
ShipConstructor

HVAC

By ARL - Albacore Research Ltd.

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 31. Amendment. Any amendment of this Agreement must be in writing and signed by duly authorized representatives of the parties.
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 38. Language. It is the express will of the parties that this Agreement and related documents have been prepared in English. C'est la volonté expresse des parties que la présente Convention ainsi que les documents qui s'y rattachent soient rédigés en anglais.
-

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Cover model courtesy of *Bender Ship Building*

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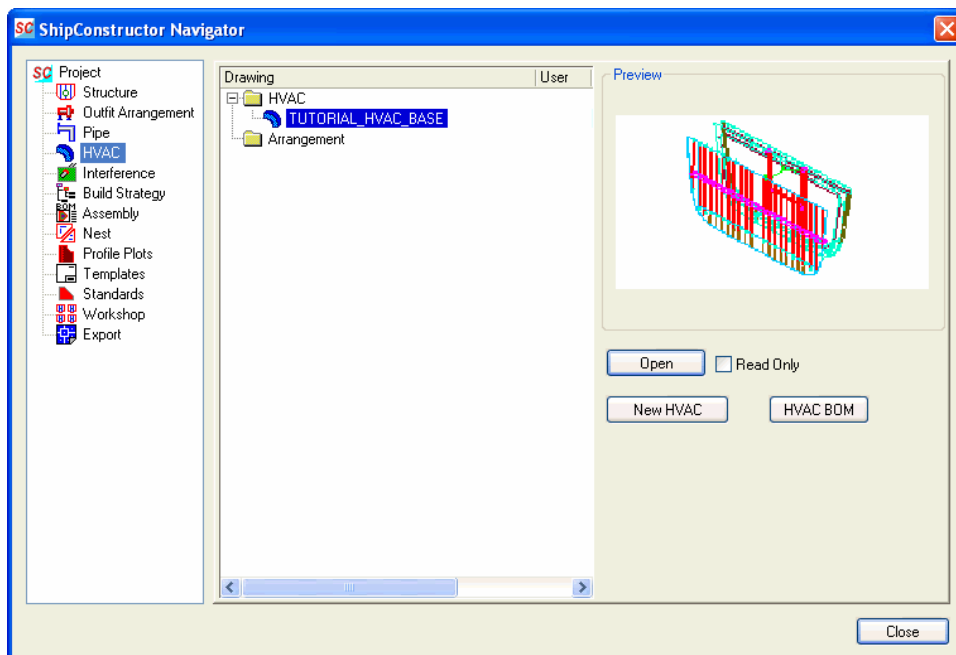
HVAC Reference

Overview

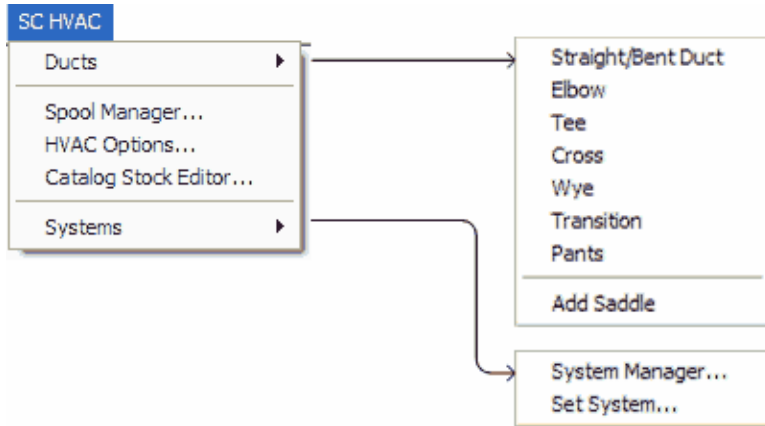
The Heating, Ventilation, and Air Conditioning (HVAC) module is fully integrated with the ShipConstructor suite to provide an easy method of designing HVAC systems in a ship.

The HVAC module's functionality can be accessed through the ShipConstructor Navigator and the SC HVAC menu. Several of the more common functions have been added to the HVAC toolbar. Below are screenshots of the HVAC ShipConstructor Navigator page, SC HVAC menu, and the HVAC toolbar.

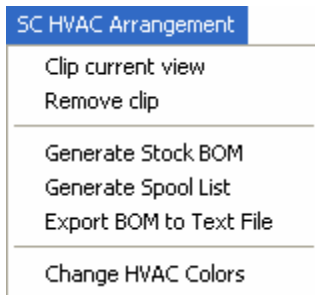
Note: The SC HVAC menu will only be available when in an HVAC model.



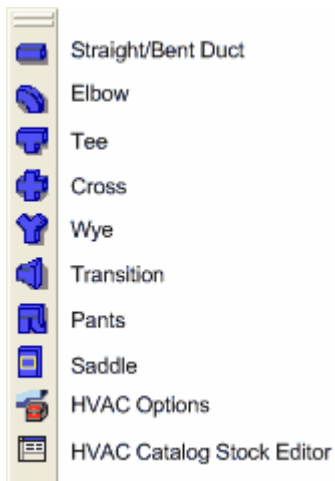
Whenever an HVAC model is open, the SC HVAC Menu is available:



Whenever an HVAC Arrangement Drawing is open, the SC HVAC Arrangement Menu is available:

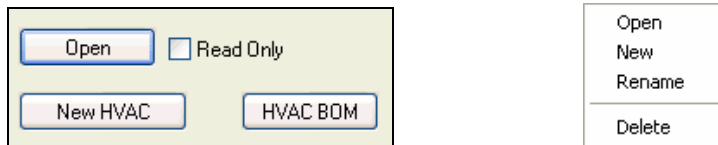


Many functions are conveniently available using the HVAC Toolbar as shown below:



Navigator / HVAC

When HVAC is selected from the list on the left of the Navigator window, the HVAC page is available, including several HVAC buttons shown below. Right-clicking in the list area brings up a context menu.



The HVAC page provides a number of important HVAC and Arrangement Drawing related functions:

- **Open**
- **New HVAC**
- **New Arrangement**
- **Rename**
- **Delete**
- **HVAC BOM**

Open

Menu: **None**

Toolbar: **None**

Command Line: **None**

Navigator: **HVAC / Open**

Permissions: **None**

Opens an existing HVAC model or Arrangement Drawing.

1. Highlight the drawing to be opened in the tree dialog.
2. Click **Open**.

New HVAC

Menu: **None**

Toolbar: **None**

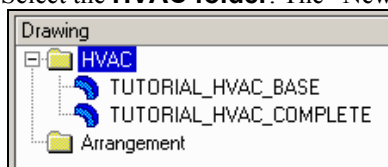
Command Line: **None**

Navigator: **HVAC / New HVAC**

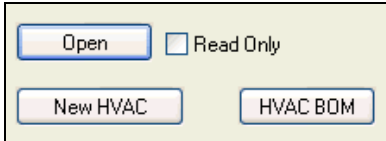
Permissions: **HVAC / Modeling**

Creates a new, empty HVAC model.

1. Select the **HVAC folder**. The “New” button is now labeled **New HVAC**.



2. Click **New HVAC**.



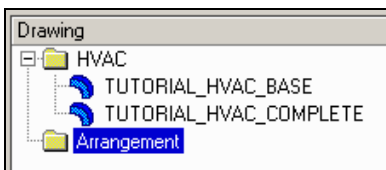
3. In the **New Drawing Dialog** enter the HVAC model name.
4. Press **OK**. An empty model is created and immediately opened for editing.
5. The **SC HVAC menu** is now displayed.

New Arrangement

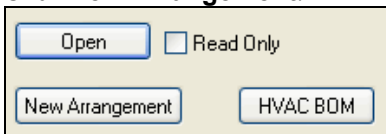
Menu: **None**
 Toolbar: **None**
 Command Line: **None**
 Navigator: **HVAC / New Arrangement**
 Permissions: **HVAC / Modeling**

Creates a new, empty HVAC Arrangement Drawing.

1. Select the Arrangement folder. The “New” button is now labeled **New Arrangement**.



2. Click **New Arrangement**.



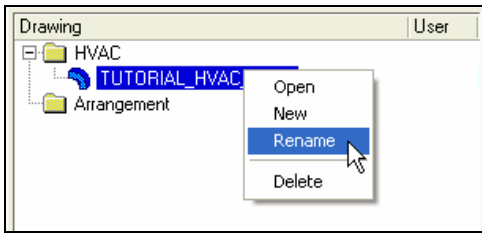
3. In the New Drawing Dialog enter the Arrangement Drawing name.
4. Press **OK**. An empty drawing is created and immediately opened for editing.
5. The **SC HVAC Arrangement** menu is now displayed.

Rename

Menu: **None**
 Toolbar: **None**
 Command Line: **None**
 Navigator: **HVAC / Right-Click/ Rename**
 Permissions: **HVAC / Modeling**

Renames the selected HVAC model or Arrangement Drawing.

1. Highlight the drawing to be renamed in the tree dialog.
2. Right click and choose **Rename**.



3. Type the new name.

Delete

Menu: **None**

Toolbar: **None**

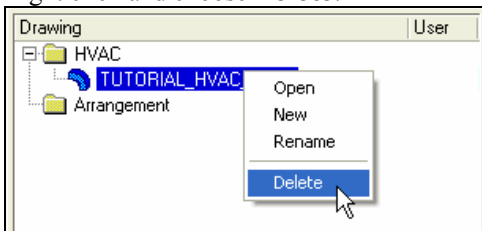
Command Line: **None**

Navigator: **HVAC / Right-Click/Delete**

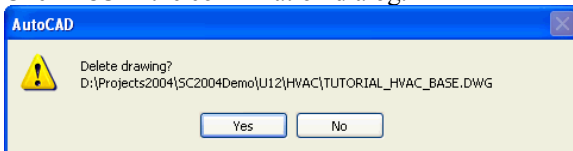
Permissions: **HVAC / Modeling**

Deletes the selected HVAC model or Arrangement Drawing.

1. Highlight the drawing to be deleted in the tree dialog.
2. Right click and choose **Delete**.



3. Click **Yes** in the confirmation dialog.



HVAC BOM (Bill of Materials)

Menu: **None**

Toolbar: **None**

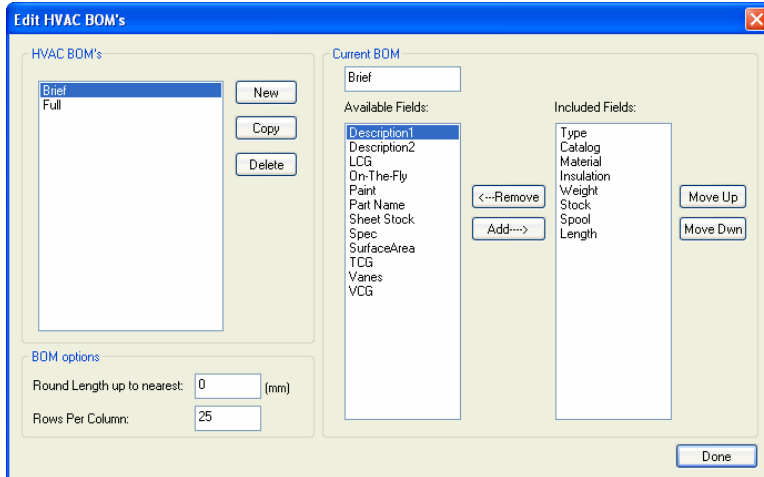
Command Line: **None**

Navigator: **HVAC / HVAC BOM**

Permissions: **None**

Used for editing and/or creating HVAC BOM definitions. A BOM Definition allows the user to store which variables are included in the bill of material and the format in which they appear in the HVAC Arrangement Drawing.

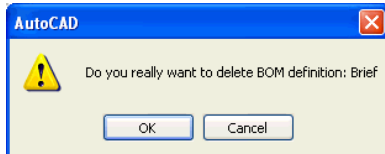
1. Select **HVAC BOM**. The Edit HVAC BOM's dialog appears. It allows creation or modification of BOM Definitions.



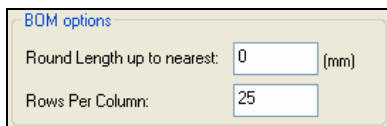
HVAC BOM's (Bills of Materials)

This area of the dialog lists all HVAC BOM's and allows the user to create, copy or delete BOM Definitions.

- **New** - Create a new BOM Definition name. The first default name is New BOM. If this name is accepted, subsequent names will be incremented, i.e. New BOM1. The name can be edited immediately or later in the edit box under **Current BOM**.
- **Copy** – Creates a copy of a selected HVAC BOM definition. Use this if a new BOM Definition is similar to an already existing one.
- **Delete** - Deletes a selected BOM Definition. If you attempt to delete a BOM Definition, which existed before the current session of BOM editing, the following warning will appear to confirm your intentions.

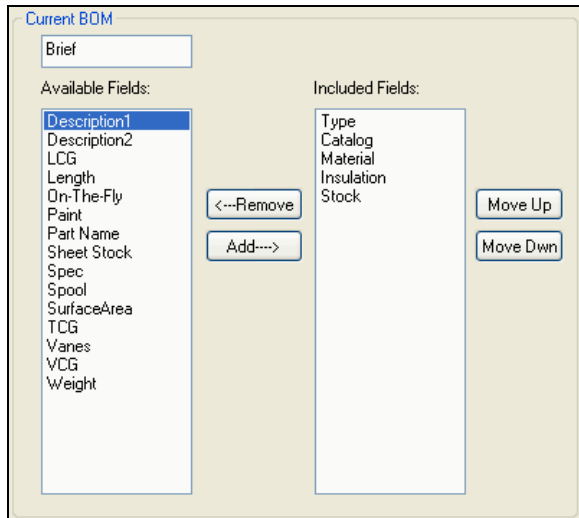


BOM Options



- **Round Length up to nearest** - This option is only available when the Length Field is included
- **Rows Per Column** - The number of rows of text per BOM table column. The BOM table will create more columns as needed. The default is 25 rows.

Current BOM



The currently selected BOM Definition is reflected in the edit box under **Current BOM**. It may be edited here. Fields not already added to the definition remain in the Available Fields list and those already added appear in the Included Fields list. BOM's defined here may be used in Arrangement Drawings and Spool Drawings.

- **Add→** - Add the selected Available field to the Included Fields list
- **←Remove** - Remove the selected field from the Included Fields list.
- **Move Up / Move Down** - Move the selected fields up or down in the Included Fields list. The order from top to bottom represents the order from left to right in which the columns appear in the BOM.

HVAC

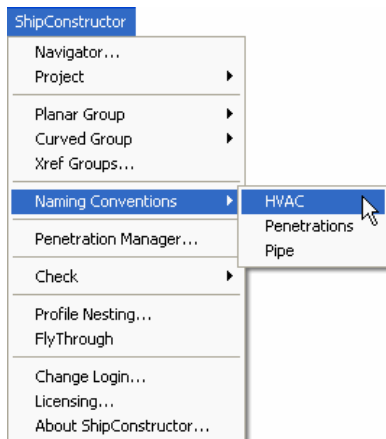
Menu: **ShipConstructor / Naming Conventions / HVAC**

Toolbar: **None**

Command Line: **SCNAMECONHVAC**

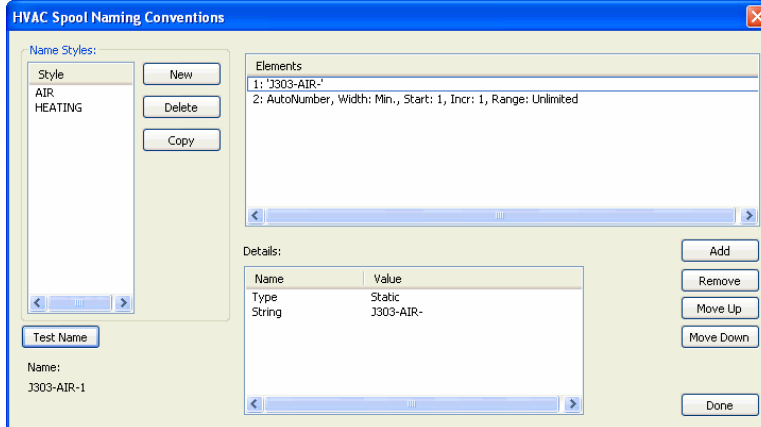
Navigator: **None**

Permissions: **HVAC / Edit Spool Naming**

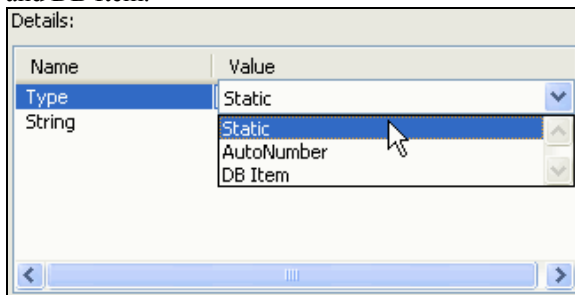


Opens the **Edit Name Components** dialog that allows the user full control of the automatic, custom naming of spool drawings. For Spool creation, use the Spool Manager.

1. Select **Spool Naming**. The HVAC Edit Name Conventions dialog appears.



2. The first task is to add the spool label that defines which information a spool name will include, and the order in which it appears. Press **New**. A default label name is created in the **Name Styles** list. Edit the label name by clicking it twice. You also have the option of copying a previous label definition and then altering it. Simply highlight the Style name in the **Name Styles List** and press **Copy**.
3. Select the Style to which additions or changes to the Naming Convention are to be made.
4. For each new element of the spool name, press **Add**. Elements are shown in the large list box at the top of the window. In the example, the AIR style contains two elements:
 1. **J303-AIR-**: The static string J303-AIR-.
 2. **001**: an automatically generated and incremented number.
5. Format each element within the **Details** list box. There are three types of elements available: Static, AutoNumber and DB Item.



- a. **Static** - A non-changing string value, e.g. the job number. Set the string **Value** to whatever is preferred.

Name	Value
Type	Static
String	J303-AIR-

- b. **AutoNumber** - An automatically indexed value.

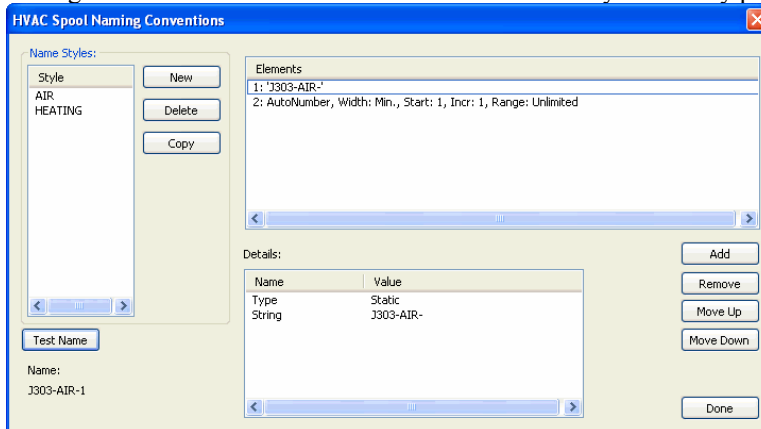
Name	Value
Type	AutoNumber
Width	auto
Start Value	1
Increment Value	1
Max Range	1

- **Width** - The minimum number of characters to display.
- **Start Value** - The starting value.
- **Increment Value** - The number to increment by.
- **Max Range** - Not implemented yet.

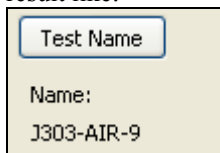
- c. **DB Item** - Used to include the drawing name or a level of the PWBS name, or System name. There are five options within **DB Item type**:

Name	Value
Type	DB Item
Field	PWBS - Name
# Characters	auto
Start From	End
Offset	0
Tree Level	PROJECT

- **Field** – Three options exist:
 - **PWBS - Name** – Select this to have the spool name contain a level of the PWBS (Production Work Breakdown Structure). From the Tree Level list, choose the level of the PWBS to extract the text from. For example if the spool were to be assigned to a panel level then PANEL would be selected.
 - **System - Name** – Includes the system in the spool name. From the Tree Level list, choose the System level to extract the text from (Project, System, or Branch).
 - **Drawing - Name** – Includes the pipe drawing name in the spool name.
 - **# Characters** – Choose the maximum number of characters to display or choose auto.
 - **Start From** – Works in conjunction with Offset and # Characters to extract selected text from the Tree Level name. If set to From Start then the Offset number of characters are removed from the start of the string. If set to From End then the Offset number of characters are removed from the end of the string. For example, let the panel name be FP2114S, and you would like to extract P21. Set Start From to Yes. Offset to 1 and # Characters to 3.
 - **Offset** – The number of characters removed from the Start or End of the string. This works in conjunction with **Start From**.
 - **Tree Level** – The context sensitive list that depends on what Field value has been selected. For PWBS - Name, the name will contain an entry for each level in the PWBS. For System - Name the only valid levels are Project, System, and Branch.
6. The Elements list shows a summary of the included components and their order of appearance in the spool name. Arrange the elements in the Elements list into the order you wish by pressing **Move Up** or **Move Down**



7. Press **Test Name** for a preview of the spool name. For example, the scheme developed above would produce a result like:



8. Press **Done** when finished.

ShipConstructor Osnap

Menu: **SC Utilities / Snap**

Toolbar Button: 

Command Line: **OSNAP / ShipCon Snap tab**

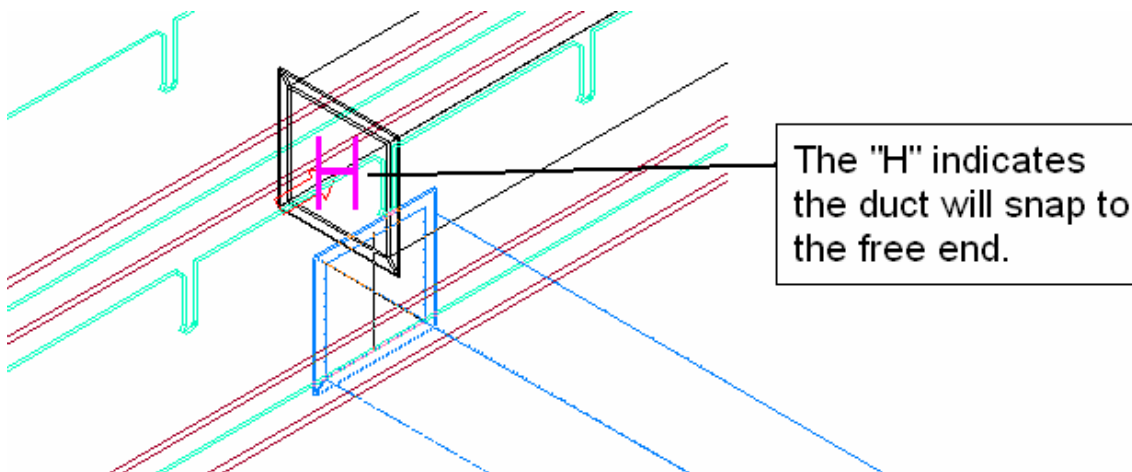
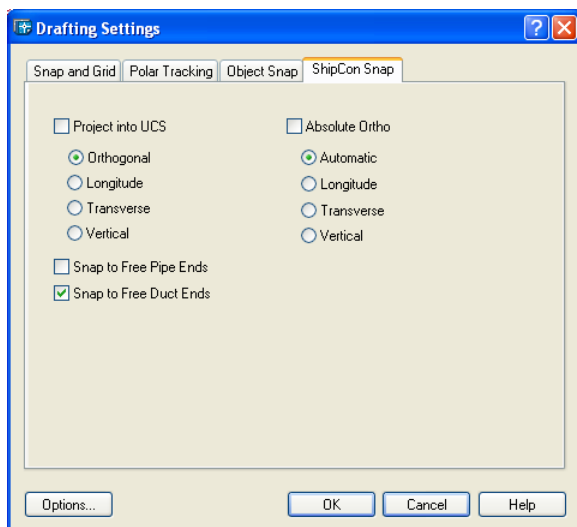
Navigator: **None**

Permissions: **None**

Snap To Free Duct Ends

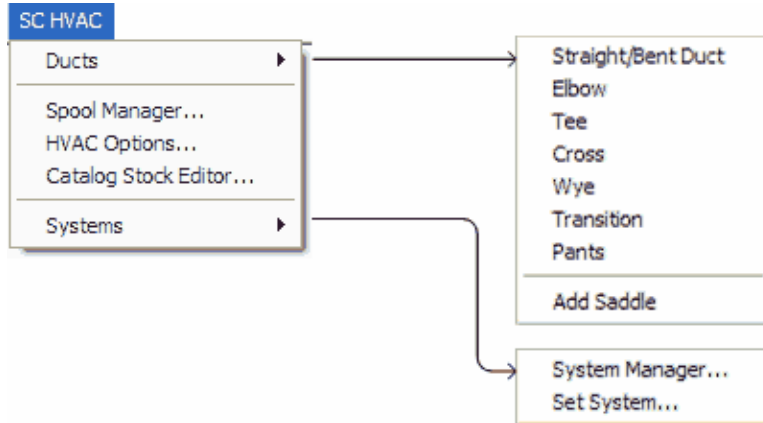
When designing an HVAC system, it will sometimes be useful to have multiple systems of the ship displayed at the same time. When this is done it can be difficult to attach new ducts to an existing duct with AutoCAD **Object Snap** enabled. Enabling **Snap To Free Duct Ends** will cause the cursor to only snap to free duct ends, instead of all objects in the model.

Snap To Free Duct Ends can also be enabled by typing **hvac** while placing a duct in an HVAC Model.



Note: **Snap To Free Duct Ends** will not be enabled unless the AutoCAD **Object Snap** is turned off.

SC HVAC Menu



The SC HVAC Menu is available whenever an HVAC model is open. It provides functionality for creating ducts and manipulating HVAC options. It also affords easy access to HVAC Spool, Stock and System settings.

SC HVAC – Spool Manager...

Menu: **SC HVAC / Spool Manager**

Toolbar Button: **None**

Command Line: **SCHVACSPOLMAN**

Navigator: **None**

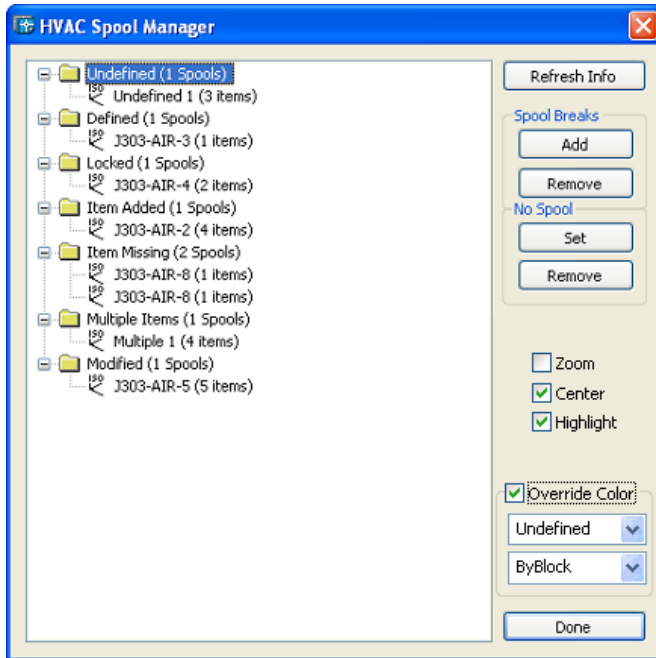
Permissions: **HVAC / Edit HVAC Spools**

This option launches the Spool Manager, which is used to create, edit and otherwise modify the spool breakdown for the current HVAC model. In order to run this command you must have permission to edit/modify HVAC spools (as set in ShipConstructor Manager).

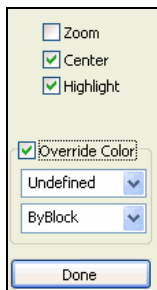
Note: The spool manager for HVAC has been designed in the same fashion as the spool manager for piping. If you are already familiar with using the spool manager for piping you will find the HVAC spool manager to be similar in functionality.

Note: The current HVAC model is automatically saved when the Spool Manager is launched.

When the Spool Manager is first launched it searches through the drawing and builds a list of all the logical spools it finds. It then displays the groups according to their status.

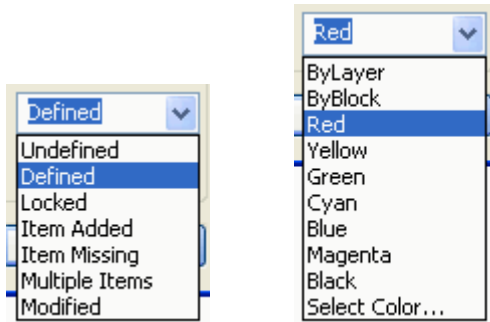


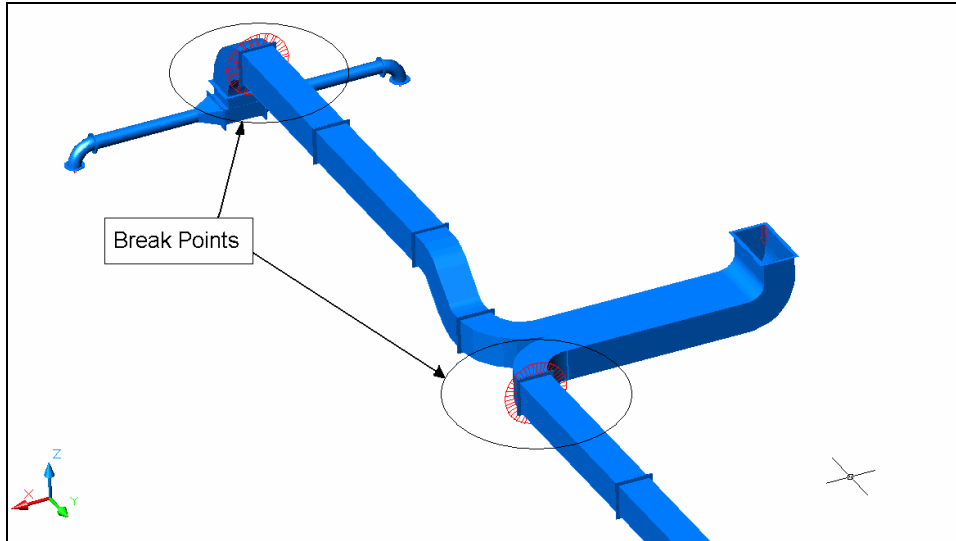
Double clicking on an item in the tree causes that item to be selected.



The lower right area of the Spool Manager provides display options to customize how the ducts are displayed in the drawing while managing spools. You may have a model of an HVAC system with dozens of spools in various stages. Adjusting these options can greatly simplify your work.

- **Zoom** - The drawing window will zoom to center on the selected spool with the spool filling the screen.
- **Center** - The drawing window will shift to center on the selected spool, but the zoom level will not change.
- **Highlight** - The ducts in the selected spool will become highlighted in the drawing window.
- **Override Color** – When checked, this feature changes the color used for the system to a selected color when the condition in the first drop down list is matched. For example when defining spools, you should change the color of the defined spools to be in good contrast to the undefined spools to make picking the correct spools easier.





The above diagram shows a spool grouping consists of a series of interconnected ducts and fittings. A grouping ends when an unconnected end or a break is reached. Initially a system is treated as if it is one large spool. Typically, when spooling a system you should:

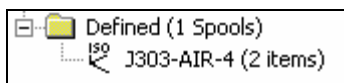
- First, define all ducts that are not going to be spooled. This will divide the system into several spools already, as any item set as no-spool causes a spool break.
- Second, add **break points** to split the spools into manageable sizes.

Spool Categories

Each spool can belong to one spool category. Each category defines the state the spool is in. These are:

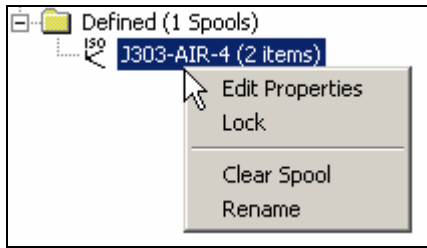
- Defined
- Undefined
- Locked
- Item Added
- Item Missing
- Multiple Stocks
- Modified

Defined

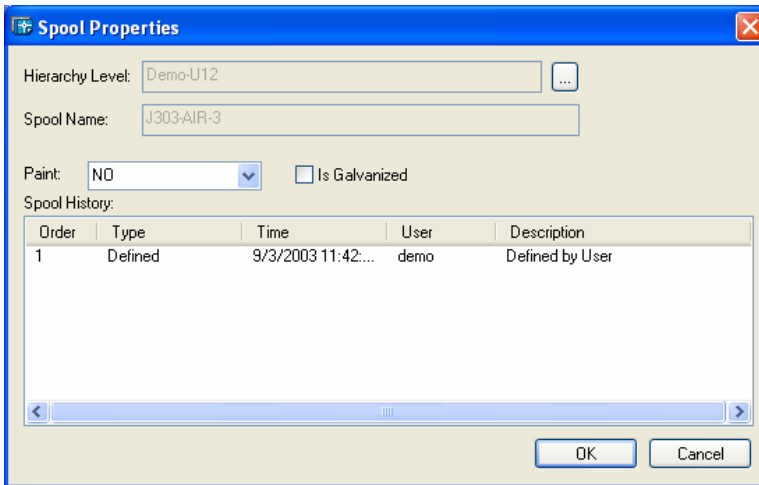


A **defined** spool is a run of ducts which have been defined as a spool (see **Undefined - define** on how to define a spool) and given a name according to the projects **spool naming convention**. In the example above the name of the current job (“J303”) has been combined with the name of the system (“AIR”) and an automatically generated number (“4”) to give the spool a unique designation. When a spool is defined a layer will be created for that spool and the items belonging to that spool are moved to that layer.

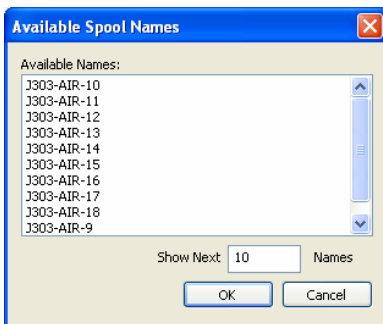
Item Menu - Right clicking on a defined spool will bring up the following menu:



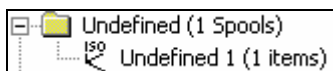
- **Edit Properties** - Brings up the Spool Properties dialog allowing you to edit the hierarchy level and coating of the spool. It will also display the spool's revision history, which includes all the changes that have been made to the spool since it was defined. To change the hierarchy level, use the button to bring up a tree displaying the hierarchy for the entire project.



- **Lock** - Locks a defined spool and will prevent any editing of the ducts in the spool that would change the spool (erasing, moving, adding new ducts to unconnected ends, **freeing** duct, **adding/removing** breaks etc.).
- **Clear Spool** - This will undefine the selected spool.
- **Rename** - This option allows you to rename the spool. This option opens the Available Spool Names dialog, which displays unused spool names for you to choose from; by default it displays the next 10 available names. However, you can increase or decrease this number by editing the number in the lower right hand corner of the Available Spool Names dialog box.



Undefined



Undefined spools are ones whose ducts do not belong to any defined spool grouping.

Root Menu - Right clicking on the root of the undefined branch brings up the following menu:



Define All - Specify multiple spools by picking them in the model.

1. First pick the properties the spools will be assigned from the **Spool Properties Dialog**.
2. Next the dialog will disappear and the command line will display:

Choose spool:

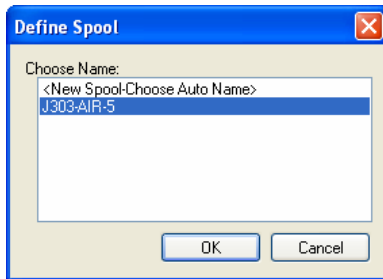
3. Choose the spools in the order they are to be named by selecting any duct in a spool. Already **defined** spools will be ignored.
4. Once the last undefined spool has been picked the **Spool Manager** will reappear displaying the newly defined spools. Hitting **ESC** at any time while picking spools will halt the command and will return to the **Spool Manager** leaving the remaining spools undefined.

Item Menu - Right clicking on one of the individual undefined spools will bring up the following menu:

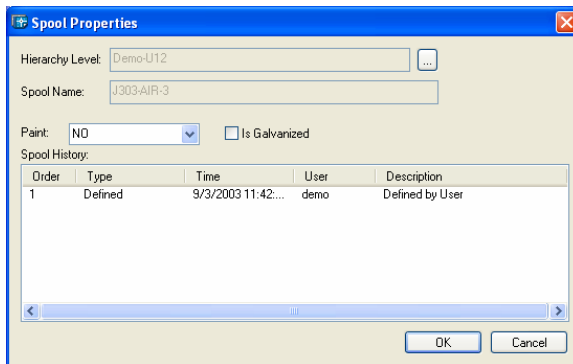


Define - Allows you to define an undefined spool by picking it from the list.

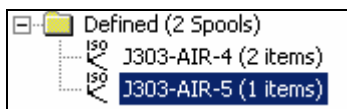
1. Choose how the spool will be named.
 - **New Spool - Choose Auto Name** - Pick the name from a list of available names (see **Define - Rename**).
 - **J303-AIR-005** - The next available name will be displayed in this list.



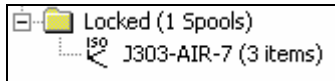
2. Next select the properties for the spool from the **Spool Properties** dialog.



3. Click **OK** and the newly defined spool will be added to the **Defined** category.



Locked



Locked spools are **defined** spools that have been locked to prevent editing. A spool that has been locked will not allow any of its ducts to be modified in any way, nor will it allow ducts to be added in such a way as to change the configuration of the spool. This is done so that once a spool has been “finished” it can be locked preventing a **modeler** from changing the spool without first checking with the person in charge of the project.

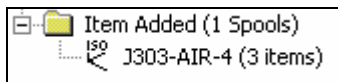
Item Menu - Right clicking on an item will bring up the following menu:



Properties - Displays the spools properties in a read only state. See **Defined - Edit Properties**.

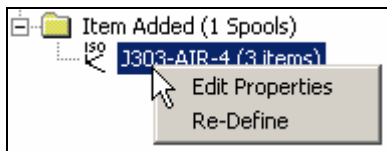
Unlock - Unlocks the spool moving it to the **Defined** Category.

Item Added



Item Added spools are ones that have had new ducts or fittings added since the spool was defined.

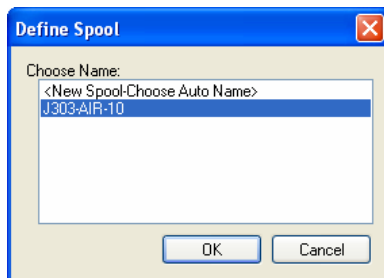
Item Menu - Right clicking on an item will bring up the following menu:



Edit Properties - Allows the user to edit the spool properties. See category: **Defined - Edit Properties**.

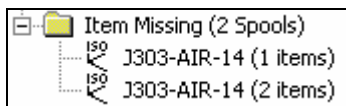
Re-Define - Redefines the spool using an entirely new name.

1. First choose the name you wish the redefined spool to have.
 - **Choose Auto Name** - Pick the name from a list of available names (see **Define - Rename**).
 - **<Next Auto Name>** - Pick the next available name.



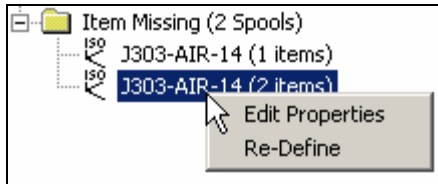
2. Press **OK** and the spool will be added to the **Defined** Category.

Item Missing



Item Missing spools are ones that have been separated by the use of break points. In the screen shot above you can see the **Item Missing** list contains two entries, both with the same name. In this case the user split off one duct into its own spool, while the other 2 remained together. They need to be re-defined before they can be turned into a spool drawing.

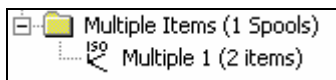
Item Menu - Right clicking on an item will bring up the following menu:



Edit Properties - Allows the user to edit the spool properties. See **Defined - Edit Properties**.

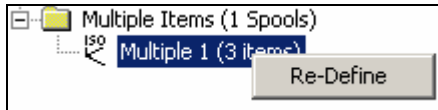
Re-Define - Redefines the spool using a new name (see **Item Added - Re-Define**).

Multiple Items



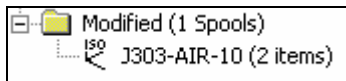
Multiple Item spools contain ducts that used to belong to two or more defined spools. This typically happens when removing No Spools or **break points** which cause two formerly separate spools to become one.

Item Menu - Right clicking on an item will bring up the following menu:



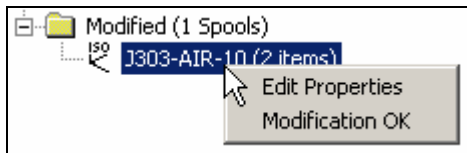
Re-Define - Redefines the spool using a new name (see **Item Added - Re-Define**).

Modified



Modified spools are **Defined** spools that have had ducts in the spool modified by moving or stretching. They have been moved to the **Modified** Category to let the spooler know they have changed and may need to have their drawings regenerated. Use the **Modification OK** command to move the spool back to the **Defined** Category.

Item Menu - Right clicking on an item will bring up the following menu:

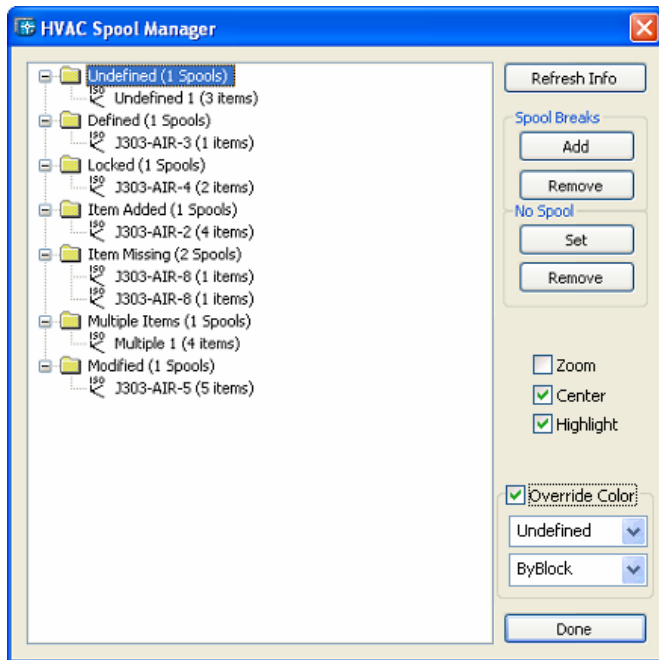


Edit Properties - Allows the user to edit the spool properties. See **Defined - Edit Properties**.

Modification OK - Moves the spool back to the **Defined** Category, indicating that the spooler has OK'd the changes made.

Spool Buttons

To the right of the tree is the side bar menu, which contains a number of functions for managing the spools.



Refresh Info - Causes the **Spool Manager** to rebuild the list of spools from the model. Due to the fact that the spool manager stays up while you work in the model, the spool group listings will need to be updated when a duct is added, removed, or modified. The window will grey out when it is necessary to refresh the spool listings.

Spool Breaks / Add - This button switches the focus to the model and prompts the user:

```
Pick duct near connection to break at:
```

It then inserts a break point at the closest valid connection, and asks the user to select another connection. Pressing **Enter** or **ESC** will end the command. This command will disable the tree until **Refresh Info** is pressed.

Spool Breaks / Remove - This button switches the focus to the model and prompts the user:

```
Pick duct near connection to break at:s
```

It then removes the nearest break point, and repeats the command again. Pressing **Enter** or **ESC** will end the command. This command will disable the tree until **Refresh Info** is pressed.

No Spool / Set - This button switches the focus to the model and prompts the user:

```
Pick duct to set as no spool:
```

Any ducts selected will be set to **No-Spool**, and repeats the command again. Pressing **Enter** or **ESC** will end the command. This command will disable the tree until **Refresh Info** is pressed.

No Spool / Remove - This button switches the focus to the model and prompts the user to

```
Pick duct to remove no spool setting:
```

It then removes the free setting from this duct, and repeats the command again. Pressing **Enter** or **ESC** will end the command. This command will disable the tree until **Refresh Info** is pressed.

SC HVAC – HVAC Options...

Menu: **SC HVAC / HVAC Options**

Toolbar Button: 

Command Line: **SCHVACOPTIONS**

Navigator: **None**

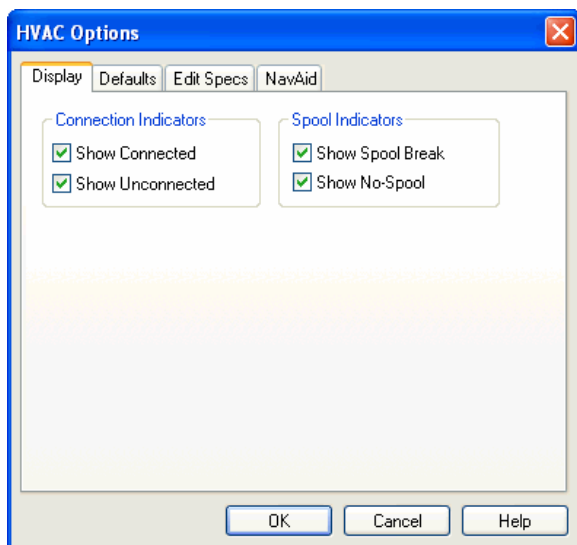
Permissions: **None**

This option launches the HVAC Options dialog. From here you can configure:

- **Display**
- **Defaults**
- **Edit Specs**
- **NavAid**

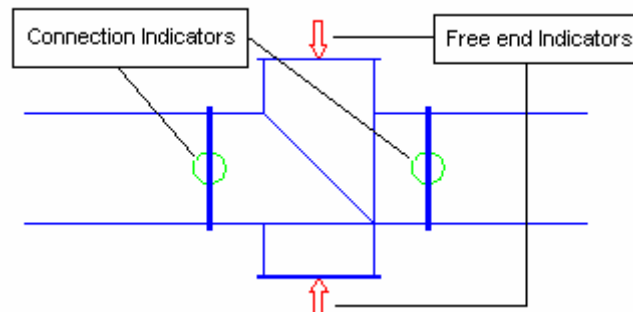
Display

The Display tab contains various settings to assist in the placing of ducts and editing spools.



Connection Indicators

The duct Connection Indicators can be enabled to assist in designing HVAC systems by making it clear if a duct's end is connected to another end. The Connection Indicators can also be disabled for a less cluttered model.



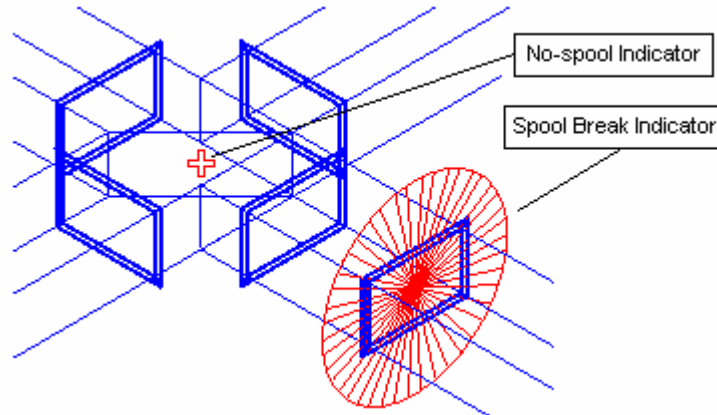
Show Connected – Draws a circle at all the connected duct ends in the model.

Show Unconnected – Draws an arrow pointing at all the Free Duct Ends in the model.

Note: The Connection Indicators will not be visible when Shaded or Hidden Line mode is on.

Spool Indicators

Spool Indicators can be displayed to show where spool breaks have been placed and which ducts have been declared as No-Spool. It is strongly recommended to show all spool indicators while editing spool groupings.



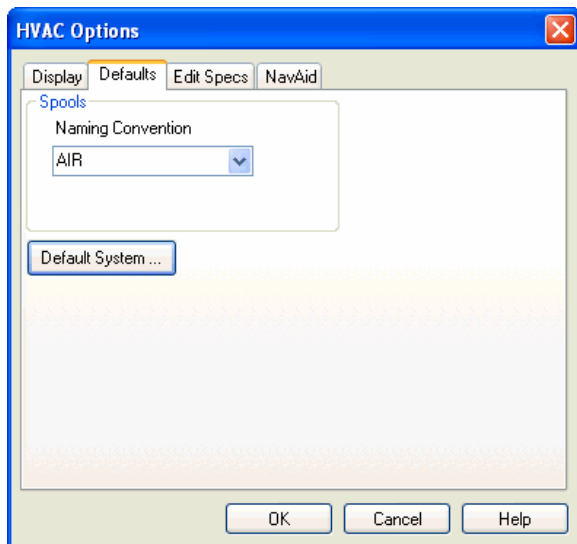
Spool Break Indicator – Draws a large spoked circle for each spool break.

No-Spool Indicator – Draws a small cross at the center of each duct flagged as No-Spool.

Note: The No-Spool Indicator will not be visible when Shaded or Hidden Line mode is on.

Defaults

The settings on this tab are applied to the currently open HVAC model only.

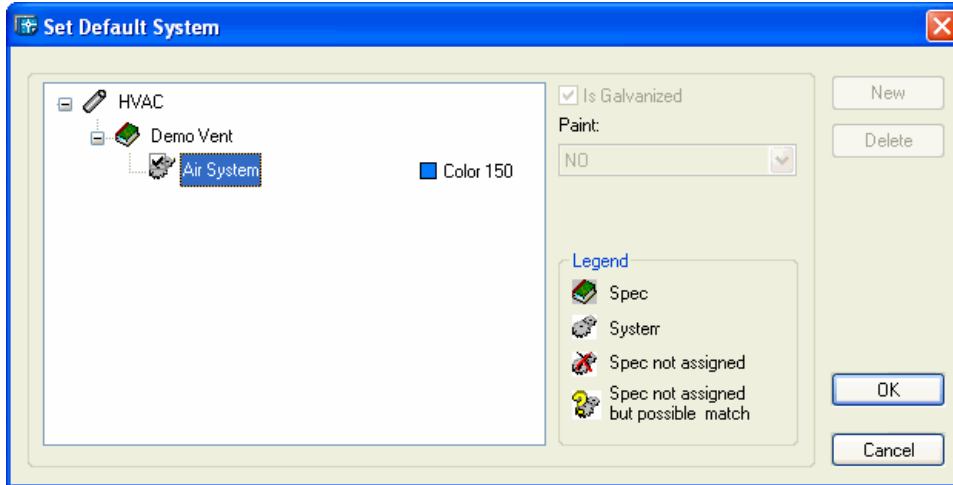


Naming Convention – Sets the spool naming convention to use when spools are defined or renamed. **HVAC / Edit Spool Naming** permissions are required to change the naming convention.

Default System... – Displays the **Set Default System dialog**, which allows the user to select the default system to be used for new HVAC entities. Every system is based on a spec. Thus all ducts routed for that system would be from the same spec.

Note: For full editing of Systems and Branches, from the **SC HVAC** menu select **Systems / New – Modify Systems**.

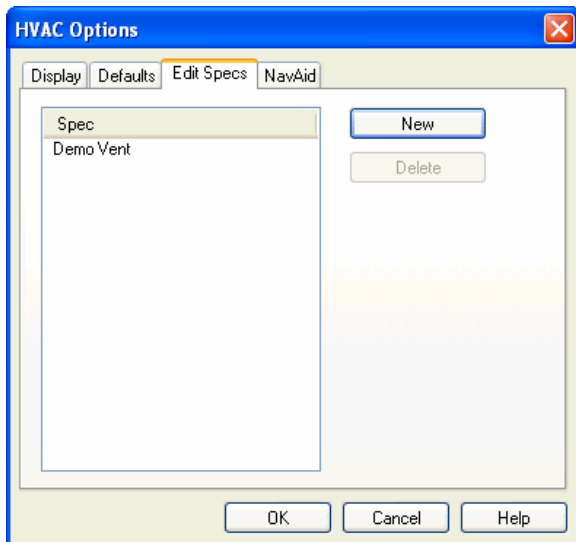
Set the Default System



1. Select **SC HVAC / Options**. The **HVAC Options** dialog is displayed.
2. Select the **Defaults** tab and press the **Default System ...** button.
3. Select a system to become the default system.
4. Press **OK**.

Edit Specs

The Edit Specs tab allows new specs to be added and existing ones to be removed. Specs, or specifications, specify details for design, fabrication, and construction. In ShipConstructor - HVAC, specs allow you to group systems that conform to a particular design specification. Ducts created on-the-fly will automatically be assigned to the current spec.



New Spec

1. Select SC HVAC / HVAC Options. The **HVAC Options** dialog is displayed.
2. Select the Edit Specs tab and press the New button. A new spec will be added to the list.
3. Set the new spec's name.
4. Press OK.

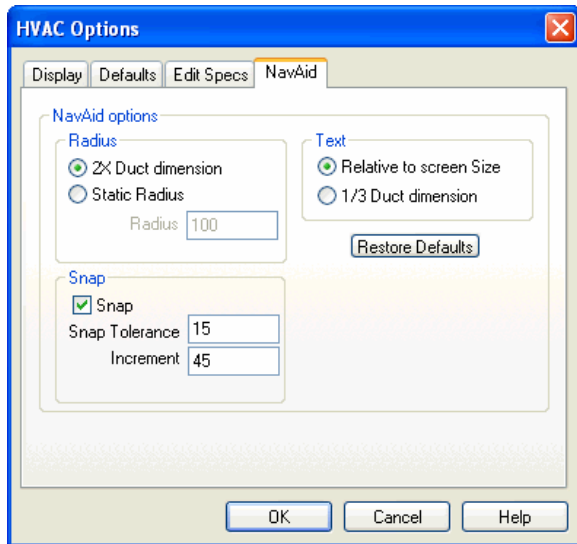
Delete Spec

1. Select SC HVAC / Options. The **HVAC Options** dialog is displayed.
2. Select the Edit Specs tab.
3. Select the spec to be removed.
4. Press **Delete**.
5. Press OK.

Note: If a spec has systems below it then it cannot be deleted until the systems have been removed.

NavAid

The NavAid tab displays options for the NavAid tool, which appears during placement of HVAC to aid in the orientation of the duct. Generally the NavAid tool appears any time there is a request for directional input.

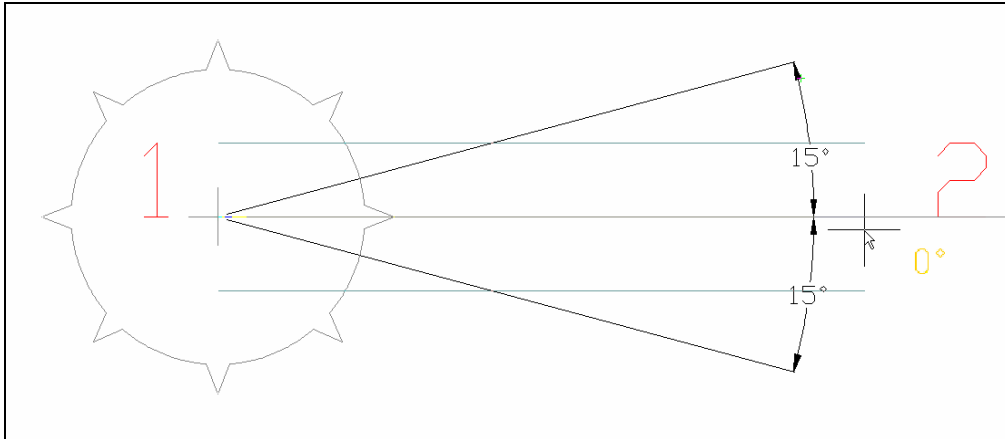


Radius

- **2X Duct Dimension** – The NavAid icon is set to double the width or diameter of the duct that is being placed.
- **Static Radius** – Sets the absolute size of NavAid icon.

Snap – Can be activated or deactivated by selecting the Snap check box.

- **Snap Tolerance** – Retains the closest **Increment** angle within the Snap Tolerance. For example, in the following figure the Increment value is set to 45° and the Snap Tolerance is set to 15°. Notice that within the tolerance zone, the angle (shown in yellow as 0°) maintains the closest increment regardless of where the cursor is.



- **Increment** – The angle between each successive snap. Each “spike” on the NavAid tool represents a snap angle. For the example above, there are eight spikes, one for every 45° increment.

Text

- **Relative to Screen Size** – The angle text (“0°” in the diagram) maintains its size regardless of zoom factor.
- **1/3 Duct Dimension** – The angle text (“0°” in the diagram) is a fixed size i.e. when zooming out the text gets smaller.

NavAid Options during Placement of HVAC

During the placement of ducts in an HVAC Model, several options appear on the command line. The options available depend on the type of duct. (See the sections on placing ducts.) The NavAid options are listed below.

plane– Successively aligns the UCS of the NavAid icon to that of the world coordinate’s X-Y, Y-Z, and Z-X planes.

Rotate – Rotates the 0° mark of the NavAid about the Z-axis of the tool’s UCS, using the right hand rule.

angle of Plane – Rotates the NavAid’s X-Y plane about the tool’s X-axis using the right hand rule.

SNap<on/off> – Enables or disables the angle snap mode.

SC HVAC – Catalog Stock Editor...

Menu: **SC HVAC / Catalog Stock Editor**

Toolbar Button: 

Command Line: **SCHVACSTOCK**

Navigator: **None**

Permissions: **HVAC / Catalog Stock Editor**

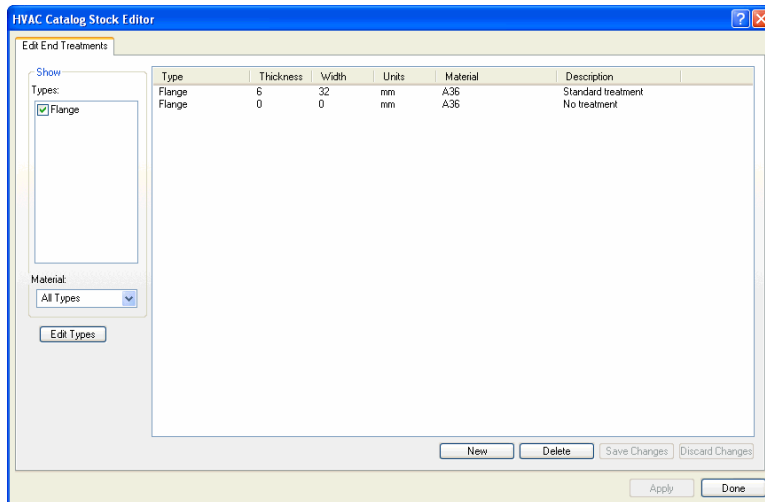
Edit End Treatments Tab

End Treatments are required to create ducts. At least one End Treatment Type must be defined, and an empty End Treatment must be created. This allows ducts to be created with no end treatments.

Required Steps to create an empty End Treatment:

1. Open the **HVAC Stock Editor**.
2. Select the **Edit End Treatments** Tab
3. Click the **Edit Types** button

4. In the **Edit End Treatment Types** dialog, click **New** , enter a type name, and click **OK**
5. Back in the **HVAC Catalog Stock Editor**, click the **New** button
6. In the new row, type in a name for the empty End Treatment in the **Description** column. Edit the Units and Material columns if desired. *Leave the Thickness and Width as 0.*
7. Click **Save Changes**
8. Click **Done**



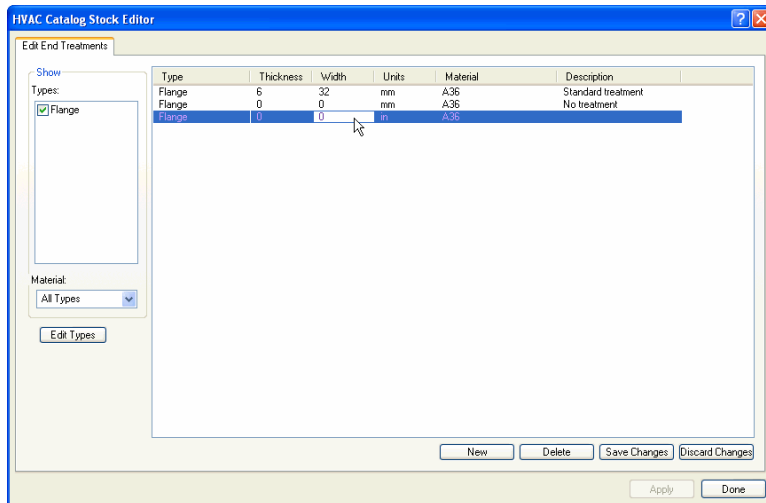
Edit End Treatment Types

The dialog displays a list of End Treatment Types. Using the New and Delete buttons, you can add or delete End Treatment Types. Double clicking on an item in the list will allow you to edit existing types. Click Done to save your changes.

Edit End Treatments

To Create End Treatments:

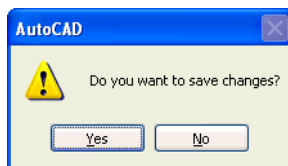
1. Click on **New**, at the bottom center of the Edit End Treatments Tab of the **HVAC Stock** Editor. A new row is added to the list of End Treatments.
2. Select the proper End Treatment Type by double clicking on the Type cell, and choosing from the drop-down list.
3. Select the Thickness and Width by double clicking those cells and typing in values.
4. Select the units of the Thickness and Width values by double clicking the Units cell.
5. Double click the Description cell and enter a description of the new End Treatment. (This step is optional.)
6. Click **Save Changes** if you want to keep the changes.
7. Click **Done** to close the Stock Editor.



To Edit End Treatments:

1. In the Edit End Treatments Tab of the **HVAC Stock Editor**, double click on any value to change it.
2. Click **Save Changes** if you want to keep the changes.
3. Click **Done** to close the Stock Editor.

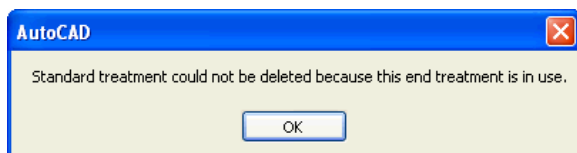
Note: If you click **Done** without saving changes, the following dialog will prompt you to save.



To Delete End Treatments:

1. In the Edit End Treatments Tab of the **HVAC Stock Editor**, select the End Treatment to delete by clicking anywhere on that row.
2. Click on **Delete**, near the bottom of the dialog
3. Click **Save Changes** if you want to keep the changes.
4. Click **Done** to close the Stock Editor.

Note: You cannot *delete* an End Treatment Type if it is in use. The following dialog will appear after you click on Done in the Edit End Treatment Types dialog. Clicking OK in the following dialog cancels any changes made in the Stock Editor.



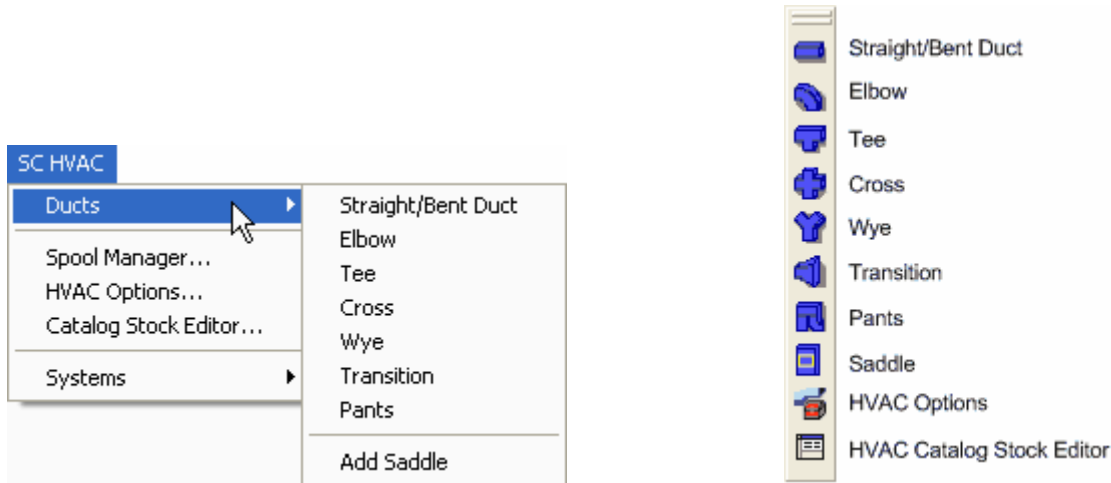
Show Types

It is useful to filter the number of listed End Treatments to help find a particular item. In the list of Types on the left, check one or more types to display in the list on the right. Check each Type to display the entire list of End Treatments.

Show Materials

Use the Material drop-down list to the left to only view End Treatments with a certain material. Select All Types to view End Treatments with any type.

Ducts Sub-Menu



The **Ducts Sub-Menu** allows for the creation of each type of HVAC fitting. Each fitting is also available on the **HVAC Toolbar**.

This Section covers the placement HVAC parts in an HVAC Model. To specify the properties of any HVAC part, either before or after placing it in model space, use the **HVAC On-The-Fly** dialog.

Ducts – Straight/Bent Duct

Menu: **SC HVAC / Ducts / Straight/Bent Duct**

Toolbar Button: 

Command Line: **SCHVACBent**

Navigator: **None**

Permissions: **HVAC**

Creates a bent or straight duct specified by two or more points. The bends may be rounded or mitered.

Placing a Straight or Bent Duct

Bent or Straight Ducts can only be placed in end mode. The first selected point determines the start location for the duct's run. The following points determine the orientation and direction the duct will travel.

Creation of Straight/Bent Duct

1. Start the Straight/Bent Duct function
2. Complete any On-The-Fly duct creation required (see **On-The-Fly Duct Creation**)
3. A straight duct is displayed with End 1 at your cursor. You are prompted to pick the first point or choose another option.

.Please pick point : or [Modify/Rotate profile/Toggle bent mode <rounded>]:

- **Pick Point** – Select the start point of the duct anywhere in the model. To connect to another duct, select any free end in the model (indicated by an arrow pointing at the end of a duct). The new duct will connect to this end, changing its size, orientation and direction to make the connection.
- **Modify** – Make changes to the On-The-Fly part before placing it
- **Rotate profile** – Rotate the duct’s profile from the direction shown at your cursor.
- **Toggle bent mode** – Swap between rounded corners and mitered corners at bend points. The Current mode is shown in <angle brackets>.

Note: If the picked point is a free end of an existing duct then the new duct will connect to the free end, and the profile will be changed to match the profile it is connecting to. In step 4, only Pick Point, Radius and Length will be available.

4. The NavAid is shown at the first picked point, and End 1 of the bent duct is anchored there. You are prompted again:

Please pick point : or [SNap<off>/Radius/LLength/Fwd/Aft/Stbd/Port]:

- **Pick Point** – Select the second point of the duct anywhere in the model.
- **Snap** – Snap the direction to the NavAid. This may be turned on or off. The current status is displayed in <angle brackets>.
- **Radius** - For rounded corner mode only, edit the corner radius.
- **Length** – Provide a length for this section of the duct. The direction of this section is provided by the cursor position, unless one of the following directions has been specified.
- **Fwd** – Set the direction of this section to forward.
- **Aft** – Set the direction of this section to aft.
- **Stbd** – Set the direction of this section to starboard.
- **Port** – Set the direction of this section to port.

5. After picking the second point, the NavAid moves to End 2, and the duct bends around End 2.
6. For a straight duct (only two ends) complete the command by pressing Enter, or for a bent duct:

Please pick point : or [SNap<off>/plaNE/Radius/LLength/Fwd/Aft/Stbd/Port/Up/Down]:

- **Pick Point** – Select the next point of the duct anywhere in the model.
- **Snap** – Snap the direction to the NavAid. This may be turned on or off. The current status is displayed in <angle brackets>.
- **Plane** – Toggles the plane of the bend. (It can bend either around the profile “Width” or “Height”.)
- **Radius** – For rounded corner mode only, edit the corner radius.
- **Fwd** – Set the direction of this section to forward.
- **Aft** – Set the direction of this section to aft.
- **Stbd** – Set the direction of this section to starboard.
- **Port** - Set the direction of this section to port.
- **Up** - Set the direction of this section to up.
- **Down** - Set the direction of this section to down.

7. Repeat step 5 as required for as many bends as needed in the duct.

Ducts – Elbow

Menu: **SC HVAC / Ducts / Elbow**

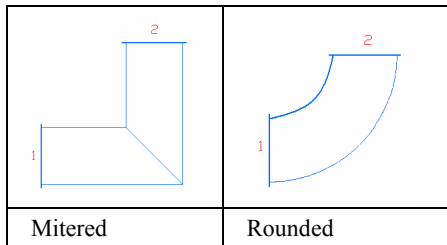
Toolbar Button: 

Command Line: **SCHVACElbow**

Navigator: **None**

Permissions: **HVAC**

Places an elbow duct.



Placing an Elbow

Placement of an elbow can be broken down into three different categories:

- **End Mode** – The first picked point determines the location of the elbow's selected end.
- **Corner Mode, Disconnected** – The first picked point determines the elbow's center point.
- **Corner Mode, Aligned** – The elbow will be aligned to the selected ducts.

Creation of Elbow in End Mode

1. Start the Elbow function.
2. Complete any On-The-Fly duct creation required (see **On-The-Fly Duct Creation**).
3. An elbow duct is displayed with your cursor at one end (in End Mode). You are prompted to pick the first point or choose another option:

Please pick point : or [Corner mode/Modify/sWap ends <1>]:

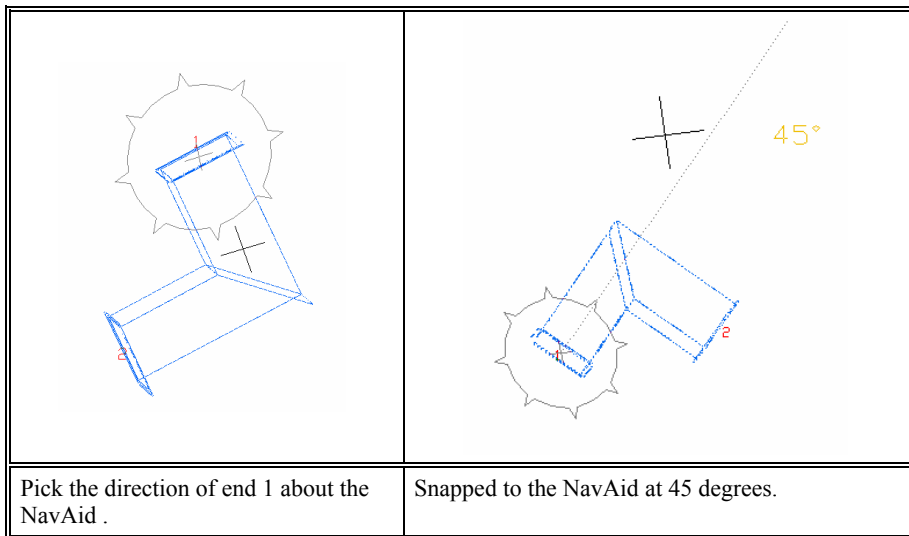
- **Pick Point** – Select the location anywhere in the model.
- **Corner Mode** – Toggle between Corner mode and End mode. End mode places the elbow by one of the ends. Corner mode places the elbow by the center.
- **Modify** – Make changes to the On-The-Fly part before placing it in the model.
- **Swap Ends** – Swaps the current end, displayed in <angle brackets>. In End Mode, the current end is located at the cursor and is placed at the picked point.

Note: If the picked point is a free end of an existing duct then the elbow will connect to the free end, the elbow's connecting end's profile will be changed to match the profile it is connecting to, and step 4 will be skipped.

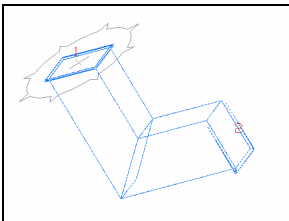
4. The NavAid is shown at the first picked point, and the current end of the elbow is anchored there. You are prompted again:

Please pick direction : or [plANE/anGle of plane/SNap<on>]:

- **Pick Direction** – Select the direction of the elbow’s next point.
- **Plane** - Toggles the plane of the first end between the three standard UCS planes.
- **Angle of plane** – Rotate the angle of the plane of the NavAid away from the current plane.
- **Snap** – Snap direction to the NavAid. This may be turned on or off. The current status is displayed in <angle brackets>.



5. The NavAid remains at the first picked point, but it rotates to the plane of the first profile.

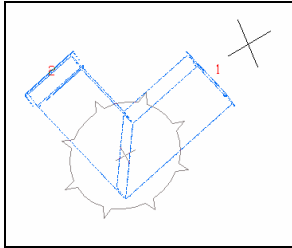


Please pick orientation : or [anGle of plane/SNap<on>]:

- **Pick Orientation** – Select the orientation of the elbow.
- **Angle of plane** – Rotate the angle of the plane of the NavAid away from the current plane.
- **Snap** – Snap direction to the NavAid. This may be turned on or off. The current status is displayed in <angle brackets>.

Note: If the elbow is connected to another duct then only orientations that will maintain the connection will be available.

Creation of Elbow in Corner Mode, Disconnected



1. Start the Elbow function.
2. Complete any On-The-Fly duct creation required (see **On-The-Fly Duct Creation**).
3. An elbow duct is displayed with your cursor at the center of the elbow (in Corner Mode). You are prompted to pick the first point or choose another option:

Please pick point : or [Corner mode/Modify/sWap ends <1>]:

- **Pick Point** – Select the location of the elbow’s center, anywhere in the model. To connect to another duct, see *Creation of Elbow in Corner Mode, Aligned to Other Ducts* below.
- **Corner Mode** – Toggle between Corner mode and End mode.
- **Modify** – Make changes to the On-The-Fly part before placing it in the model.
- **Swap Ends** – Swaps the current end, displayed in <angle brackets>. In Corner Mode, the current end is used to pick the direction of the elbow in the next step.

4. The NavAid is shown at the first picked point where the center of the elbow is anchored. You are prompted again:

Please pick direction : or [plANE/anGLE of plane/SNap<on>]:

- **Pick Direction** – Select the orientation of the elbow.
- **Plane** – Toggles the plane of the NavAid .
- **Angle of plane** – Rotate the angle of the plane of the NavAid away from the current plane.
- **Snap** – Snap direction to the NavAid. This may be turned on or off. The current status is displayed in <angle brackets>.

Creation of Elbow in Corner Mode, Aligned to other Ducts

1. Start the Elbow function.
2. Complete any On-The-Fly duct creation required (see **On-The-Fly Duct Creation**).
3. An elbow duct is displayed with your cursor at the center of the elbow (in Corner Mode). You are prompted to pick the first point or choose another option:

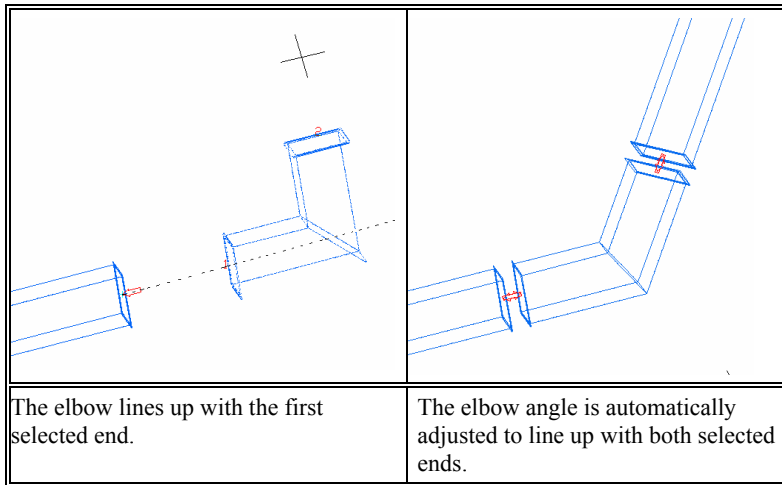
Please pick point : or [Corner mode/Modify/sWap ends <1>]:

- **Pick Point** - To align to another duct, select any free end in the model (indicated by an arrow pointing at the end of a duct).
- **Corner Mode** - Toggle between Corner mode and End mode.
- **Modify** - Make changes to the On-The-Fly part before placing it in the model.
- **Swap Ends** – Swaps the current end, displayed in <angle brackets>. In Corner Mode, the current end is used to pick the direction of the elbow in the next step.

4. The current end's direction is locked to the axis of the selected free end. The Elbow can move in a line away from the free end. You are prompted again:

Please pick direction :

- **Pick Direction** - Select the location and direction of the elbow along the line of the picked free end. You may pick another free end to center the elbow at the intersection of the two free ends. (See diagram below.)



Ducts – Tee

Menu: **SC HVAC / Ducts / Tee**

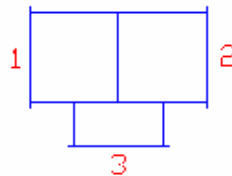
Toolbar Button: 

Command Line: **SCHVACTEE**

Navigator: **None**

Permissions: **HVAC**

Places a Tee duct. A tee may be placed by one of its ends or by the center of the tee. The diagram below lists the end points for a tee.



Placing a Tee

Placement of a tee can be broken down into three different categories:

- **End Mode** – The first picked point determines the location of the tee's selected end.

- **Corner Mode, Disconnected** – The first picked point determines the tee’s center point.
- **Corner Mode, Aligned** – The tee will be aligned to the selected ducts.

Creation of a Tee in End Mode

1. Start the Tee function.
2. Complete any On-The-Fly duct creation required (see **On-The-Fly Duct Creation**).
3. A tee duct is displayed with your cursor at one end (in **End Mode**). You are prompted to pick the first point or choose another option:

```
Please pick point : or [Corner mode/Modify/sWap ends <1>]:
```

- **Pick Point** – Select the location anywhere in the model.
- **Corner mode** – Type “C” to toggle to Corner mode. End mode places the tee by one of the ends. Corner mode places the tee by the center.
- **Modify** – Make changes to the On-The-Fly part before placing it in the model.
- **Swap Ends** – Swaps the current end, displayed in <angle brackets>. In End Mode, the current end is located at the cursor and is placed at the picked point. The current end number is also stated after the keyword.

Note: If the picked point is a free end of an existing duct then the tee will connect to the free end, the tee’s connecting end’s profile will be changed to match the profile it is connecting to, and step 4 will be skipped.

4. The NavAid is shown at the first picked point, and one end of the tee is anchored there. You are prompted again:

```
Please pick direction : or [plANE/anGLE of plane/SNap<on>]:
```

- **Pick Point** – Select the direction of the tee’s next end.
 - **Plane** – Toggle the plane of the NavAid.
 - **Angle of plane** - Rotate the angle of the plane of the NavAid away from the current plane.
 - **Snap** - Snap to the NavAid. This may be turned on or off. The current status is displayed in <angle brackets>.
2. The NavAid remains at the first picked point, but it rotates to the plane of the first profile.
 3. The last point to pick is the duct’s orientation. You are prompted again:

```
Please pick orientation : or [anGLE of plane/SNap<on>]:
```

- **Pick Point** – Select the orientation of the tee.
- **Angle of plane** - Rotate the angle of the plane of the NavAid away from the current plane.
- **Snap** - Snap to the NavAid. This may be turned on or off. The current status is displayed in <angle brackets>.

Note: If the tee is connected to another duct then only orientations that will maintain the connection will be available.

Creation of Tee in Corner Mode, Disconnected

1. Start the Tee function.
2. Complete any On-The-Fly duct creation required (see **On-The-Fly Duct Creation**).
3. Type “C” to toggle to corner mode. A tee duct is displayed with your cursor at the center of the tee. You are prompted to pick the first point or choose another option:

Please pick point : or [eNd mode/Modify/sWap ends <1>]:

- **Pick Point** – Select the location of the tee’s center, anywhere in the model.
- **Corner Mode / End Mode** – Toggle between Corner Mode and End Mode.
- **Modify** – Make changes to the On-The-Fly part before placing it in the model.
- **Swap ends** – Swaps the current end, displayed in <angle brackets>. In Corner Mode, the current end is used to pick the orientation of the tee in the next step.

4. The NavAid is shown at the first picked point, where the center of the tee is anchored. You are prompted again:

Please pick direction: or [plANE/angle of plane/SNap<on>]:

- **Pick Point** – Select the orientation of the tee.
- **Plane** – Toggle the plane of the NavAid.
- **Angle of plane** - Move the angle of the plane of the NavAid away from the current plane.
- **Snap** - Snap to the NavAid. This may be turned on or off. The current status is displayed in <angle brackets>.

Creation of Tee in Corner Mode, Aligned to other Ducts

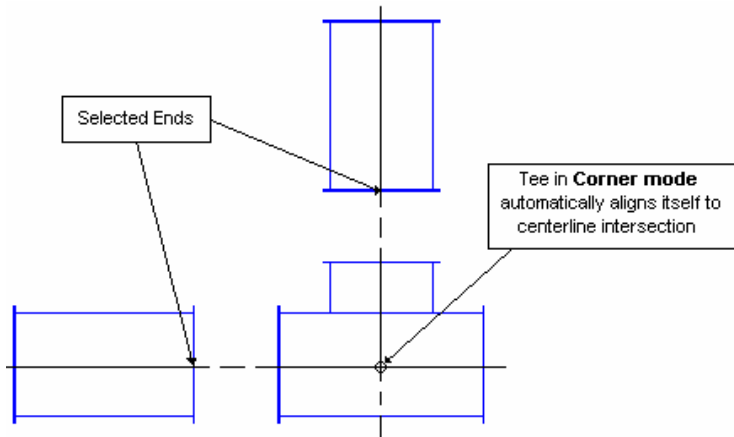
1. Start the Tee function
2. Complete any On-The-Fly duct creation required.
3. A tee duct is displayed with your cursor at the center of the tee. You are prompted to pick the first point or choose another option, picking an existing duct’s free end as the point will cause the tee to be aligned to the free end:

Please pick point : or [Corner mode/Modify/sWap ends <1>]:

- **Pick Point** – To align to another duct, select any free end in the model (indicated by an arrow pointing at the end of a duct).
 - **Corner Mode** – Toggle between Corner mode and End mode.
 - **Modify** – Make changes to the On-The-Fly part before placing it in the model.
 - **Swap Ends** – Swaps the current end, displayed in <angle brackets>. In Corner Mode, the current end is used to set the direction of the tee in the next step.
4. The current end’s direction is locked to the selected free end axis. The tee can move along the axis away from the free end. You are prompted again:

Please pick direction :

- **Pick Direction** - Select the location and direction of the tee along the line of the picked free end. You may pick another free end to center the tee at the intersection of the two free ends. The angle of the third end will change to align to the picked free end.



Ducts – Cross

Menu: **SC HVAC / Ducts / Cross**

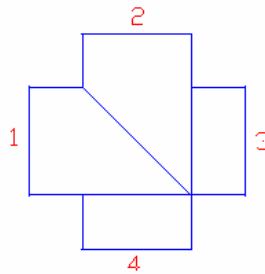
Toolbar Button: 

Command Line: **SCHVACCROSS**

Navigator: **None**

Permissions: **HVAC**

Places a Cross Duct. A cross duct may be placed by one of its ends or by the center of the cross. The diagram below lists the end points for a cross.



Placing a Cross

Placement of a cross can be broken down into three different categories:

- **End Mode** – The first picked point determines the location of the cross's selected end.
- **Corner Mode, Disconnected** – The first picked point determines the cross's center point.
- **Corner Mode, Aligned** – The cross will be aligned to the selected ducts.

Creation of a Cross in End Mode

See Tee Duct.

Creation of Cross in Corner Mode, Disconnected

See Tee Duct.

Creation of Cross in Corner Mode, Aligned to other Ducts

See Tee Duct.

Ducts – Wye

Menu: **SC HVAC / Ducts / Wye**

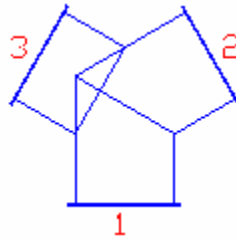
Toolbar Button: 

Command Line: **SCHVACWYE**

Navigator: **None**

Permissions: **HVAC**

Places a Wye duct. A wye may be placed by one of its ends or by the center of the wye. The diagram below lists the end points for a wye.



Placing a Wye

Placement of a wye can be broken down into three different categories:

- **End Mode** – The first picked point determines the location of the wye's selected end.
- **Corner Mode, Disconnected** – The first picked point determines the wye's center point.
- **Corner Mode, Aligned** – The wye will be aligned to the selected ducts.

Creation of a Wye in End Mode

See Tee Duct.

Creation of Wye in Corner Mode, Disconnected

See Tee Duct.

Creation of Wye in Corner Mode, Aligned to other Ducts

See Tee Duct.

Ducts – Transition

Menu: **SC HVAC / Ducts / Transition**

Toolbar Button: 

Command Line: **SCHVACTRANSITION**

Navigator: **None**

Permissions: **HVAC**

Places a transition duct. A transition duct is primarily used to connect two ducts with different profiles.

Placing a Transition

Transitions can only be placed in end mode. The selected point determines the location for the transition. The diagram below lists the end points for a transition.



Creation of Transition Duct

1. Start the Transition function.
2. Complete any On-The-Fly duct creation required (see **On-The-Fly Duct Creation**).
3. A transition is displayed with End 1 at your cursor. You are prompted to pick the first point or choose another option:

Please pick point : or [Modify/sWap ends <1>]:

- **Pick Point** – Select the start point of the duct anywhere in the model. To connect to another duct, select any free end in the model. The new duct will connect to this end.
- **Modify** – Make changes to the On-The-Fly part before placing it in the model.
- **Swap Ends** – Swaps the current end, displayed in <angle brackets>. The current end is used to set the direction of the transition in the next step.

Note: If the picked point is a free end of an existing duct then the transition will connect to the free end, the transition's connecting end's profile will be changed to match the profile it is connecting to, and step 4 will be skipped.

4. The NavAid is shown at the first picked point, and End 1 of the transition duct is anchored there. You are prompted again:

Please pick direction : or [plANE/anGle of plane/SNap<on>]:

- **Pick Point** - Select the direction of the transition.
 - **Plane** – Toggle the plane of the NavAid.
 - **Angle of plane** - Rotate the angle of the plane of the NavAid away from the current plane.
 - **Snap** - Snap direction to the NavAid. This may be turned on or off. The current status is displayed in <angle brackets>.
5. The last point to pick is the duct's orientation. You are prompted again:

Please pick orientation : or [anGle of plane/SNap<on>]:

- **Angle of plane** - Rotate the angle of the plane of the NavAid away from the current plane.
- **Snap** - Snap to the NavAid. This may be turned on or off. The current status is displayed in <angle brackets>.

Note: If the transition is connected to another duct then only orientations that will maintain the connection will be available.

Ducts – Pants

Menu: **SC HVAC / Ducts / Pants**

Toolbar Button: 

Command Line: **SCHVACPANTS**

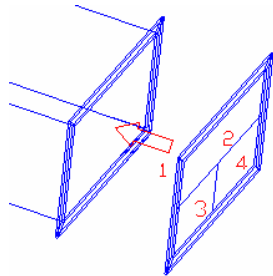
Navigator: **None**

Permissions: **HVAC**

Places a Pants duct. A pants duct is used to split a larger profile into as many as 6 smaller ends.

Placing a Pants Duct

Pants can only be placed in end mode. The selected point determines the location for the pants. The diagram below lists the end points for a pants duct. End 1 is always the main end, and subsequent ends are splitters.

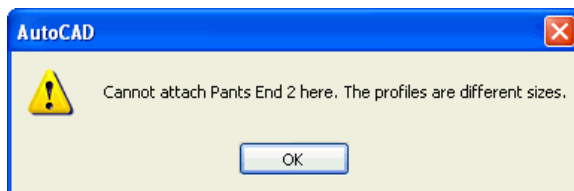


Creation of Pants Duct

1. Start the Pants function.
2. Complete any On-The-Fly duct creation required, including choosing the number and arrangement of splits (see **On-The-Fly Duct Creation**).
3. The Pants duct is displayed with the first end at your cursor. You are prompted to pick the first point or choose another option:

Please pick point : or [Modify/sWap ends <1>]:

- **Pick Point** – Select the start point of the duct anywhere in the model. To connect to another duct, select a free end. The free end must be rectangular, without rounded corners. If the current end is End 1, the Pants Duct will automatically resize to connect to the selected free end. If the current end is not End 1, the Pants Duct will not be resized, and the following dialog will appear:



- **Modify** – Make changes to the On-The-Fly part before placing it in the model.

- **Swap Ends** – Swaps the current end, displayed in <angle brackets>. The current end is used to set the direction of the pants in the next step.

Note: If the picked point is a free end of an existing duct then the transition will connect to the free end and step 4 will be skipped.

Note: If a pants duct is connected to existing ducts only the first end's profile will change to match the duct the pants is connected to. If this happens then the other ends of the pants duct will be scaled.

4. The NavAid is shown at the first picked point, and End 1 of the transition duct is anchored there. You are prompted again:

Please pick direction : or [plane/angle of plane/Snap<on>]:

- **Pick Point** – Select the direction of the pants.
- **Plane** – Toggle the plane of the NavAid.
- **Angle of plane** – Rotate the angle of the plane of the NavAid away from the current plane.
- **Snap** – Snap direction to the NavAid. This may be turned on or off. The current status is displayed in <angle brackets>.

5. The last point to pick is the duct's orientation. You are prompted again:

Please pick orientation : or [angle of plane/Snap<on>]:

- **Angle of plane** – Rotate the angle of the plane of the NavAid away from the current plane.
- **Snap** – Snap direction to the NavAid. This may be turned on or off. The current status is displayed in <angle brackets>.

Note: If the Pants are connected to another duct then only orientations that will maintain the connection will be available.

Ducts – Add Saddle

Menu: **SC HVAC / Ducts / Add Saddle**

Toolbar Button: 

Command Line: **SCHVACSADDLE**

Navigator: **None**

Permissions: **HVAC**

Adds a Saddle to an existing duct.

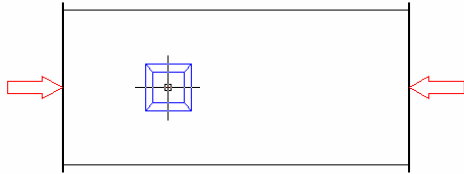
Adding a Saddle

1. Start the Add Saddle function.
2. Choose a duct to add the saddle to.

Select a duct:

- The HVAC Saddle dialog appears every time a saddle is added. It specifies a profile and end treatment for the saddle. See On-The-Fly duct creation, HVAC Saddle.
- Choose the location of the saddle along the duct's center.

Choose the Saddle location:

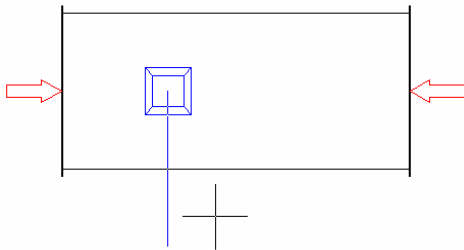


- Choose the direction of the saddle end. Pick one of the directions indicated by the blue guidelines, or enter one of the directions listed. The saddle is added to the side of the duct that is closest to the picked direction.

Choose the Saddle end direction [Centered/Port/Aft/Fwd/Starboard/Up]:

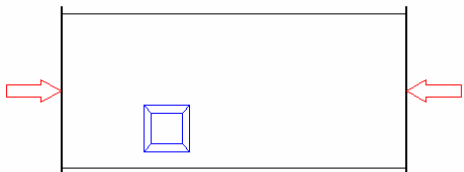
- Choose the direction of the saddle's offset from the center of the duct. Either use the blue guidelines displayed on the duct, or enter one of the two directions listed.

Choose the Saddle Offset direction [Aft/Fwd]:

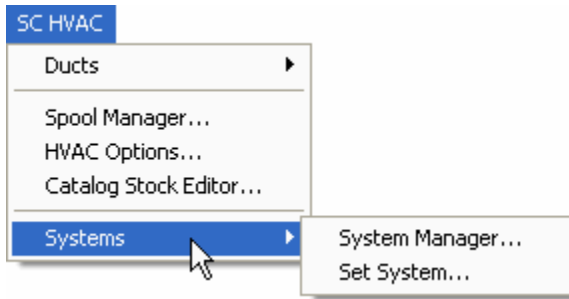


- Enter the Saddle Offset value. For no offset, press Enter or type 0 and the saddle is placed in the center of the duct's side.

Enter the Saddle Offset value:



Systems Sub-Menu



The Systems Sub-Menu provides functionality for creating and modifying HVAC systems, and for assigning systems to ducts in an open HVAC model.

Note: Systems for HVAC has been designed in the same fashion as for piping.

System Manager

Menu: **SC HVAC / System / System Manager**

Toolbar Button: **None**


Command Line: **SCHVACEDITSYSTEM**

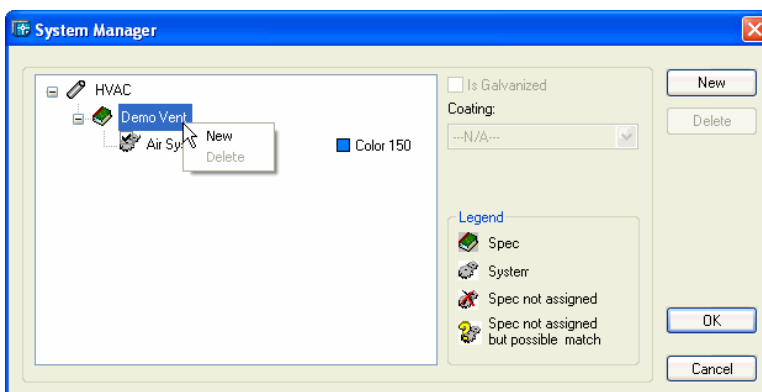
Navigator: **None**

Permissions: **None**

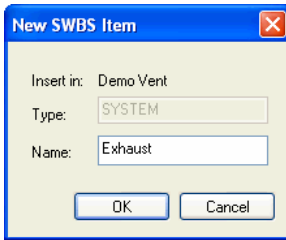
Creates and edits HVAC systems. HVAC systems can have two levels: system and branch. Systems are grouped under Specs thereby indicating that they are assigned to that spec. Therefore, when placing ducts, one only needs to specify the System as the Spec is implied by the System.

New System

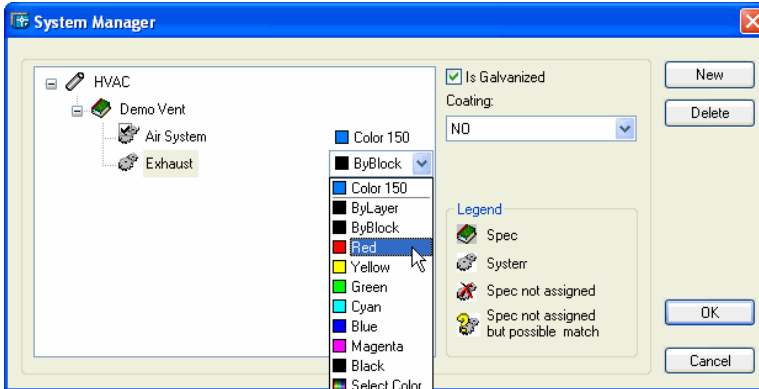
1. Select **SC HVAC / Systems / System Manager...** The **System Manager** dialog is displayed.
2. Select a Spec, , from the System tree.
3. Press **New** or use the right-click menu and click New.



4. Enter a name for the new System in the New SWBS Item dialog box.





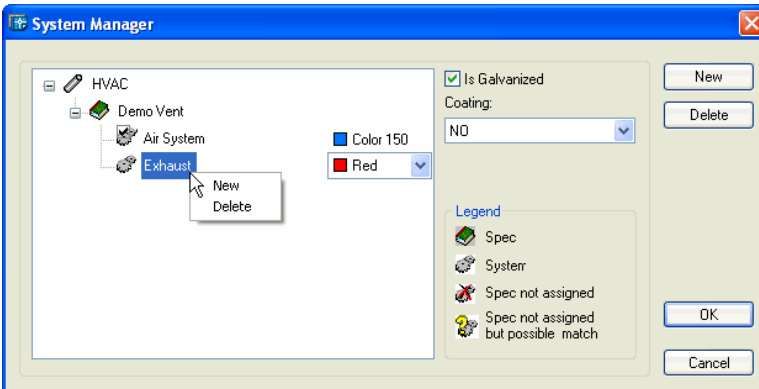
5. Select the Color you wish the systems' ducts to appear in your HVAC models.



6. Set the properties of the new system i.e. Coating and Is Galvanized.
7. Press **OK** when done to accept or **Cancel** to discard the changes.

New Branch

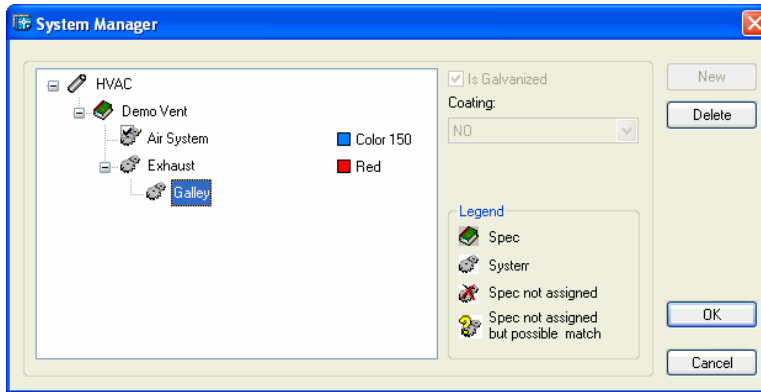
1. Select **SC HVAC / System / System Manager...** The **System Manager** dialog is displayed.
2. Select a System () under a Spec (), from the tree.
3. Press **New** or use the right-click menu and click New.



4. Enter a name for the new Branch in the New SWBS Item dialog box.



5. The properties of the new branch are derived from the parent System and cannot be edited.



6. Press **OK** when done to accept or **Cancel** to discard the changes.

To Promote/Demote or reassign Systems to other Specs

1. Select **SC HVAC / System / System Manager...** The **System Manager** dialog is displayed.
2. Select the system or branch.
3. Drag the node to another location on the tree, creating either a System or Branch.
4. Press **OK** to accept the changes or **Cancel** to discard the changes.

Note: Dragging a node to a system reassigns the properties of the dragged node to that of the target system.

To Modify System Properties

Note: System property changes propagate through all branches

1. Select **SC HVAC / System / System Manager...** The **System Manager** dialog is displayed.
2. Select the system for which the properties are to be changed.
3. Change the properties of the system (Name, Color, Coating, and Is Galvanized) as desired.

To Delete a System or Branch

1. Select **SC HVAC / System / System Manager...** The **System Manager** dialog is displayed.
2. Select the system or branch to delete.
3. Press **Delete** or use the right-click menu and click Delete.

Note: You cannot delete a system or branch if ducts have been assigned to it.

Set System

Menu: **SC HVAC / Systems / Set System**

Toolbar Button: **None**

Command Line: **SCHVACSETSYSTEM**

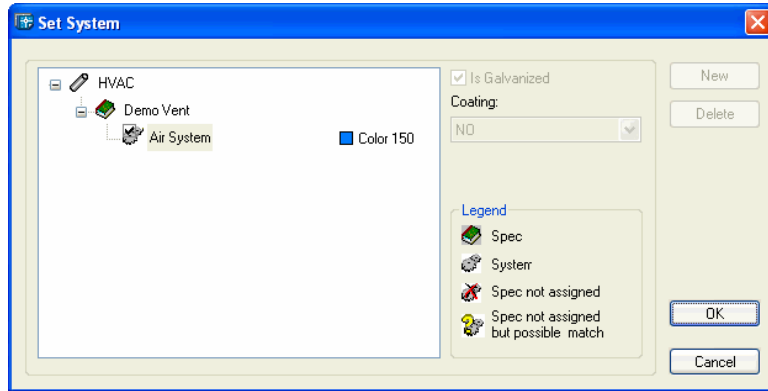
Navigator: **None**

Permissions: **None**

Changes the System assigned to a duct when in an HVAC Model Drawing.

Set the System or Branch of a Duct

1. Start the **Set System** command.
2. Select the duct whose system needs to be changed.
3. From the Set System dialog, select the desired system.
4. Press **OK**.

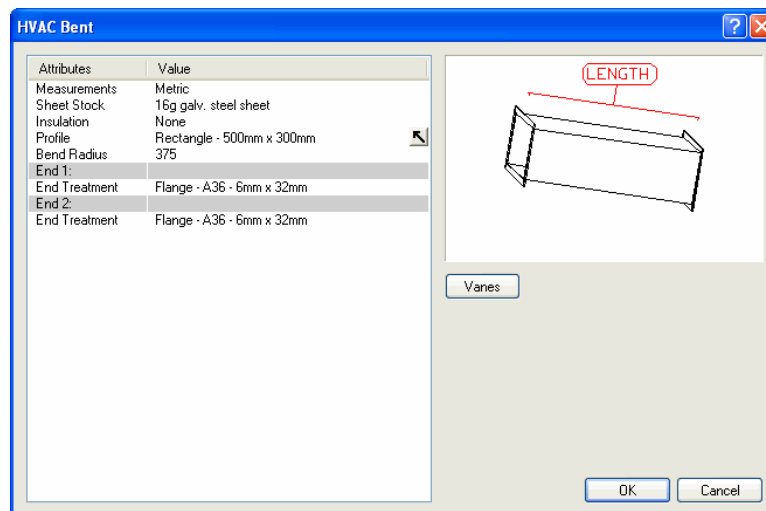


On-The-Fly Duct Creation

Overview

It is convenient to create individual ducts on the fly, by specifying their dimensions, end treatments, sheet stock, etc. as they are needed. The **On-The-Fly Dialog** allows for the creation and modification of on-the-fly ducts.

The **On-The-Fly Dialog** appears when an HVAC type is first created. It is used to specify the HVAC part before it is created and placed in the model. Subsequent ducts will use the same specifications. During the placement of a duct (see **Ducts Sub-Menu**) the “Modify” command will bring up the On-The-Fly Dialog to view and edit the current duct’s attributes.



On-The-Fly Basics

Before you can create your first duct, you must define the following:

1. Sheet Stock, defined in **Manager**.
2. Insulation, defined in **Manager**.
3. End Treatments, defined in the **HVAC Stock Editor**.

Duct Attributes

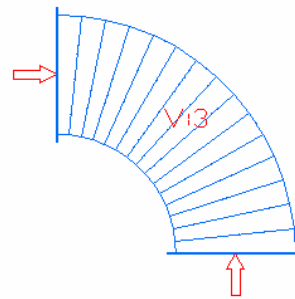
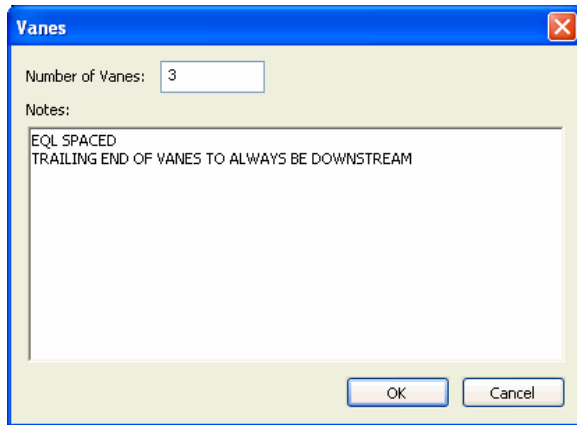
The rows in the list on the left side may be edited by double-clicking on them. In some cases, such as when editing a connected duct, some ends or attributes may be locked.

Duct Preview

The preview on the right may be rotated and zoomed, using the same controls as AutoCAD's orbit. It also has a right-click display options menu, and it contains red dimension labels.

Vanes

The Vanes button is located below the preview pane. It brings up the Vanes dialog. Enter the number of vanes and any information about them. The number of vanes will be indicated on the duct when it is placed in a model (see below).



Common Attributes

Each type of HVAC has these common attributes, always at the top of the attributes list.

Measurements – The system of measurement, either Metric or Imperial. This defaults to the project settings, found in Manager.

Sheet Stock – Select from the stocks defined in Manager. The sheet stock specifies the material and thickness of this duct's sheet.

Insulation – Select from the Insulation types defined in Manager.

End Attributes

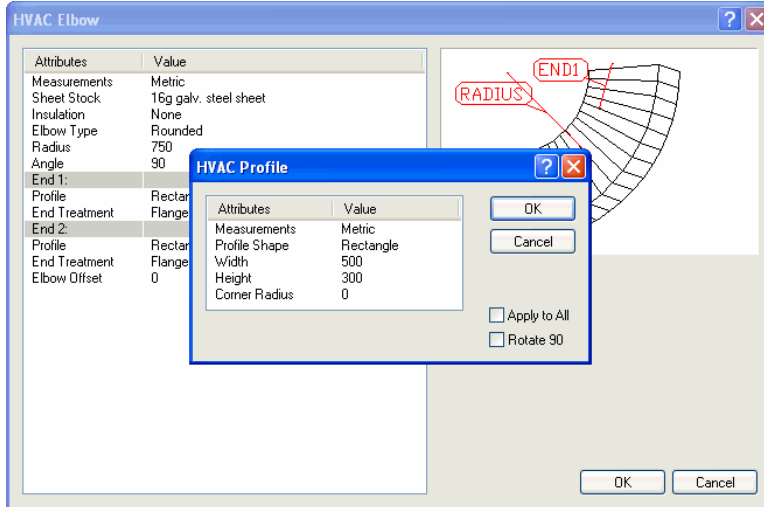
Ducts have multiple ends. Depending on the HVAC type, each end has several properties listed under a grey end heading row. Some types may have unique end attributes, as described in the next section.

Profile


The shape and dimensions of the end profile. A profile row in the HVAC On-The-Fly Dialog can be edited in two ways. Double-clicking the profile value brings up the **HVAC Profile Dialog** which is used to set the exact dimensions and shape of the profile. The **Profile Picker** button picks up a profile definition from any existing duct in your model.

Profile Dialog

Enter the Profile type and dimensions. Check “Apply To All” to set each of the ducts ends. Check “Rotate 90” to swap the two dimensions of a rectangle or ellipse.



Profile Picker

Click the Profile Picker button  and you are prompted to select a profile on a duct in your model. The text in the Profile row will change to describe the new profile.

End Treatment

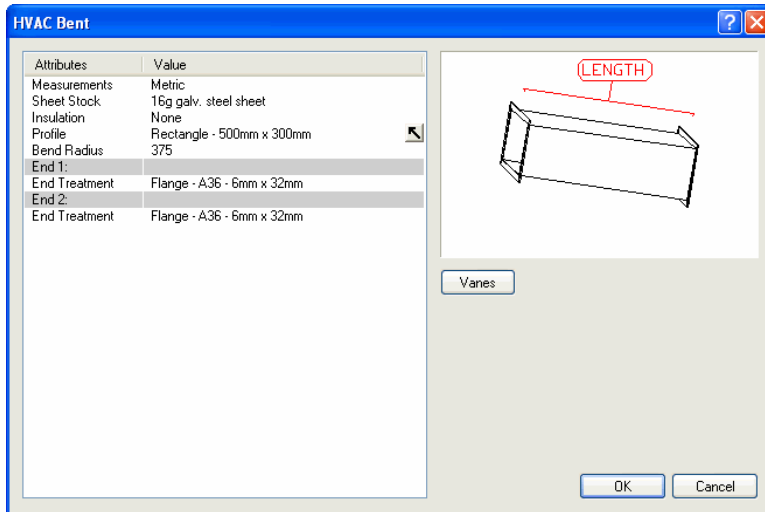
Select from a list of End Treatments which you defined in the Stock Editor.

Modify/Create HVAC of Different Types

Each HVAC type may be created On-The-Fly. Each type has the Common Attributes listed above, as well as a unique combination of other attributes.

HVAC Bent

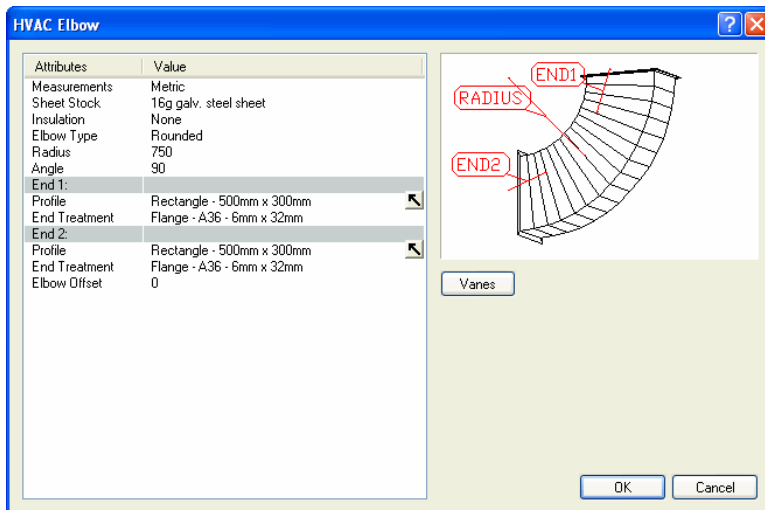
An HVAC Bent is used to draw a straight or bent duct. Both ends of the duct have the same profile, but the end treatments may differ. The End Treatments are listed separately for End 1 and End 2.



Bend Radius

The bent duct has a default bend radius, which is used when the duct is placed in rounded bend mode. This specifies the radius of the rounded bend corners, and it may also be changed during duct placement for each corner.

HVAC Elbow



Elbow Type

The Elbow Type may be Mitered or Rounded.

A **Mitered** elbow has two end lengths (measured from the center of the elbow). Each end also has a Profile and End Treatment.

A **Rounded** elbow has several unique attributes:

Radius

The elbow radius measures the curve of a rounded elbow. It is shown in the preview.

Elbow Offset

The elbow offset is shown for the second end of a rounded elbow only. It allows you to set a vertical offset between the two ends.

Angle

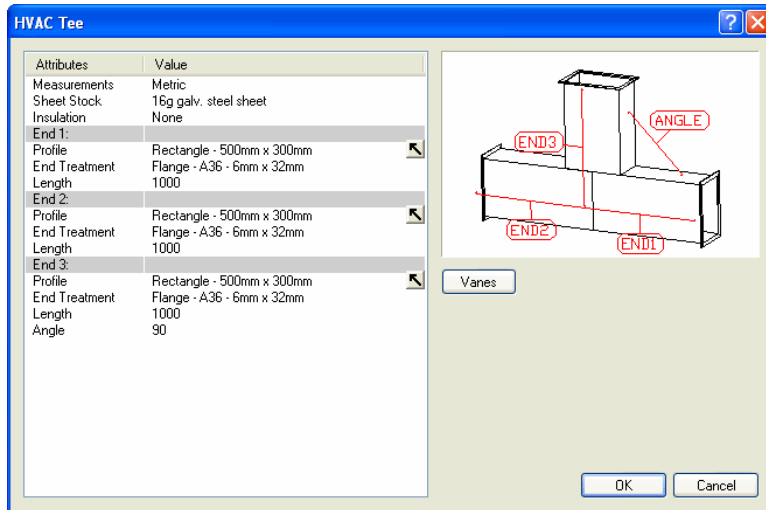
For both Mitered and Rounded elbows, the angle is measured between the two end profiles.

HVAC Tee

A tee has three ends. End 1 and End 2 are always in a straight line. End 3 is usually at 90 degrees, but this angle may be changed.

Each end may have a different Profile, End Treatment, and Length. Lengths are measured from the center of the tee where the three ends intersect.

End 3 has an Angle, measured between End 1 and End 3. The angle is shown in the preview on the right.

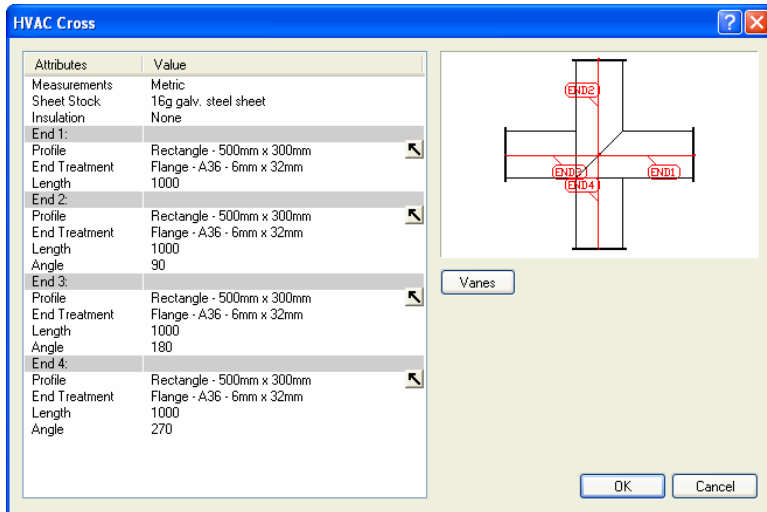


HVAC Cross

A cross has four ends.

Each end may have a different Profile, End Treatment, and Length. Lengths are measured from the center of the cross where the four ends intersect.

End 2, End 3 and End 4 have Angles, measured from End 1. For example, a square cross would have angles of 90°, 180°, and 270°.

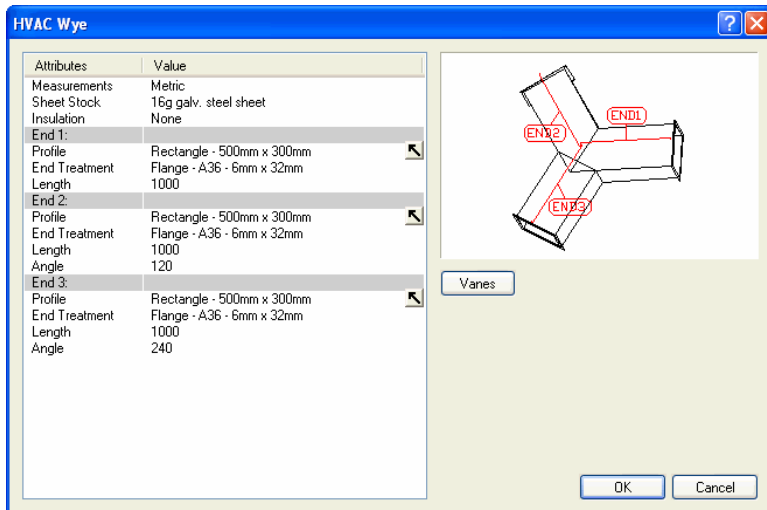


HVAC Wye

A Wye has three ends.

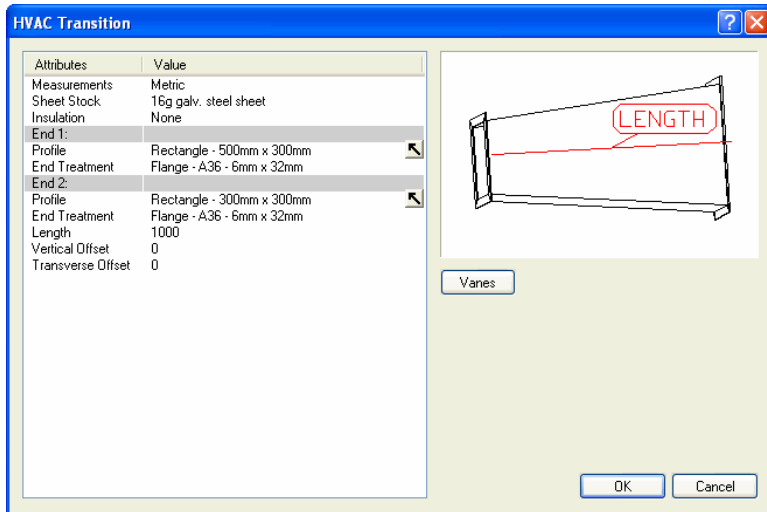
Each end may have a different Profile, End Treatment, and Length. Lengths are measured from the center of the wye where the three ends intersect.

End 2 and End 3 have Angles, measured from End 1. For example, the wye below has angles of 120°, and 240°.



HVAC Transition

A Transition has two ends, which may have different Profiles and End Treatments.



Length

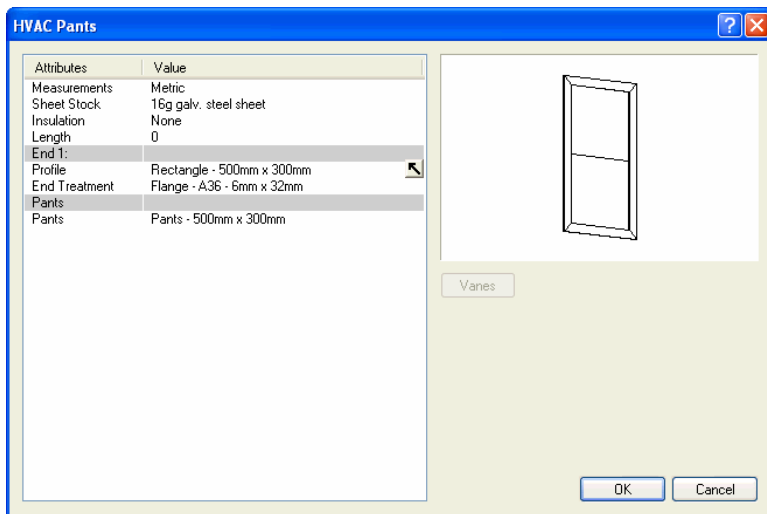
The length is the distance from End 1 to End 2, as shown in the Preview.

Vertical Offset and Transverse Offset

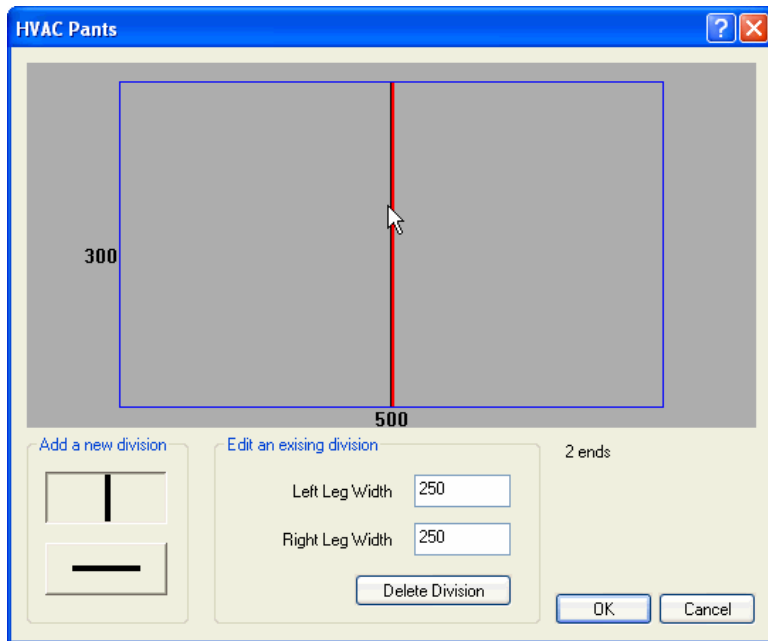
The offsets between the two transition ends.

HVAC Pants

A Pants duct is used to split a larger profile into a variable number of smaller ends. It has a length, which may be zero.





All ends in a Pants duct have rectangular profiles with square corners. End 1 is the largest, and it may also have an End Treatment. End 1's profile is split into multiple "Pants Ends", which can be edited with the HVAC Pants Editor.



The HVAC Pants Editor shows a cross-section of the Pants duct. Vertical and horizontal divisions may be added, modified and deleted.

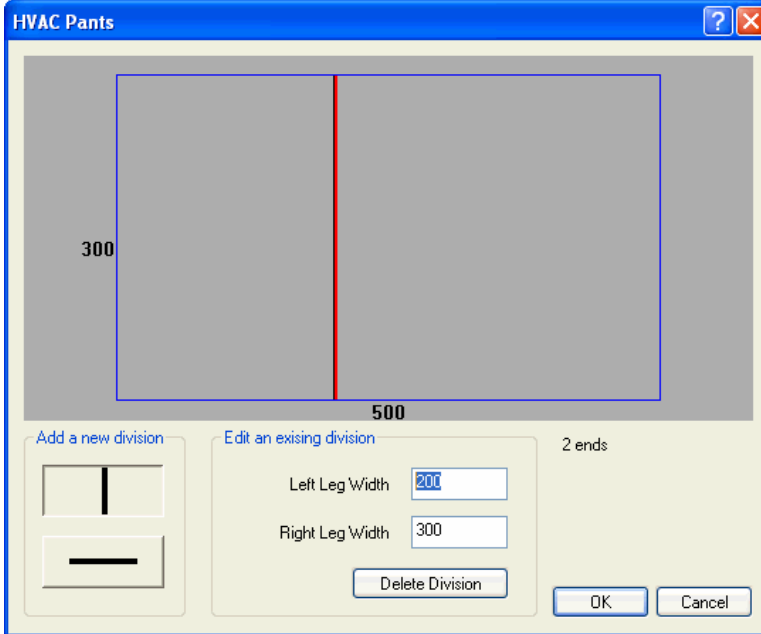
To Add Divisions

1. Select either a vertical  or horizontal  division by clicking one of the buttons under **Add a new division**
2. Click inside the Pants cross-section to divide it in half.
3. Edit the dimensions of the two ends in the **Edit an existing division** section, by changing the values of the two Pant “Legs”
4. Divisions may be added until there are up to six ends.

To Modify or Delete Divisions

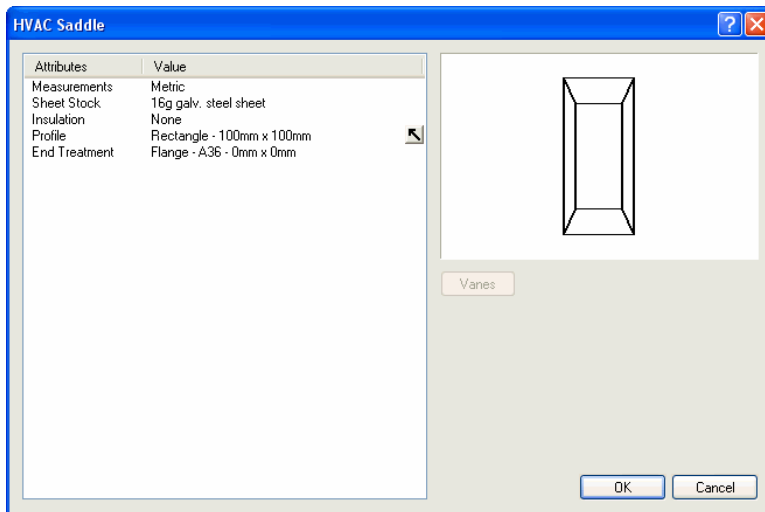
1. Select a division by clicking on the division line. The selected line will be shown in red and two ends that it splits will be shown in blue.

2. Edit or Delete the selected division by changing the values in the two leg boxes or by clicking “Delete Division”

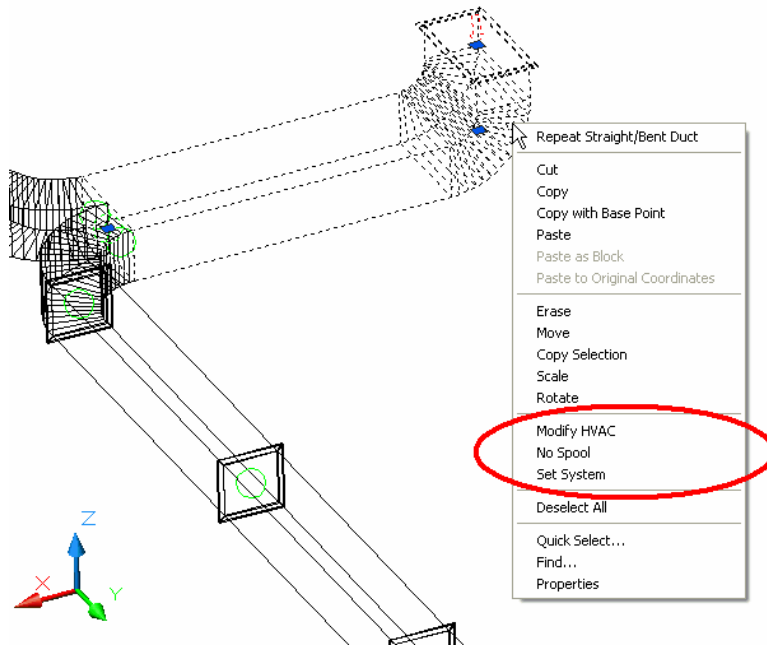


HVAC Saddle

When a Saddle is added to an existing duct, the HVAC Saddle Dialog will be displayed to specify the Saddle’s dimensions. A Saddle has one Profile and one End Treatment.



HVAC Context Menu



The HVAC Context Menu is available when an HVAC Model is open. It is used by selecting one or more ducts and right-clicking.

Modify HVAC

This option is available when only one duct is selected. It brings up the On-The-Fly dialog, which can be used to modify the selected duct. (See On-The-Fly Duct Creation) If the duct is connected, then some ends and profiles may be locked.

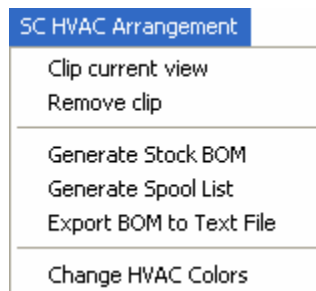
No Spool

The No Spool Option sets the selected duct(s) to No-Spool. A check next to the No Spool option indicates that the selected duct(s) are No-Spool. If the ducts(s) are already No-Spool, they will be unspooled.

Set System

This option is identical to **SC HVAC / Systems / Set System...** It changes the System assigned the selected duct(s).

Arrangement Menu



The SC HVAC Arrangement menu is only available when in an HVAC Arrangement Drawing. It includes all the functions available in the Piping Arrangement Menu.

Use the standard ShipConstructor functions, such as Attach XREF's to populate the drawing with structure, outfit, pipe and HVAC models.

SC HVAC Arrangement - Clip current view

Menu: **SC HVAC Arrangement / Clip current view**

Toolbar Button: **None**

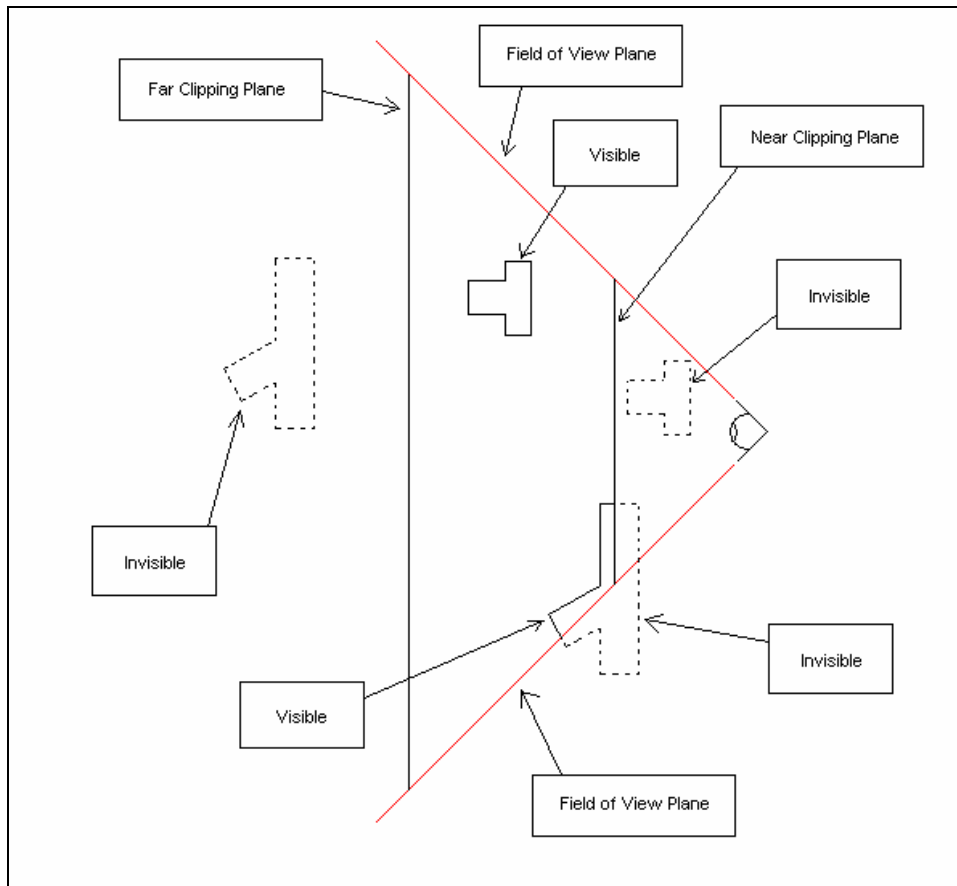
Command Line: **SCCLIPVIEW**

Navigator: **None**

Permissions: **None**

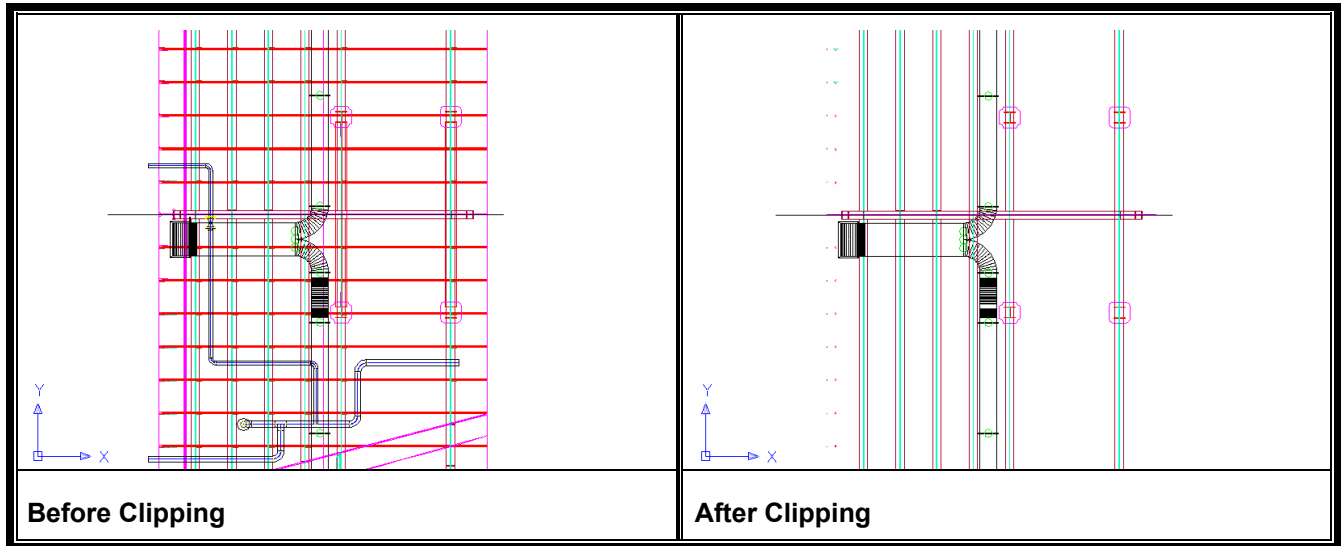
Clip current view is a utility function that provides a convenient method for setting up AutoCAD clipping planes inside of a viewport.

Note: Before running the command you need to switch to the viewport in which the clipping plane is to be applied.



1. The command line prompts you to: **Enter Point on Plane 1:** Choose a point on one of the clipping planes to be used (whether it is the front or back clipping plane will be determined by **ShipConstructor**).
2. The command line prompts you to enter a second point: **Enter Point on plane 2:** Choose a point on the second clipping plane (whether it is the front or back clipping plane will be determined by **ShipConstructor**).

Note: If the clipping planes are not to your liking you can remove them with the command **SC Arrangement - Remove clip** (see below).



SC HVAC Arrangement - Remove clip

Menu: **SC HVAC Arrangement / Remove clip**

Toolbar Button: **None**

Command Line: **SCCLEARCLIP**

Navigator: **None**

Permissions: **None**

This command removes the AutoCAD clipping planes from the currently selected viewport.

SC HVAC Arrangement - Generate Stock BOM

Menu: **SC HVAC Arrangement / Generate Stock BOM**

Toolbar Button: **None**

Command Line: **SCHVACARRANGEBOM**

Navigator: **None**

Permissions: **None**

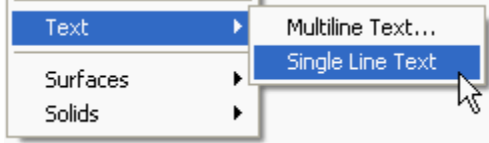
Creates a Bill of Materials in the arrangement drawing. BOM's are usually generated from paper space.

Note: In order to Generate Stock BOM's, BOM definitions must exist. See **Navigator/HVAC/HVAC BOM** on how to customize and create BOM's.

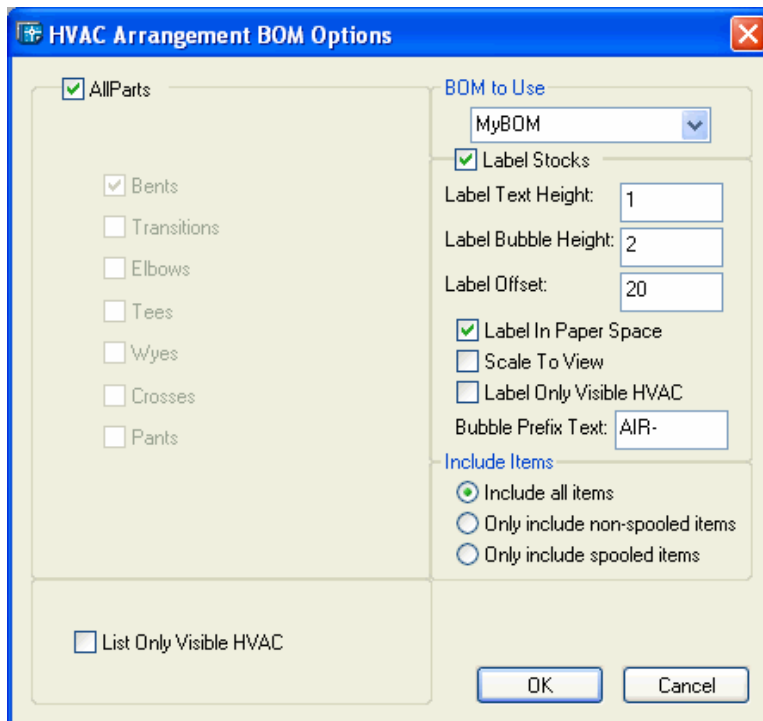
1. Start the **Generate Stock BOM** function.
2. You are prompted to select a text entity. The command requires you to select a text entity (see the Note below) in order to get the insertion point for the **BOM** and to get the text style information, which will be used to format the **BOM**. This way the user can create a custom Arrangement Template that will help keep things standardized across the project.

Select Text entity to set the BOM properties from:

Note: You cannot use a multi-line text entity with this function. Use either the “text” command on the command line to enter a single line text entity or use the **Single Line Text** from the AutoCAD Draw drop down menu.

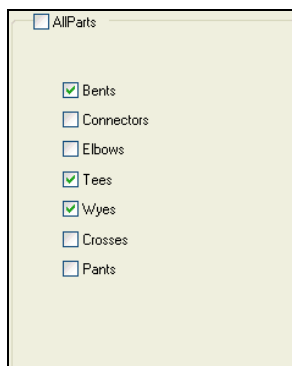


3. The **HVAC Arrangement BOM Options** dialog will appear. Once all settings (described below) have been configured to user satisfaction click **OK** to generate the **BOM** and the labels in the Arrangement Drawing.



All Parts

The left of the dialog box contains a number of check boxes which are used to customize what will be included in the **BOM** (see below). The **All Parts** option will automatically include all types of HVAC, or the individual types can be selected to fully customize the **BOM**.



List Only Visible HVAC - Only visible HVAC parts will be included in the **BOM**. Those outside the clipping planes and the viewport will be ignored.

List Only Visible HVAC

BOM To Use

BOM to Use

MyBOM

Label Stocks

Label Text Height: 1

Label Bubble Height: 2

Label Offset: 20

Label In Paper Space

Scale To View

Label Only Visible HVAC

Bubble Prefix Text: AIR-

Include Items

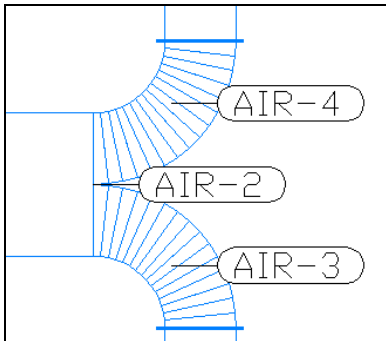
Include all items

Only include non-spoiled items

Only include spoiled items

BOM to Use - The pre-defined **BOM** to use. See **Navigator/HVAC/HVAC BOM** on how to customize and create BOM's.

Label Stocks - Checking this option will cause all items in the BOM to be labeled with fly-out markers.



In the example above, "AIR-" is **Bubble Prefix Text** used to identify the pipe as being from the Air system, the three numbers following reference parts in the **BOM** as seen below.

	TYPE:	PAINT:	SPOOL:
1.	ELBOW	NO	J303-AIR-5
2.	DUCT	NO	J303-AIR-5
3.	ELBOW	NO	J303-AIR-5
4.	CONNECTOR	NO	J303-AIR-5
5.	CONNECTOR	NO	J303-AIR-5

Label Settings:

- **Label Text Height** - Height of Label Text in drawing units.
- **Label Bubble Height** - Additional Bubble Height combined to the Text Height to generate a total bubble height.
- **Label Offset** - Length of the offset leader line in drawing units.
- **Label In Paper Space** - Place labels in Paper Space.

- **Scale To View** - Scales the labels to the zoom level of the viewport so they always appear to be the same size. This option is only available if the labels are being placed in Model Space; however the units are measured in Paper Space.
- **Label Only Visible HVAC** - Label only ducts that are visible in the current viewport (ignore those that have been clipped).
- **Bubble Prefix Text** - Text to be placed in front of the **BOM** number in the Label. (can be left blank).

Include Items

- **Include All Items** - Includes all HVAC items in the BOM. This is the default.
- **Only include non-spooled items** - Includes only HVAC that are not assigned to spools.
- **Only include spooled items** - Include only HVAC items that have been assigned to spools.

SC HVAC Arrangement - Generate Spool List

Menu: **SC HVAC Arrangement / Generate Spool List**

Toolbar Button: **None**

Command Line: **SCHVACSPPOOLBOM**

Navigator: **None**

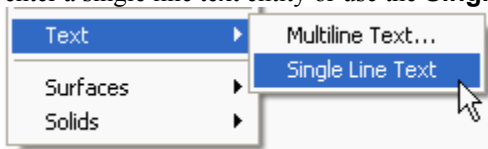
Permissions: **None**

Creates a list of Spools in the arrangement drawing. Spool Lists are usually generated from paper space.

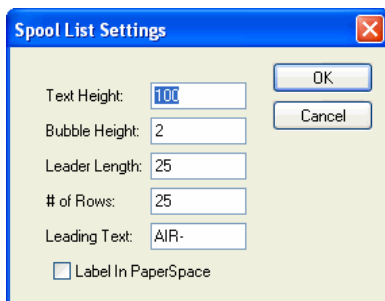
1. Start the **Generate Spool List** function.
2. You are prompted to select a text entity. The command requires you to select a text entity (see the Note below) in order to get the insertion point for the List and to get the text style information, which will be used to format the **BOM**. This way the user can create a custom Arrangement Template that will help keep things standardized across the project.

Select Text entity to set the BOM properties from:

Note: You cannot use a multi-line text entity with this function. Use either the “text” command on the command line to enter a single line text entity or use the **Single Line Text** from the AutoCAD Draw drop down menu.



3. The **Spool List Settings** dialog will appear allowing the user to customize many facets of the List to be generated. Once the settings have been configured to user satisfaction (see below) click **OK** to generate the List and labels.



Text Height - Height of Label Text in drawing units.

Bubble Height - Additional Bubble Height combined to the Text Height to generate a total bubble height.

Leader Length - Length of the offset leader line in drawing units.

of Rows - Maximum number of rows the Listing can have. If the number of spools exceeds this number then multiple columns will be created.

Leading Text - Text to be placed in front of the Listing number in the Label (can be left blank).

Label In Paper Space - If this option is checked then the labels are placed in Paper Space.

Below is an example Spool List, as inserted into an Arrangement Drawing.

	SPPOOL NAME
1.	J303-AIR-4
2.	J303-AIR-5
3.	J303-AIR-6

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