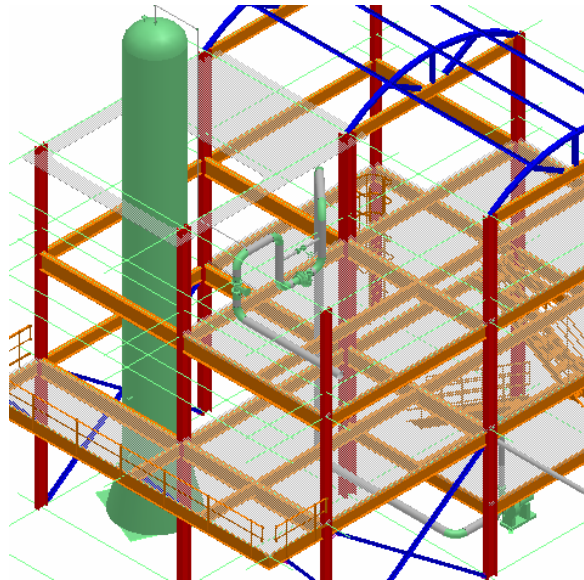
Figure 1: Isometric View

9. Click the **Named Views** button on the **Common** toolbar.

10. In the **Named Views** dialog box, type **Area 1** in the **Name** box and then type **Area around the equipment T-101** in the **Description** box.

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

12. After defining the Named View, you can isolate the equipment **P-101** to work with it. 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 idea is simply to pick objects to define the range. 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.

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

14. 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 so you see the third story bay.

15. Set the active view to **Isometric** view using the **Common Views** control. Select the **Active View Control** command on the **Common** toolbar if you want to modify the view parameters of the active view from a separate window display of the view parameters. 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.

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

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

18. For this example, from the standard **Isometric** view, select the vertical pipe connected to the pump discharge nozzle and key in **15 deg** in the field on the **Rotate** ribbon.

SP3D Common Practice Lab: Selecting Objects in a Model

Selecting Individual Objects by Using the Select Command:

Select Pipeline **400-P** of Unit **U04** in your workspace by using the **Select** command. After selecting the specified object, the view of the structure should resemble Figure 1.
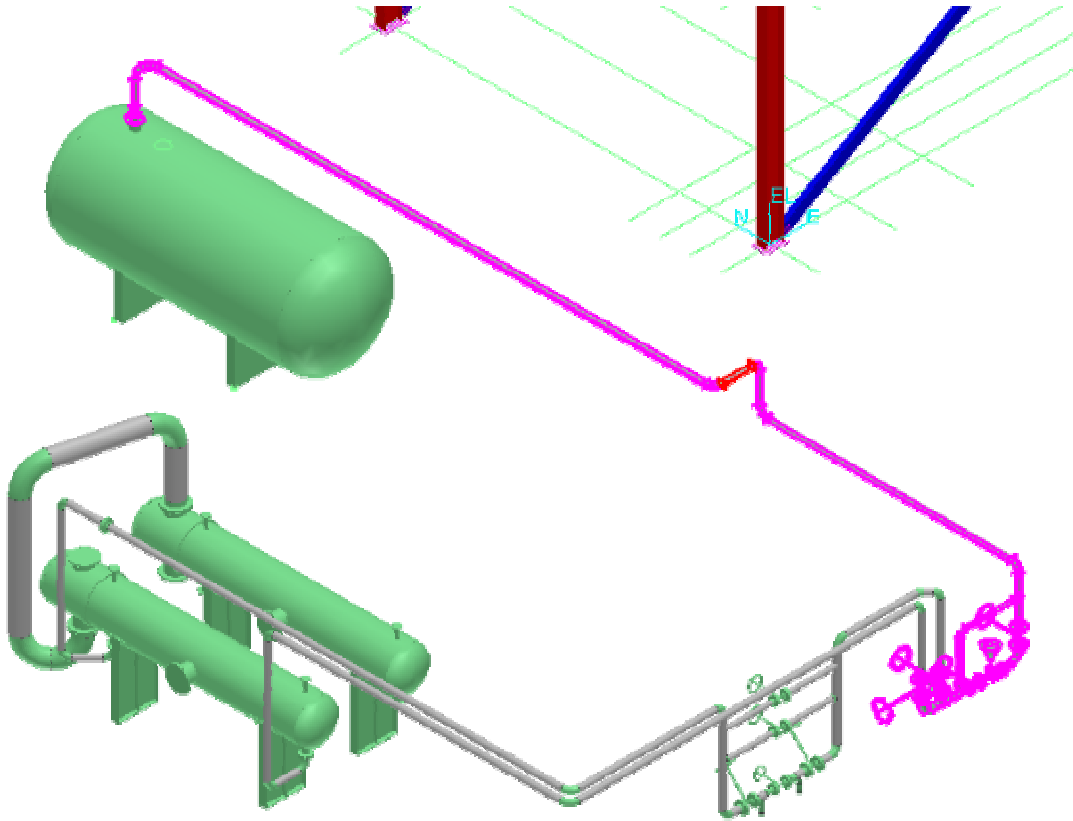
Figure 1: Selecting a Specified Object by Using the Select Command

Steps:

1. Define your workspace to display Unit **U04** and coordinate system **U04 CS.** In your training plant, select **U04** from **Plant Filters > Training Filters** in the **Select Filter** dialog box.

2. Select the **View > Fit** command.

3. Select the **Piping** task by using the **Tasks > Piping** command.

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

5. Select the filter **Pipelines** in the **Locate Filter** drop-down list to locate only the Pipeline objects. The pipeline is a system that uses the graphics of all the components in the system as its graphic visualization. It does not have any graphics of its own to directly manipulate.

6. 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.

Selecting Objects by Using Inside Fence:

Select Pipeline **403-P** of **Unit U04** in your workspace by using **Inside Fence** option of the **Select** command. After selecting the specified objects, the view should resemble Figure 2.
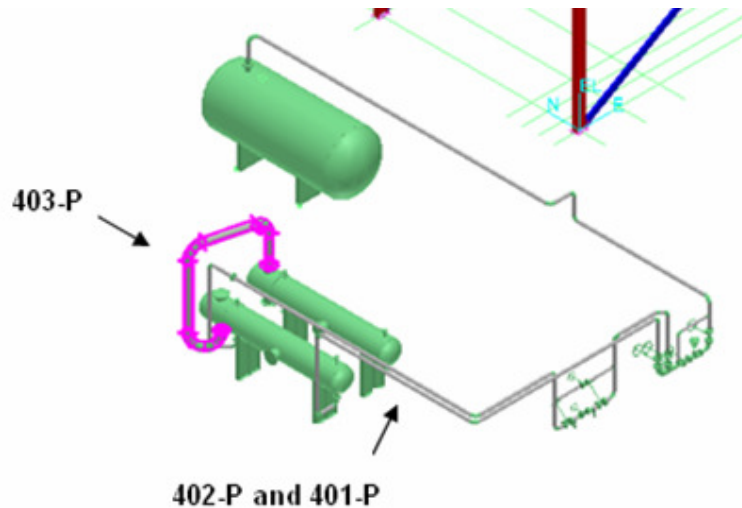
Figure 2: Selecting Specified Objects Using the Inside Fence Option of the Select Command

Steps:

1. Select the **Piping** task by using the **Tasks > Piping** command.

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

3. Select the filter **Pipelines** in the **Locate Filter** drop-down list to locate only the pipelines. 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.

4. Place a fence around Pipeline **403-P** by dragging the dashed fence lines around Pipeline **403-P**. The fence may cover portions, but not all of Pipelines **402-P** and **401-P**. After creating the fence around the pipeline and releasing it, the Pipeline **403-P** that lies fully inside the fences is selected.

Selecting Objects by Using Overlapping Fence:

Select Pipelines **403-P**, **402-P**, and **401-P** of **Unit U04** in your workspace by using the **Overlapping fence** option of the **Select** command.

Steps:

1. Select the **Piping** task by using the **Tasks > Piping** command.

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

3. 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.

4. Place a fence around Pipeline **403-P** by dragging the dashed fence lines around Pipeline **403-P**. The fence should cover the parts of Pipelines **402-P** and **401-P**, as highlighted in Figure 3.
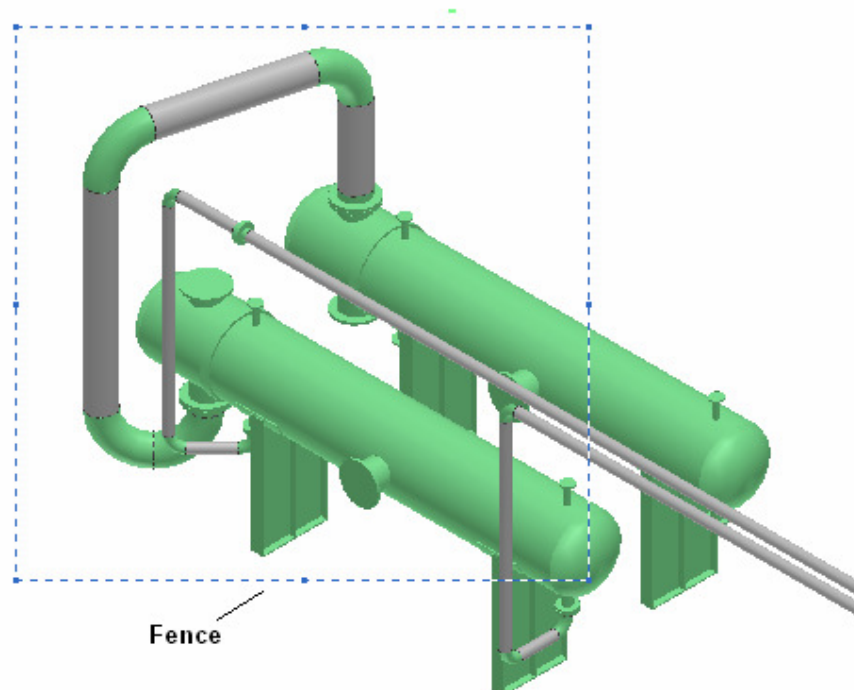
Figure 3: Overlapping Fence

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

Selecting Objects by Using Select by Filter:

Select all the structural objects in Unit **U01** of your workspace by using an object type filter. After

selecting the specified objects, the view of the structure should resemble Figure 4.
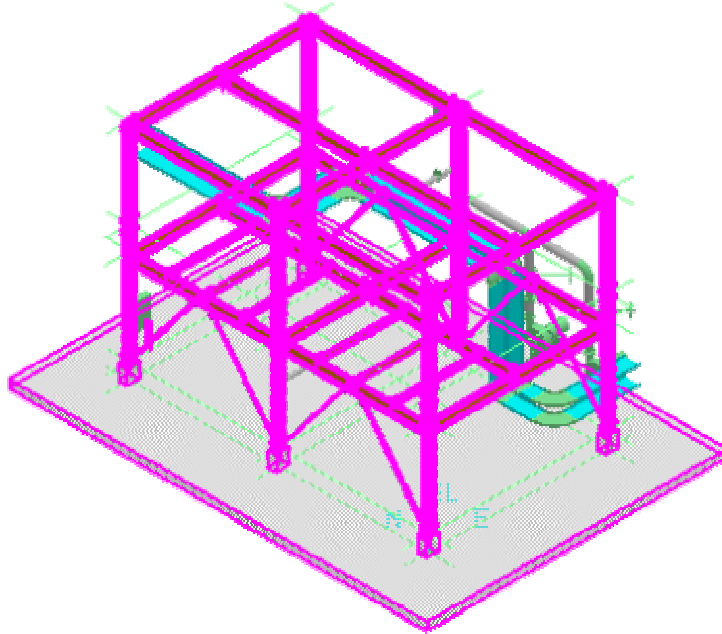
Figure 4: Selecting a Structural Object by Using an Object

Steps:

1. Define your workspace to display Unit **U01** and coordinate system **U01 CS**. In your training plant, select **U01** from **Plant Filters > Training Filters** in the **Select Filter** dialog box.

2. Select **View > Fit** command.

3. Select the **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.

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

5. Click **OK** to select all structural objects in the defined workspace.

6. Select the **Common** task by using the **Tasks > Common** command.

SP3D Common Practice Lab: Applying Surface Style Rules

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 of the structure should resemble the highlighted area in Figure 1.
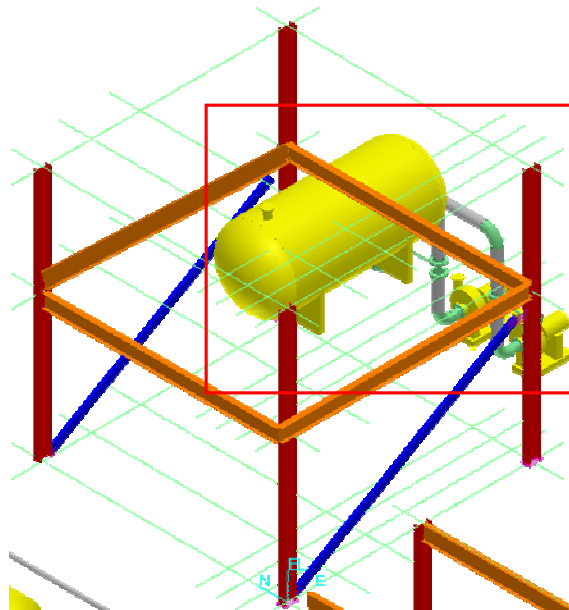
Figure 1: Final Output: After Applying Surface Style Rule to Equipment Objects

Steps:

1. Define your workspace to display Unit **U04** and coordinate system **U04 CS**. In your training plant, select **U04** from **Plant Filters > Training Filters** in the **Select Filter** dialog box.

2. Select the **View > Fit** command

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

4. 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.

5. The **Surface Style Rules Properties** dialog box appears. It displays 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.

6. Click the **Filter** drop-down list and select the **More…** option to specify the filter.

7. 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.

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

9. 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**.

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

11. Click **OK**.

SP3D Common Practice Lab: Assigning Objects to WBS Items

Manually Assigning Objects to a WBS Item:

Assign all the structural objects of Unit **U02** to a WBS Item, **Civil Contract1**, of your active project.

Steps:

1. Define your workspace to show Unit **U02**, **U02 CS** and projects under the WBS hierarchy. This involves the following steps:

- First, create a simple filter that includes Unit **U02** and **U02 CS**.

    o   Click the **More…** option in the **Define Workspace** dialog box to open the **Select Filter** dialog box.

    o   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.

    o   In the **New Filter Properties** dialog box, type **Unit 02** in the **Name** box. Then, expand **A2** and **CS**. Press and hold the CTRL key, and then, select **U02** and **U02 CS**.

    o   Click **OK** to return to the **Select Filter** dialog box.

- Next, create a simple filter that includes projects under the WBS hierarchy.

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

    o   In the **New Filter Properties** dialog box, type **WBS** in the **Name** box.

    o   Click the **Work Breakdown Structure** tab and select **SP3DTrain** node to include all projects under the WBS hierarchy.

    o   Click **OK** to return to the **Select Filter** dialog box.

- Finally, create a compound filter to include the previous filters.

    o   Select **My Filters** and click the **New Compound Filter** icon to open the **New Compound Filter Properties** dialog box.

    o   In the **New Compound Filter Properties** dialog box, type **Unit 02** and **WBS** in the **Name** box.

    o   Select **Unit 02** under **My Filters** and click the **Add to String** button.

- o Select the **Or** button as the operator to use between each filter name.

- o Select **WBS** under **My Filters** and click the **Add to String** button.

- o Click **OK** to return to the **Select Filter** dialog box.

- o Select **Unit 02** and **WBS** compound filter and click **OK**.

- o Click **OK** in the **Define Workspace** dialog box.

2. Select the **View > Fit** command.

3. Select the **Tools > Select by Filter**… command.

4. Under **Catalog Filters** in the **Select Filter** dialog box, expand **Default Filters > SP3D Object Filters > Object Types** and then, select the **Structure** filter. Then, click **OK**.

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

6. In the **Select Active Project** dialog box, select **Project1**. Click **OK**.

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

8. Click **Close** to close the **Claim** dialog box.

9. Click the **WBS** tab on the **Workspace Explorer**.

10. Make sure you have set the filter to **All** in the **Locate Filter** drop-down list.

11. Right-click **Project1** in the **Workspace Explorer** and click the **Create WBS Item** command.

12. The **Create WBS Item** dialog box appears. Set the following properties:

    - **WBS Type: Contract**
    - **WBS Purpose: Civil**
    - **Exclusive: True**
    - **WBS Assignment: System**
    - **Name: Civil Contract1**

13. Click **OK**.

14. Select the **Tools > Select by Filter…** command.

15. Under **Catalog Filters** in the **Select Filter** dialog box, expand **Default Filters > SP3D Object Filters > Object Types** and then, select the **Structure** filter. Then, click **OK**.

16. Click the **Project** menu and select the **Assign to WBS…** command, with all the

structural objects highlighted.

17. In the **Assign to WBS** dialog box, expand **Project1** and select **Civil Contract1**. Then, click **OK** to create the relationships among the objects and the selected WBS Item.

Automatically Assigning Objects to a WBS Item:

Assign objects to a project and WBS Item automatically as the objects are created.

Steps:

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

2. In the **Select Active Project** dialog box, select **Project1\Contract2**. Click **OK**.

3. Select the **Tasks > Equipment and Furnishings** command.

4. In the **Active Permission Group** drop-down list, select the **Equipment** option.

5. Click the **Place Equipment** button on the vertical toolbar.

6. In the **Select Equipment** dialog box, expand the folder **\Equipment\Safety\Safety Showers** until you see the part **SafetyShower01-E**.

7. Select the part and click **OK**.

8. The **Equipment Properties** dialog box appears.

9. Key in **S-001** in the **Name** field.

10. Select the **More…** option from the **System** drop-down list to specify the system to which the equipment belongs. The **System** dialog box appears.

11. Select **A2 > U02 > Equipment** system to indicate where the object will be placed. Then, click **OK** on the **System** dialog box.

12. To define the position of the object, select the **Position and Orientation** category in the **Category** drop-down list.

13. Set the following properties:

     - **East: 35 ft**
     - **North: -5 ft**
     - **Elevation: 0 ft**
     - **Bearing: 90 deg**
     - **Pitch: 0 deg**
     - **Roll: 0 deg**

14. Click **OK** in the **Equipment Properties** dialog box to place the equipment in the model.

15. Right-click and select the **Properties** option on the shortcut menu to open the properties page.

16. Review the properties of the equipment on the **Relationship** tab of the **Equipment Properties** dialog box to verify whether the equipment created has been assigned to the WBS item **Contract2**.

17. Click **Undo** or **Delete** on the **Common** toolbar to remove this equipment.

18. Select the **Common** task by using the **Tasks > Common** command.

SP3D Common Practice Lab: Space Management

Placing Volumes 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 resemble Figure 1.
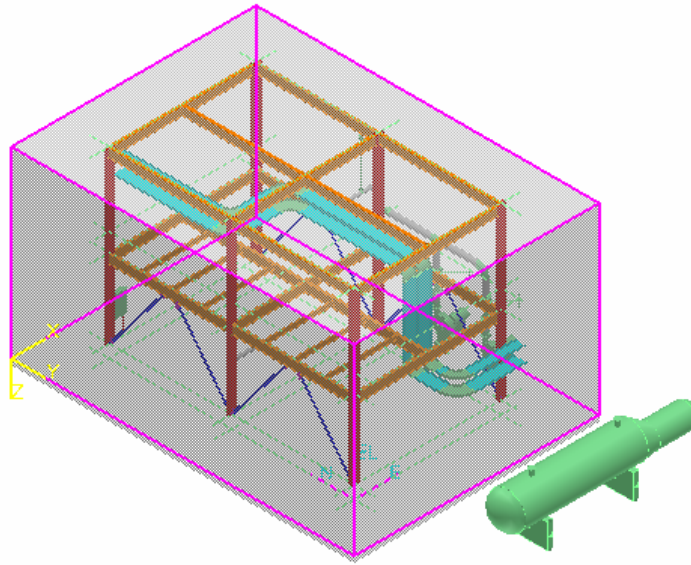

Figure 1: Output: Volume Placed by Two Points

Steps:

1. Define your workspace to include Unit **U01** and **U01 CS**. In your training plant, select **U01** from **Plant Filters > Training Filters** in the **Select Filter** dialog box.

2. Select the **View > Fit** command.

3. Switch to the **Space Management** task.

4. Activate the **PinPoint** ribbon by clicking the **PinPoint** button on the **Common** toolbar.

5. Click the **Rectangular Coordinates** option on the **PinPoint** ribbon and set the Active Coordinate System to **U01 CS**.

6. Set the target by using the **Set Target to Origin** option on the **PinPoint** ribbon.

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

8. 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. In the **Select Space** dialog box, expand the **Zones** folder, select **SPACE_DEF_HZ01** from **Hazardous Atmospheres**, and click **OK**.

9. Now select the **SP3DTrain** folder in the **Space folder** drop-down list to assign the new volume to that folder in the space management hierarchy.

10. Toggle the associative points off, as highlighted in Figure 2. Then, locate the corner of the slab with the SmartSketch key point and click to define the first point of the rectangular volume, as shown in Figure 2.
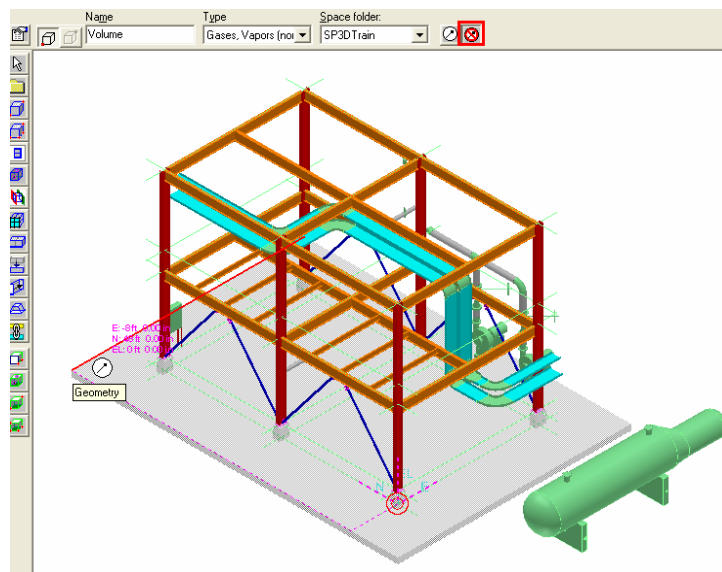

Figure 2: Locate First Point

11. 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.
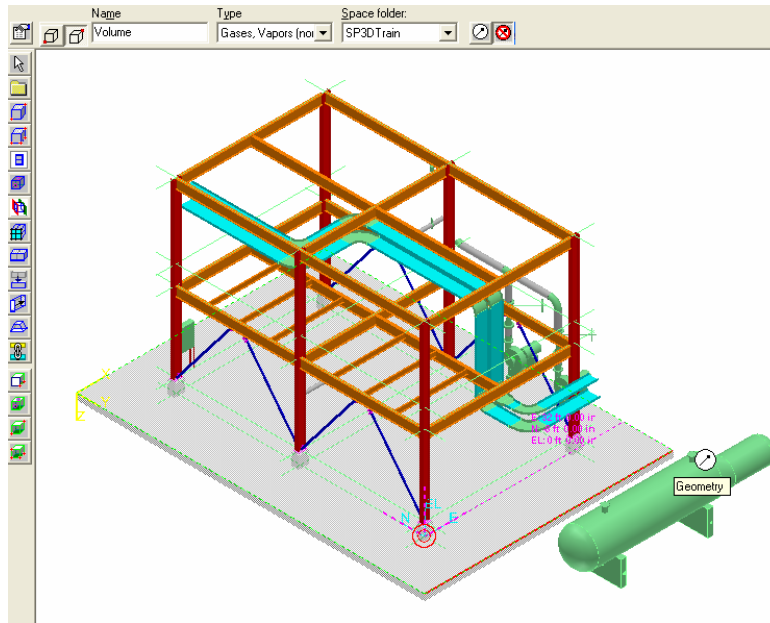
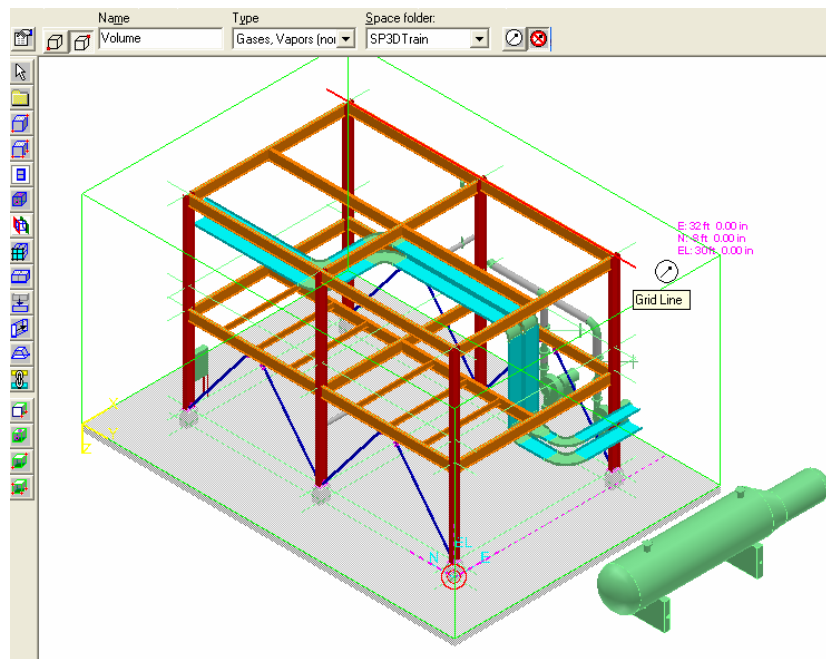*Figure 3: Lock Easting and Northing with F6 and F7*

Figure 4: Locate Grid Line to Define the Elevation

12. 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.

13. 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.

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 resemble Figure 5.
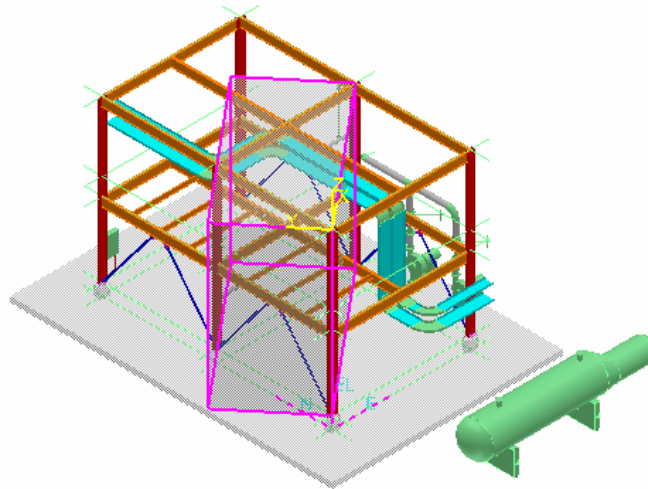

Figure 5: Output: Volume Placed by Four Points

Steps:

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

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 selected in the placing volume by two points example. Then, click **OK**.

4. Retain the default folder **SP3DTrain**.

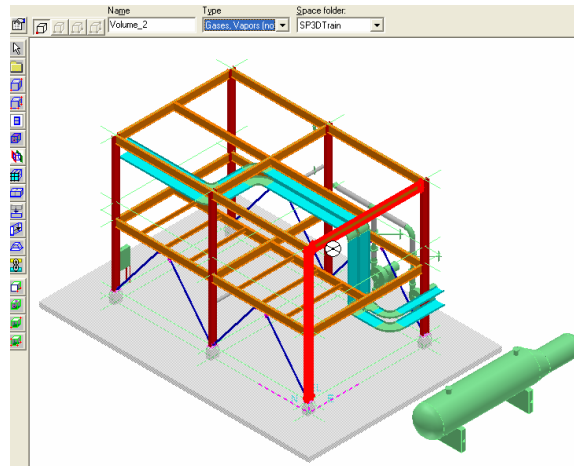5. Enter four points, as shown in Figures 6 to 9.

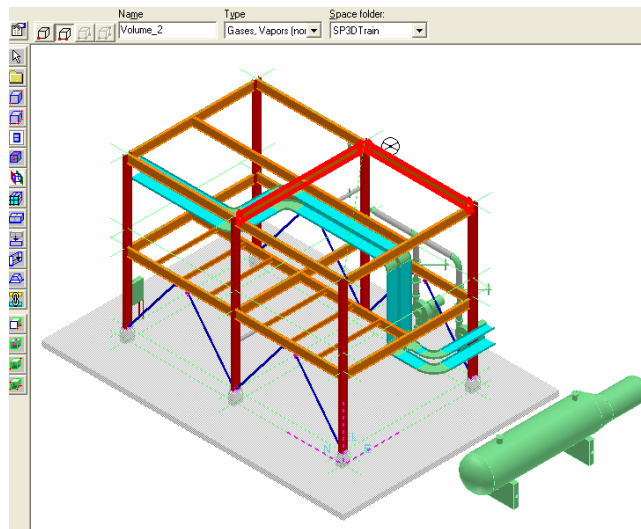*Figure 6: Enter First Point – Origin of the Volume*

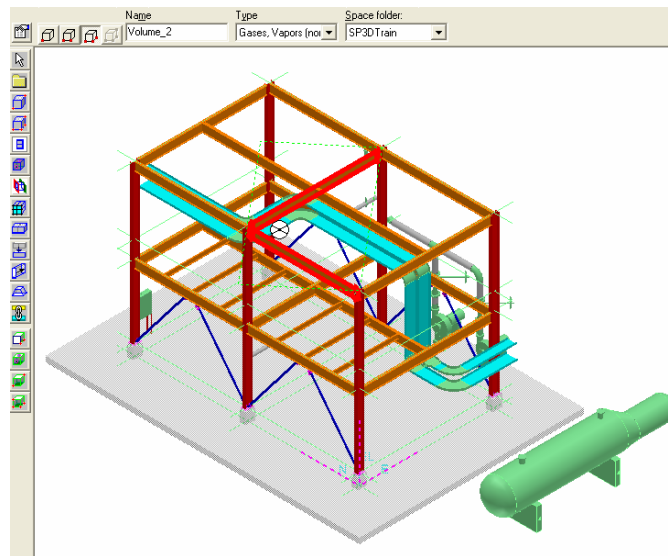*Figure 7: Enter Second Point – Local X- Axis Direction*

Figure 8: Enter Third Point – Local Y- Axis Direction
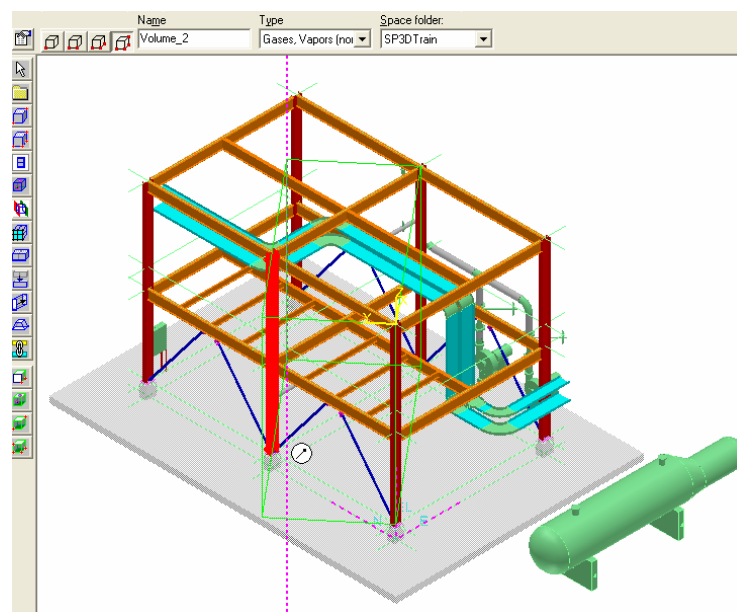
Figure 9: Enter Fourth Point – Local Z-Axis

6. Review the volume that is created and then click **Undo** or **Delete** to remove this practice volume.

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 resemble the highlighted section in Figure 10.
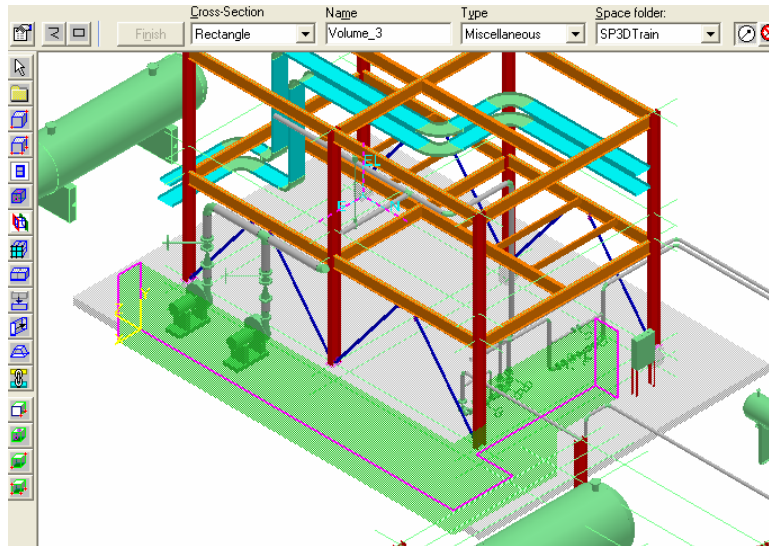

Figure 10: Output: Placed by Using Volume Along a Path

Steps:

1. Define your workspace to show Units **U01**, **U04** and coordinate systems **U01 CS** and **U04 CS.** In your training plant, select the **U01 and U04** filter from **Plant Filters > Training Filters** in the **Select Filter** dialog box.

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

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

4. Select the **View > Fit** command.

5. Switch to the **Space Management** task.

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

7. The **Place Volume Along Path** ribbon displays. Sketch the access way path, as shown in Figure 11.
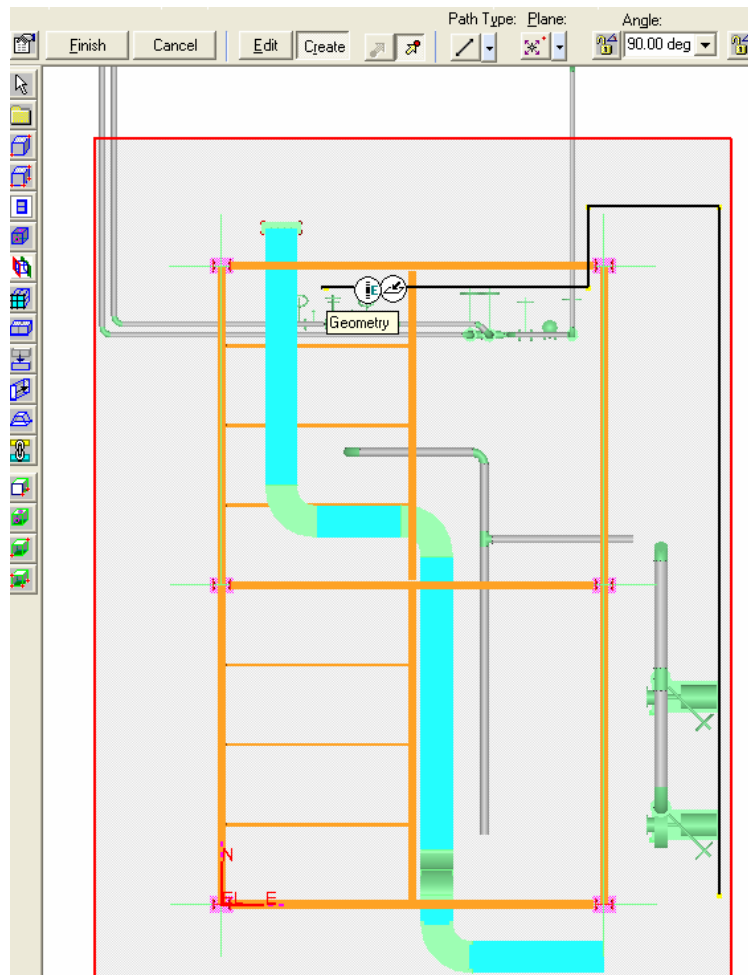
Figure 11: Defining Path of Volume

8. Click **Finish** on the **Place Volume Along 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. Set **Rectangle** in the **Cross-Section** drop down on the **Place Volume Along Path** ribbon.

9. Click the **Properties** option on the ribbon. In the **Zone Properties** dialog box, select the **Cross-Section** tab and specify the following settings:

    *   **A: 3 ft**
    *   **B: 8 ft**
    *   **Cardinality: 8**

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

11. 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. In the **Select Space** dialog box,

expand the **Interference Volumes** folder, select **SPACE_DEF_IV08** from **Interference Volumes**, and click **OK**.

12. Leave the space folder to the defaulted **SP3DTrain**.

13. Click the **Finish** option on the **Place Volume Along Path** ribbon to place the volume.

14. Select the **Common** task by using the **Tasks > Common** command.

SP3D Common Practice Lab: Inserting Reference Files

Inserting a Reference File:

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

Steps:

1. Define the workspace and create a new filter by using the **New Filter (Simple or Asking)** button on the **Select Filter** dialog box. The **New Filter Properties** dialog box is displayed.

2. In the **New Filter Properties** dialog box, type **Unit 4 with Ref file** in the **Name** box. Then, expand **A2** and **CS**. Press and hold the **CTRL** key and then, select **U04** and **U04 CS.**

3. Click **OK**.

4. In the **Select Filter** dialog box, select **Unit 4 with Ref file** and click **OK**.

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

6. Select the **View > Fit** command.

7. Select **Options…** from the **Tools** menu. Under the **General** tab check the **Reference** box. The system prompts you with a message. Click **OK**.

8. Click **OK** to close **Options** dialog box.

9. To access the **Reference** tab, save the session and 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.

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

11. The **Insert File** dialog box appears. Browse to select the file using a UNC path connection, **\\machine\TrainingSymbols\TrainingFiles.** It is important to use UNC path so that the model server will have proper access to the reference file.

Ask the instructor the appropriate UNC path of the **Training Symbol** share.

12. Click **Open** to insert the selected reference file. After you insert the file, the dgn file appears in the **Reference** tab of the Workspace Explorer. The **Reference** tab also starts appearing in the **Filter Properties** dialog box. You can define your workspace to include the reference files.

13. In the **Define Workspace** dialog box, select the **Properties** button to open the **Filter Properties** page.

---

14. In the **Filter Properties** dialog box, click the **Reference tab** and select **building.dgn**

15. Click **OK to close the Filter Properties** dialog box.

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

17. Select the **View > Fit** command to see the graphics from the reference file.

SP3D Common Practice Lab: Placing Control Points

Placing Control Points in a Model:

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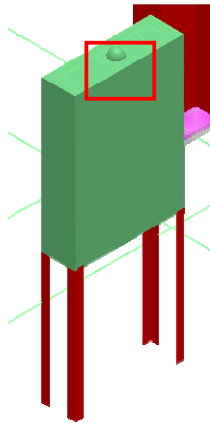

Figure 1: Final Output: Placed Control Points in a Model

Steps:

1. Define your workspace to display Unit **U01** and coordinate system **U01 CS**. In your training plant, select **U01** from **Plant Filters > Training Filters** in the **Select Filter** dialog box.

2. Select the **View > Fit** command.

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

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

5. 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.

6. Click the cable tray port of the **Electrical Device** to place the control point and create an associative relation between them.

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