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Pipe Reference

Overview

Pipe is a fully integrated program module of the ShipConstructor suite. As such, many of the functions used during Pipe are also used in the ShipConstructor - Structure, Outfit, and Nest modules. Please familiarize yourself with the basic functions as outlined in the tutorial section of the Structure module.

The functionality of the Pipe module is accessed through the ShipConstructor Navigator and the Pipe menu. Several of the most popular functions are also available in the Pipe Toolbar. Below you will find screen shots of the ShipConstructor Navigator tab, Pipe menu and the Pipe toolbar.

Note: In order to activate the Pipe tab in the ShipConstructor Navigator the user must login to the project’s database by registering a 3D unit.

Note: The Pipe menu is only available when in a Pipe drawing.
Many functions are conveniently available using the Pipe toolbar as shown below.
Navigator / Pipe

New

Menu: None
Toolbar: None
Command Line: None
Navigator: Pipe / New

Creates a new, empty Pipe drawing or arrangement drawing.

To Create a New Pipe Drawing:
1. Select the Piping folder icon.
2. Click New.
3. In the New Drawing Dialog enter the Pipe drawing name.
4. Press OK. The drawing empty is created and immediately opened for editing.
5. The SC Piping menu is now displayed.
To Create a New Arrangement Drawing:

1. Select the Piping folder icon.
2. Click New.
3. In the New Drawing Dialog enter the drawing name.
4. Press OK. The drawing empty is created and immediately opened for editing.
5. The SC Arrangement menu is now displayed.

Open

Menu: None
Toolbar: None
Command Line: None
Navigator: Pipe / Open
Opens an existing Pipe drawing.
1. Highlight the drawing to be opened in the tree dialog.
2. Click Open.

Delete

Menu: None
Toolbar: None
Command Line: None
Navigator: Pipe / Delete
Deletes the selected Pipe drawing.
1. Highlight the drawing to be deleted in the tree dialog.
2. Click Delete.

Spool BOM

Menu: None
Toolbar: None
Command Line: None
Navigator: Pipe / Edit BOM
Use for editing and/or creation of 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 spool drawing.
1. Click **Edit BOM**.

2. The following dialog allows the user to fully create or modify bill of material definitions.

### Pipe BOM’s
- **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 within the **Current BOM** frame.
- **Copy** - Copies a previous pipe BOMs definition from the currently selected one. Use this if another BOM definition is similar in layout to an already existing one.
- **Delete** - Deletes the currently selected BOM definition. If you attempt to delete a BOM definition, which existed before the current session of pipe BOM editing, the following warning will appear to confirm your intentions.

### Current BOM
- The currently selected BOM definition is reflected in the edit box. Fields not already added to the definition remain in the Available Fields list and those already added appear in the Fields list.
- **Add** - Add the selected Available field to the definition.
- **Remove** - Remove the selected field from the Fields list.
- **Move Up / Move Down** - Move the selected fields up or down. The order from top to bottom represents the order from left to right in which the columns appear in the BOM.
- **Group Stocks** - Check box if you would like straight pipes to be grouped with straight pipes, elbows grouped with elbows, etc.
- **Rows Per Column** - The number of rows of text per BOM table before the BOM table repeats. The default is 20 rows.
Spool Dwg...

Menu: None
Toolbar: None
Command Line: None
Navigator: Pipe / Spool Dwg...

Creates spool drawings using an existing drawing template and BOM.

Spools are a type of isometric or orthographic drawing containing dimensions and callouts for fabrication. The following figure is an example of a spool drawing created using the Spool Dwg command.

To create a spool drawing:

**Note**: You cannot run this command while a spool drawing is currently open.

1. Click Spool Dwg...
   - This command requires that at least one BOM definition and at least one drawing template exists. If none exist then the following error message will appear notifying the user of the situation and how to correct it.
2. Currently option one, “Use template and BOMs defined in ShipReport”, is not implemented. Therefore select a drawing template and Bill of Material definition and click Next.

3. In the Create Spool Drawings dialog select the spools for which you want to create spool drawings. The default view is to order by Pipe drawing: each drawing is listed with all the spools it contains. However, it is also possible to sort the spools by assembly or system structures.

4. Select the spool drawings to be generated and click Next.

5. Use the Spool Drawing Settings to customize the look of your spools. You can define several setting configurations for easy recall when needed.
Spool Drawing Variable Descriptions:

- **Spool Style** - The name of the spool settings style.
- **AutoCAD Dimension Style** - The AutoCAD dimension style, within the spool’s template, to use for dimensioning.
- **Dimensions**
  - Baseline dimensioning - Dimensions components included in a straight section from a common starting point.
  - Break dimensions at tees - Dimensions to the branch point of tees, saddle stitches, stubins, etc.
  - Include connections when dimensioning - Dimensions to the center of the connection points.
  - Dimension offset - The base distance the dimension text will be offset from the points being dimensioned.
- **Text**
  - Text Size - The size of the label text.
  - Paper space units - Check this option to set the text height in paper space units.
- **Connection radius** - The size of the connection icon drawn where two pipe entities connect. Enter “0” for a default size or override with another value to customize the display.
- **Spool Orientation Settings** - The Spool Orientation Settings portion of the dialog allows for a common orientation of all spools within their production drawing. This is useful to help reduce errors by keeping the orientation consistent from drawing to drawing for the fabrication shop. The dialog specifies the orientation for the start of the spool, and the direction to orient the first connected branch.
  - **Ship Coordinate System** - The Ship Coordinate System option will leave the spools in the same orientation relative to the world coordinate system as in the 3D model.
- **Spool Coordinate System** - The Spool Coordinate System orients the spool to the world coordinate system. If a spool coordinate system is selected, the start of the spool will have to be picked by the user for each spool that will be made into a spool drawing.

- **Labels** - The text identifying the item numbers of the pipes that are included in the bill of materials.
  - **Leader lines** - Select this option to include leader lines for the straight pipe labels. All other Pipe entities will retain their leader lines for clarity.
  - **Label Size** - The size of the bubble that is drawn around the labels: side length for squares, diameter for circles.
  - **Label Style** - List of available shapes to frame the labels with.

- **Override Flange Size** - Whether or not the flange icon sizes should be overridden.
  - **Flange Width** - The 2-dimensional representation of the flange diameter in paper space units.
  - **Flange Length** - The 2-dimensional representation of the flange thickness in paper space units.

6. Click **OK**.

7. If the Spool Coordinate System option was selected, ShipConstructor will prompt the user to select the start end of the spool for each spool drawing created.

One drawing is created for each spool. Watch closely to see how the views are generated, dimensions are added and the keywords replaced.

**Spool Naming**

Menu: None  
Toolbar: None  
Command Line: None  
Navigator: Pipe / Spool Naming

Opens the **Edit Name Components** dialog that allows the user full control of the automatic, custom naming of spool drawings. For instance, say that your current naming convention requires that a spool name be composed of the job number, pipe drawing name, the unit name or pipe system name, and an automatically incremented spool number. This scheme can easily generate a spool name such as J103-FUEL-U12-001.

1. Select **Spool naming**. The Edit Custom Labels dialog appears.
2. The first task is to add the spool label that holds the definition of the information, and the order in which the information appears. Press New. A default label name is created. Change this to suit your needs. You will also have the option of copying a previous Label definition and then altering its components. Simply select the Style to copy and press Copy. A uniquely indexed label name will be created.

3. Select the label to which additions or changes to the naming convention are to be made.

4. For each new component of the spool name, press Add. For example J103-FUEL-U12-001 contains six components:
   i) **J103**: The job number plus a dash.
   ii) **FUEL**: The name of the drawing from which the spool was created.
   iii) `-`: Another dash as a separator.
   iv) **U12**: The unit name.
   v) `-`: Another dash as a separator.
   vi) **001**: an automatically generated and incremented number.

5. For each of the elements you now must edit/format the added element fields within the Details list box. There are three types of components: Static, AutoNumber, and TableField. Type descriptions are as follows:
   i) **Static** - A non-changing string value, e.g. the job number. Set the string **Value** to whatever is preferred.
      
      | Name  | Value |
      |-------|-------|
      | Type  | Static|
      | String| JOB1001-

   ii) **AutoNumber** - An automatically indexed value. Choosing AutoNumber requires that three sub fields be entered and are shown below:
      
      | Name    | Value |
      |---------|-------|
      | Type    | AutoNumber |
      | Width   | 3     |
      | 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.

   iii) **DB Item** - Used to include the drawing name or a level of the PWBS name, or System name. There exists three options within **DB Item type**:
Field - Three options exist:

- **PWBS - Name** - Select this if it is desired 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 is 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 Yes then the Offset number of characters are removed from the left. If set to No then the Offset number of characters are removed from the right.

  For example, let the panel name be FP2114S, and you would like to extract P21. Set From Start to Yes. Offset to 1 and # Characters to 3.

- **Tree Level** - The context sensitive list that depends on what Field value has been selected. For PWBS - Name, the will contain an entry for each level the PWBS. For System - Name the only valid levels are Project, System, and Branch.

6. Arrange the elements into the order you wish by pressing **Move Up** or **Move Down**. The Elements list shows a summary of the included components and their order of appearance in the spool name.

7. Press **Test Label** for a preview of the spool name. For example, the scheme developed in step 4 would produce a result like:
8. Press **Done** when finished.

**Arrangement Menu**

The arrangement menu is only available when in an arrangement drawing. You can use the standard ShipConstructor functions, such as Attach XREF’s to populate the drawing with structure, outfit and pipe drawings.

**SC Arrangement - Clip current view**

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

Toolbar Button: **None**

Command Line: **SCCLIPVIEW**

Navigator: **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, 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 Arrangement - Remove clip**

Menu: SC Arrangement / Remove clip

Toolbar Button: None
Command Line: **SCCLEARCLIP**
Navigator: None

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

**SC Arrangement - Generate Stock BOM**

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

Toolbar Button: None

Command Line: **SCARRANGEBOM**

Navigator: None

Creates a BOM in the arrangement drawing.

1. **Select Text entity to set the BOM properties from:** will appear on the command line. The command requires you to select a text entity 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.

2. Next, the **Arrangement BOM Options** dialog will appear allowing the user to customize many facets of the **BOM** to be generated.
Dominating the upper left of the dialog box are a number of check boxes which can be used to customize what will be included in the **BOM** (See Above). The **AllPipes** option will automatically include all pipes/fittings, if this option is unselected then the individual fitting/pipe types can be selected to fully customize the **BOM**.

**Include Accessories** - This option determines if the accessories used by connections in this drawing will be included in the **BOM** (and labeled). If this option is checked then all accessories will be included, not just those between pipes included in the **BOM**.

**List Only Visible Pipes** - Only visible pipes will be included in the **BOM**, those outside the clipping planes and the viewport will be ignored.

**BOM to Use** - The pre-defined **BOM** to use, see **Navigator/Pipe-Spool BOM** on how to customize and create BOMs.

**Label Settings**:
- **Label Text Height** - Height of Label Text in drawing units.

In the example above, “BA-” is **Bubble Prefix Text** used to identify the pipe as being from the Ballast system, the three numbers following reference the **BOM** as seen below.

<table>
<thead>
<tr>
<th>QTY</th>
<th>NAME</th>
<th>DESCRIPTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>VAL-BFY-WF-DI-5_CLS150</td>
<td>BUTTERFLY, WAFER TYPE, SS DISC &amp; STEM</td>
</tr>
<tr>
<td>2</td>
<td>3/4-CS</td>
<td>HVY HEX NUT, CS, ASTM A-563 (GR B), ANSI B18.2.2</td>
</tr>
<tr>
<td>3</td>
<td>3/4 x 7-CS</td>
<td>STUD BOLT, CS, ASTM A-307 OR B, ANSI B18.2.1</td>
</tr>
</tbody>
</table>

The label, combined with the BOM, indicate that the valve in question is a Butterfly valve that requires nuts and bolts of the indicated types.

**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. **Note:** If this option is selected then the drawing units in Paper Space will be used even though the labels are inserted into Model Space.

• **Label Only Visible Pipes** - Label only pipes 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.

**Include Free Pipes** - Indicates how free pipes are to be treated when creating the BOM and the labels.

• **Standard** - Free pipes are treated like any other pipe when building the BOM.

• **Ignore All Free Pipes** - Free pipes are ignored regardless of the other settings.

• **Show All Free Pipes** - Free pipes are included regardless of the other settings.

3. Once the settings have been configured to user satisfaction then click **OK** to generate the BOM and the labels (if there are no Remote Connections, see below for details).

4. If the **Include Accessories** option has been selected and there are connections between the pipes in the arrangement drawing and a remote pipe (a pipe in another drawing) then another dialog (**Select Remote Connections to Include**) will appear displaying all the these connections and querying the user to indicate which ones to include. The connections are organized by the remote drawing they are connected too, and then listed in a tree for easy selection.

5. Select the connections you wish to include or leave it blank to ignore all remote connections. Click **OK** to continue on and generate the BOM and Labels.

**SC Arrangement - Generate Spool BOM**

*Menu:* SC Arrangement / Generate Spool BOM

*Toolbar Button:* None
Command Line: **SCSPOOLBOM**

Navigator: **None**

1. **Select Text entity to set the BOM properties from:** will appear on the command line. The command requires you to select a text entity 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.

2. Next the **Spool BOM Settings** dialog will appear allowing the user to customize many facets of the BOM to be generated.

   ![Spool BOM Settings Dialog](image)

   **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 BOM 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 BOM number in the Label (can be left blank).
   
   **Label In PaperSpace** - If this option is checked then the labels are placed in Paper Space.

3. Once the settings have been configured to user satisfaction then click **OK** to generate the BOM and the labels.

   **SPool Name**:
   1. Job1-SeaWater-001
   2. Job1-SeaWater-002
   3. Job1-SeaWater-003
   4. Job1-SeaWater-004
   5. Job1-SeaWater-005
   6. Job1-SeaWater-006
   7. Job1-SeaWater-007
SC Pipe Menu

SC Pipe - Outfit
Menu: SC Pipe / Outfit
Toolbar Button:  
Command Line: CLINSOUTFIT
Navigator: None

Inserts an outfit part into a Pipe drawing.

**Note:** All users who have purchased ShipConstructor - Pipe have the ability to insert outfit parts. However, to create outfit parts and drawings, an Outfitting module license must be purchased.

**To insert an outfit part**
1. Start the insert outfit part command.
2. From the dialog box choose the outfit part to be inserted:
3. Double click an outfit or select one and press **OK**.

**SC Pipe - Spool Manager**

**Menu:** SC Pipe / Spool Manager  

**Toolbar Button:** None  

**Command Line:** SCSPOOLMAN  

**Navigator:** None  

**Definition:** Free - A Pipe/Fitting that is not going to be included in any spool and is to be shipped loose.

This option launches the **Spool Manager**, which is used to create, edit and otherwise modify the spool arrangements for the current Pipe drawing. In order to run this command you must have permission to edit/modify spools (as set in **ShipConstructor Manager**). If the drawing is locked for spooling then the **Spool Manager** will only allow viewing of the spools, but will not allow any changes to be made.

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 three check boxes in the lower right corner control the selection behavior.

- **Zoom** - The drawing window will zoom to center on the spool with the spool filling the screen.
- **Center** - The drawing window will shift to center on the spool (but the zoom level will not change).
- **Highlight** - The pipes in the spool will become highlighted in the drawing window.

A spool grouping consists of a series of interconnected pipes. A grouping ends when an unconnected end or a break is reached. A break can be an **Outfit** Item (as they are not included in spools), a pipe stock that has been marked as **FREE** (free pipes are marked with a cross), or an explicitly defined **break point**. Typically, when breaking a system up the user will define all stocks that are not going to be spooled as **FREE**. Once this is done, manual **break points** are added to split the spools into manageable sizes.

- **Override Color** – By selecting this option the user can choose to override a **Spool Categories** color. For example making all the undefined **Spools** yellow (overriding the system color of the pipes) to make it easier to pick them out while editing.

### Spool Categories

#### Defined

- **Defined (1 Spools)**
- **Job1-SeaWater-004 (4 items)**
A **defined** spool is a run of pipes which have been defined as a spool (see **Undefined - define** on how to define a spool) and been given a name according to the projects **spool naming convention**. In the example above the name of the current job (“Job1”) has been combined with the name of the system (“SeaWater”) and an automatically generated number (“004”) to give the spool a unique designation. Spools need to be defined before spool drawings can be generated.

**Item Menu** - Right clicking on an individual defined spool will bring up the following menu:

- **Edit Properties** - Brings up the Edit Properties dialog allowing you to edit the hierarchy level, galvanization, and paint of the spool. It will also display the spool 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. **Warning**: If the spool naming convention depends on the hierarchy level and you change the hierarchy you will be asked if you want to rename the spool, this will also rename any spool drawings that have previously been generated.

- **Lock** - Locks a defined spool and will prevent any editing of the pipes in the spool that would change the spool (erasing, moving, adding new pipes to unconnected ends, freeing pipes, adding/removing breaks etc.). This will move the selected spool out of the **Defined** category and into the **Locked** one.

- **Clear Spool** - This will clear the spool information from the database and move the spool to the **Undefined** category. **Warning**: this will erase any spool drawings that have been created for this spool.

- **Rename** - This option allows you to rename the spool (including any drawing that have been generated for it). This option opens the Spool Name 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.
**Undefined**

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

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

**Choose Order** - Allows you to specify the order in which to define the spools by picking them in the drawing.
1. When running this command the user will first be asked to pick the properties the spools will be assigned from the Properties Dialog box, see Defined - Edit Properties.
2. Next the spool manager dialog will disappear and the command line will display:
   
   Choose spool:

3. The user will then choose the spools in the order they want them to be named by selecting any pipe in that 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 define an undefined spool by picking it from the list.
1. Choose how the spool will be named.
   - **Choose Auto Name** - Pick the name from a list of available names (see Define - Rename).
   - **Next Auto Name** - Pick the next available name.
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 for editing. A spool that has been locked will not allow any of its pipes to be modified in any way, nor will it allow pipes 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.

Stock Added

Stock Added spools are ones that have had new stock added since the spool was first created.
**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, either with the old name, or using an entirely new name.
  1. **Choose Auto Name** - Pick the name from a list of available names (see **Define - Rename**).
  2. **Next Auto Name** - Pick the next available name.
  3. **<Current Name of Spool>** - Keep the same name as it was previously defined with.

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

**Stock Missing**

Stock Missing spools are ones that have had stock separated by the use of break points. In the screen shot above you can see the **Stock Missing** list contains two entries, both with the same name. In this case the user split off one pipe into its own spool, while the other 5 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, either with the old name, or using an entirely new name (see **Stock Added - Re-Define**). In the case of **Stock Missing** pipes, if you choose to select the old name then the other **Stock Missing** spools with the same name will be changed to **Undefined** spools.

**Multiple Stocks**

Multiple Stock spools contain pipes that used to belong to two or more named spools. This typically happens when removing free pipes or **break points** which causes 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, either with one of the old names, or using an entirely new name (see Stock Added - Re-Define). In the case of Multiple Stocks pipes, if you choose to select an old name then any other Multiple Stocks and Missing Stocks that might contain pipes from the named spool may be modified as well.

Modified

Modified spools are Defined spools that have had pipes in the spool modified by moving, stretching or erasing. 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 Okayed the changes made.

Side Bar

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

- Refresh Info - Causes the Spool Manager to rebuild the list of spools from the drawing. Due to the fact that the spool manager stays up while you work in the drawing, it sometimes gets out of sync, and needs to be updated.
- Spool Break - Add - This command switches the focus to the drawing and prompts the user:
  
  Pick pipe near connection to break at:

  It then inserts a break point at the closest valid connection, and asks the user to select another connection. Hitting ESC will cancel the command.

- Spool Break - Remove - This command switches the focus to the drawing and prompts the user:
  
  Pick pipe near connection to remove break from:

  It then removes the nearest break point, and repeats the command again. Hitting ESC will cancel the command.

- No Spool - Set - This command switches the focus to the drawing and prompts the user to
Select pipes that will not be spooled.

Any pipes selected will be set to **No Spool**. This command will disable the tree until **Refresh Info** is run.

**No Spool - Remove** - This command switches the focus to the drawing and prompts the user to

Select pipes that will be spooled.

It then removes the free setting from this pipe. This command will disable the tree until **Refresh Info** is run.

**Pick - Select Spool** - Switches the focus to the drawing and asks the user to

Choose spool:

The chosen spool is then selected in the tree.

**Pick - Clear Spool** - Switches the focus to the drawing and asks the user to

Choose spool to clear:

The chosen spool will be cleared and moved to the **Undefined** category. **Warning**: This action cannot be Undone and all information on the spool will be lost (database information and a drawing if one has been created).

---

**SC Pipe - Break Pipe**

**Menu:** SC Pipe / Break Pipe

**Toolbar Button:** None

**Command Line:** CLBREAKPIPE

**Navigator:** None

Breaks straight and bent pipes at a selected point creating two pipes of the same stock. Pipes are automatically reconnected at the break point if a valid connection exists.

Before After

---

**To Break a Pipe**

1. Start the **Break Pipe** command.
2. Select the straight or bent pipe to break.
3. Select the point at which to break the pipe.
   - If only one connection is possible between the newly created ends of the pipes, they will be joined automatically by the valid connection.
   - If more than one valid connection exists between the newly created ends of the pipes, one must be selected from the following dialog:
Double click a connection or select one and press OK.

**SC Pipe - Connect Pipe**

Menu: SC Pipe / Connect Pipe

Toolbar Button: None

Command Line: CLCONNECTPIPE

Navigator: None

Connects two pipe entities with a valid connection, if it exists. For example:

<table>
<thead>
<tr>
<th>First End</th>
<th>Second End</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Note:** Use this command to connect to free pipe ends in remote drawings, as you cannot connect to free pipe ends in remote drawings using the standard method.

**Note:** This command is run automatically if you move one end of a pipe to the end of another pipe, either with grips, or with the move command.

**To Connect Two Pipes**

1. Start the **Connect Pipe** command.
2. Select the first pipe end to be joined. The first pipe selected remains stationary when connecting.
Specify First End.

3. Select the second pipe end to be joined. The second pipe will be moved/transformed, if allowed, to join the other pipe with a valid connection. If more than one valid connection exists between two entities’ End Treatments then the desired connection must be chosen from the following dialog box:

**SC Pipe - Disconnect Pipes**
Menu: **SC Pipe / Disconnect Pipe**
Toolbar Button: **None**
Command Line: **CLDISCONNECTPIPE**
Navigator: **None**

Disconnects two or more pipe entities.

**To Disconnect two Pipes**
1. Start the **Disconnect Pipes** command.
2. Select the entities with the connections to be removed. Multiple connections can be deleted during this command, so only choose the pipes that share the connections that are to be deleted. The number of selected pipes is echoed on the command line.

   Select objects: 1 found, 3 total

3. Press **Enter**. The number of connections that were broken is printed on the command line.

**SC Pipe - Disconnect from Remote Pipe**
Menu: **SC Pipe / Disconnect Pipe from Remote Pipe**
Toolbar Button: **None**
Command Line: **SCDISREMOTE**
Navigator: **None**

Disconnects a pipe in the current drawing from a pipe in a remote drawing (XREF’d drawing). Only one pipe can be disconnected each time the command is run.

**To Disconnect from remote Pipe**
1. Start the **Disconnect from Remote Pipe** command.
2. To disconnect the pipe:
Select a pipe to disconnect from remote drawings.

3. After selecting the pipe the command is automatically completed. The name of the XREF’d drawing containing the remote pipe is echoed on the command line.

**SC Pipe - Insert Pipe**

**Menu:** SC Pipe / Insert Pipe

**Toolbar Button:** None

**Command Line:** CLINSPARTINTOEXIST

**Navigator:** None

Inserts and connects, a Tee, Cross, or Valve into a straight pipe segment.

**Note:** At least one of the desired Tee, Cross, or Valve must be capable of connecting to the selected straight pipe since the inserted entity automatically connects itself to the straight pipe segment.

**To Insert a Tee, Cross, or Valve**

1. Start the **Insert Pipe** command.

2. Select the straight pipe that the Tee, Cross, or Valve will be inserted into.

3. Select the insertion point of the entity.

4. Choose the entity type: Tee, Cross, or Valve.

5. From the **Specs and Stocks** dialog, select the entity to be inserted.

6. With the mouse or one of the optional directions, orient the Tee, Cross, or Valve with respect to the World UCS.

**Outlet orientation or [Port/Stbd/Up/Down]:**

**Note:** Only valid directions are shown as optional constraints.

**SC Pipe - Combine to Bent**

**Menu:** SC Pipe / Combine to Bent

**Toolbar Button:** None

**Command Line:** CLAUTOMELD
Combines straight pipes, elbows and bent pipes into one bent pipe.

**To Combine Entities to a Bent Pipe**

1. Either pre-select the pipes to be combined, or start the *Combine to Bent* command, and then select the pipes.
   - If multiple straight stocks are encountered during the conversion, then the user is prompted to select the stock of the final bent pipe.

   **Note**: Bent pipes are allowed to exceed the maximum length specified by the stock. The user will be notified if the combined length exceeds the maximum length of the converted pipe. Use the Shortcut (right click) menu to split the pipe to its maximum length.

2. Press **Enter** to finish the command if selecting stocks after starting the *Combine to Bent* command.

**SC Pipe - Find Replace Stock**

- **Menu**: SC Pipe / Find Replace Stock
- **Toolbar Button**: None
- **Command Line**: CLFINDREPLACEPIPE
- **Navigator**: None

Replaces selected Straight, Bent or Flange stocks with those of the same type and in the same spec. The replacement stocks must have identical End Treatments and replacement Flanges must also have the same length to be selectable.

**Note**: Multiple, differing stocks can be selected for replacement when running this command, however only one stock at a time is replaced.

**Warning**: The replacement pipe’s length may exceed its maximum length as specified in the Catalog Stock Editor.

**To Replace Stocks**

1. Start the Find Replace Stocks command.
2. Select the Straight pipes, Bent pipes or Flange stocks for which the stock is to be changed.
3. The *Find And Replace Pipe Stock* dialog lists all the stocks selected, filtering out stocks other than Straight pipes, Bent pipes, and Flanges, if they were selected.
4. From the **Find All Stocks** list, select the stock to be replaced.

The stocks eligible for replacement are displayed in the **And Replace With Stock From Spec** list.

5. Select a replacement stock from the **And Replace With Stock From Spec** list and click **Replace**.

6. The number of stocks replaced is echoed on the command line.

**SC Pipe - Find Stock**

Menu: **SC Pipe / Find Replace Stock**

Toolbar Button: **None**

Command Line: **SCFINDPIPE**

Navigator: **None**

Scans the drawing and compiles a list of all the stocks in the current drawing. From the stock can be easily located within the drawing by using the stock selector buttons combined with the viewing options.
SC Pipe - Change Connections

Menu: SC Pipe / Change Connections

Toolbar Button: None

Command Line: CLCHANGEPIPECONS

Navigator: None

Replaces connections used between pipe entities. To be eligible for replacement, the connections must have the identical End Treatments and gap assigned to them. This command is particularly useful for swapping connections that are similar except for the assigned connection accessories.

1. Start the Change Connections command.
2. Select the entities that include the connections that are to be replaced.
3. The Change Connections dialog lists the connections used between the selected pipes. The information displayed includes the connection name, the connection’s End Treatments, and the number of times the connection was used in the selection set of pipes.

Select the Connection to be replaced from the Selected Connections list. Connections with identical End Treatments and gaps are displayed in the Available Connections list.
4. Select the replacement connection from the **Available Connections** list.
5. Click **OK** or press **Enter**.
6. The number of connections updated is echoed on the command line.

### SC Pipe - Pipe Options

**Menu:** SC Pipe / Pipe Options  
**Toolbar Button:**  
**Command Line:** CLPIPEOPTIONS  
**Navigator:** None

Allows the user to customize the pipe drawing options and to set various Pipe default options.

#### Display Options

The display options are specific to each drawing and as such the information is stored with the drawing. You can change these options at any time and the pipes will be redrawn immediately in the new display style.

**Pipe Drawing Style** - These modes are designed to augment AutoCAD’s shading modes. The combination of modes is shown table form below.

<table>
<thead>
<tr>
<th>Pipe Display Mode</th>
<th>AutoCAD - Shade Mode</th>
<th>Single Line</th>
<th>Double Line</th>
<th>Dbl Ln + Hide</th>
<th>3D Mesh</th>
</tr>
</thead>
<tbody>
<tr>
<td>2D Wireframe Display</td>
<td>Line mode</td>
<td>Simplified Outline</td>
<td>Simplified Outline - Hidden lines removed</td>
<td>3D Mesh</td>
<td></td>
</tr>
<tr>
<td>Rotate</td>
<td>Line mode</td>
<td>3D Wireframe</td>
<td>Simplified Outline - Hidden lines removed</td>
<td>3D Mesh</td>
<td></td>
</tr>
<tr>
<td>Plot</td>
<td>Line mode</td>
<td>Simplified Outline - Hidden lines removed</td>
<td>Simplified Outline - Hidden lines removed</td>
<td>3D Mesh</td>
<td></td>
</tr>
<tr>
<td>3D Wireframe Display</td>
<td>3D Wireframe</td>
<td>Simplified Outline - Hidden lines removed</td>
<td>3D Mesh</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Pipe Status (Anchor, No Spool) - Use this option to display which pipes are anchored or not to be spooled by displaying the **Anchor** symbol, the **No Spool** symbol, or both. Set the symbol size and color to customize the display.

- **Anchors** - Anchoring a pipe or outfit locks it in place so movement is not allowed.
- **No Spool** - Setting a pipe to **No Spool** means it will not be included in any spool.

**Note:** To anchor a pipe, select the pipes or fittings to anchor. Right-click the mouse and select Anchor. If Pipe Anchors are turned on you will immediately see the Pipe Anchor icon. To No Spool a pipe use the same method as Anchor only select **No Spool**, or use the spool manager.

Spool Break Indicators - Spool breaks indicate at which positions a Pipe system is divided into individual pre-fabricated spools. ShipConstructor Pipe can automatically generate spool drawings for each individual spool. You can set the spool break indicator size and spool break indicator color.

Center Line - Use this option when in the various 3D modes. You can select to display the centerlines in the same color as the pipe color or any other color.

Pipe Connections - Pipe connections are logical connections between pipes. Two pipes are “connected” if they have a common end point and Manager knows that they are logically connected. Connected pipes make ShipConstructor -Pipe a very powerful system. For example, you can move one pipe and the logical connections are checked to see if the move is allowed and how it might have to be restricted. Any connected pipe will automatically adjust to any legal move and Manager is automatically updated with all relevant new information. You can select the color for the unconnected and connected state separately.

Use Database Colors - Overrides the color selection for the Pipe Anchors, Spool Break Indicators, Center Line, and Pipe Connections with the default colors set in the database.

Auto UCS - Temporarily moves and reorients the UCS when connecting to a Pipe end point so that the axis of the end point lies in the X-Y plane of the temporary UCS.
**Default Options**

**Spools**

- **Naming Convention** - Selects the spool naming convention as defined through the ShipConstructor / Pipe tab’s Spool Naming button. From the drop list, select the desired spool naming convention. This takes effect immediately after closing the dialog.

- **Lock Spools** - Locks out any editing of pipe entities such as adding, deleting, or moving pipes. This is a useful option when the drawing has already been spooled and any changes made would invalidate previous work. This serves mainly as a warning to those opening the drawing that any changes made could have significant ramifications.

When attempting to modify a locked drawing the user will be notified through the following dialog that the drawing has been locked for spooling.

**Default System**… - Brings up the Set System dialog, which allows the user to select the default system to be used for new Pipe entities.

**Note:** For full editing of Systems and Branches, use the Pipe menu command **Pipe / System / Edit System** or type SCEDITSYSTEM at the command line.

Every pipe must belong to a system. The type of material the pipe will carry usually specifies a system. Any system can have branches. You can assign the pipes to be part of the system or a branch. The names for the system and branches can be numbers, letters, or both.

**To Set the Default System**

1. Select the desired default system from the system tree.
2. Click **OK**.
SC Pipe - Spec Defaults...
Menu: SC Pipe / Spec Defaults...

Toolbar Button: SPEC
Command Line: SCSETPIPESTOCK
Navigator: None

Allows the user to set the default spec and stock for each type of Pipe to be used in a Pipe drawing. Specs are a logical grouping of stocks.

The dialog should be familiar to you as it is the same dialog you use to select the stocks for pipes; however, in this instance the Type drop list is not disabled. By selecting a Type and then a stock from the Stocks list you are setting the default stock for that type. The next time you choose to lay a straight pipe, elbow, etc., the default stock is that chosen from the list.

You still have the ability to override and thus change the default stock at the time of placing a Pipe entity, but only for that entity type.

To Set the Default for a type of Pipe
1. Start the Spec Defaults command.
2. Select the Spec from the Spec drop list.
3. Select the Type by clicking on the Drop List indicator, and then select the type of Pipe.
4. From the list of Stocks select the one to be the default.
5. Press OK.

SC Pipe - Penetration List
Menu: SC Pipe / Penetration List

Toolbar Button: None
Command Line: SCPIPEPENLIST
Navigator: None
This command accesses the global Pipe penetrations list where you can Apply or Remove penetration objects to locations where pipes interfere with parts of the ship structure. The list reflects all the pipe penetration objects that have been added to the project regardless of whether any of the pipes exist in your current drawing.

In ShipConstructor - Pipe, penetration objects are cutouts that allow pipes to pass through a part of the structure. They are associated with a particular stock item, and their name, size, and shape is defined in the Catalog Editor. Since a particular pipe stock item may be used in many instances, many different penetration objects will likely be defined for each stock.

The **Penetration List** is a result of several previous commands.

Using ShipConstructor’s Interference checking, you first find the locations where components of the ship model interfere with each other; in the case of Pipe, the locations where Pipe entities penetrate the ship structure. Rather than have the user manually find the location and then create the necessary clearance holes, ShipConstructor automatically places them for you. Before these clearance holes are created, however, the penetration object must be added to the associated interference and then applied. Applying the penetrations is the step that actually creates the clearance hole.

**To Add Items to the Penetration List**

1. Go to the **Navigator / Interference** tab and select the **New** button. In the **New Interference Drawing** dialog, select the parts that need to be included for interference checking.
2. Press **OK**. The **New Drawing** dialog will appear. Give the drawing a name. Press **OK**.
3. Run the **ShipConstructor** menu command **Check Interferences**. This brings up the Interference List dialog.
4. In the **Interference List** dialog click on the **Run Check** button. This performs the actual interference check between the included parts.

5. The resulting list represents all the interference objects that were detected. Interferences may be found between any of the parts that were included for checking, but since you are interested in Pipe penetrations you should filter the list to include only the pipe - structure interferences. In the **Show Int. Types** box, check **StraightPipe-Struct** and **OtherPipe-Struct**.

6. Select an interference item from the list and press **Add Pipe Pen**. The Penetration list is updated to reflect the new addition. If multiple penetration objects exist for the particular pipe stock, then choose the correct one by clicking the **Cutout** entry. A drop list will appear, from which you can select the correct penetration.

7. Click **Save** when you are done to update the database.

**To Remove a Penetration**
1. In the Pipe Penetration List dialog box select the penetration to be removed from the interference.
2. Click **Remove**.
3. Press **Save** when done.

**To Create the Cutouts in Waiting**
1. Open the Pipe Penetration List dialog.
2. Click the **Apply Pens** button. The database is updated accordingly and a log file is generated notifying the user of the changes.

### Pipe - Catalog Editor

**Menu:** SC Pipe / Catalog Editor  
**Toolbar Button:**  
**Command Line:** SCCATALOGEDITOR  
**Navigator:** None  

Starts the Catalog Editor. See the chapter entitled Catalog Editor for a description and instructions in its use.

### Pipes Sub-Menu

| Sub-Menu | Straight Pipe  
|-----------|----------------|
|           | Elbow          
|           | Tee            
|           | Flange/Connector  
|           | Reducer        
|           | Caps           
|           | Cross Pipe     
|           | Wye Pipe       
|           | Valve          
|           | Bent Pipe      
|           | Add Saddle     
|           | Copy Saddle    
|           | Remove Saddle  

### Pipes - Straight Pipe

**Menu:** SC Pipe / Pipes / Straight Pipe  
**Toolbar Button:**  
**Command Line:** MAKEPIPE  
**Navigator:** None  

Creates a straight pipe segment specified by two points. Straight pipes by definition must have End Treatments of the same size.

The following figure shows two flanged, straight pipes, joined with a valid connection.
**Placing a Straight Pipe**

Creation of a straight pipe may be broken down into three different categories:

1. **Disconnected** - The pipe’s first point is not connected to any other Pipe entities.
2. **Connected** - The pipe’s first point is connected via a predefined connection to another Pipe entity.
3. **Connected via Saddle Stitch** - The pipe’s first point will be connected to another entity through a fabricated connection.

**To Create a Disconnected Straight Pipe:**

1. Start the Straight Pipe function.
2. The user is shown the name, spec, standards information, and End Treatment types of the current pipe. The user is then prompted to provide one of four options:

   PIP 4:120, DemoSpec, ASME-B31-120, PL, PL
   Specify First Point or [eXchange ends/Stock/sAddle stitch]:

   - **Specify First Point** - Select the start point of the pipe anywhere on the drawing except for on another Pipe entity’s end point.
   - **Exchange ends** - Swaps the ends of the pipe. Use this option when the selected stock has two dissimilar ends and the starting end, End Treatment 1, is not the correct end.
   - **Stock** - Allows the user to change the stock of the current pipe type. The following dialogue appears prompting the user to select a type from the current or alternate spec. Select **Default System** to change the system of the pipes.
   - **Saddle Stitch** - See To Connect a Straight Pipe via Saddle Stitch.

3. The user can immediately constrain the pipes cardinal direction or, by default, specify the length and direction with the mouse. The directions correspond to the world UCS. The direction may also be specified using the regular AutoCAD options of entering an angle using @ and/or <. The @ is used by entering relative coordinates: @ 100, 50. This would be an angle defined by 100 units in the X direction and 50 units in the Y direction. The < is used by entering the angle in degrees with respect to the standard X-Y plane CCW direction – unless AutoCAD is set to something else. Or combine the two by entering a length and a direction. Example: @1000<45. (This also works for 3D as well. @100,100,100 or <45<45 or @100<45<60.)

   Specify direction or [Fwd/Aft/Port/Stbd/Up/Down]:

4. With the mouse specify length and direction or constrain the direction first by entering one of the directional options.
5. If specifying a direction the user will be prompted for the **Second point** or, optionally, the **Length**.

   Specify second point or [Length]:

   - **Second point** - Finish the command by entering a second point with the mouse.
   - **Length** - With the keyboard enter the exact length of the pipe.
6. Specify the end point or choose the length option. Straight pipe stocks have a minimum and maximum length associated with them. Should the length that you provide exceed the defined maximum length, the user will be prompted with:

Choosing **Yes** will allow the user to lay as many pipes as necessary to cover the specified length. Choosing **No** will default the pipe length to the maximum length and terminate the command.

**To Connect a Straight Pipe to Another Pipe Entity:**

**Note:** A logical connection must exist between the End Treatments of the connecting stocks for completion of this command.

The function checks if the current default pipe can form a legal connection with the connecting pipe. The following cases may occur:

- **Connection is valid** - The pipe is drawn and you can continue as usual.
- **Connection is not valid, but legal pipes can be found within the same spec** - A dialog offers a list of choice to choose from.
- **Connection is not valid, and no legal pipe can be found within the same spec** - An error message is displayed.

1. Start the Straight Pipe function.
2. The user is shown the name, spec, standards information, and End Treatment types of the current pipe. The user is then prompted to provide one of four options:

   **PIP 4:120, DemoSpec, ASME-B31-120, PL, PL**
   
   Specify First Point or [eXchange ends/Stock/sAddle stitch]:

   or, if the straight pipe command follows the placement of another pipe, an additional option is presented, **Last end**:

   **PIP 4:120, DemoSpec, ASME-B31-120, PL, PL**
   
   Specify First Point or [eXchange ends/Stock/sAddle stitch/Last end]:

   - **Exchange ends** - Swaps the ends of the pipe. Use this option when the selected stock has two dissimilar ends and the starting end, End Treatment 1, is not the one you want.
   - **Stock** - Allows the user to change the stock of the current pipe type. The following dialogue appears prompting the user to select a type from the current or alternate spec. Select **Default System** to change the system of the pipes.
Double click a stock or select the stock and click OK.

- **Last End** - The first point of the straight pipe automatically snaps to the end of the last laid straight pipe. This option only appears if the command follows the placement of a previously connected pipe.

- **Saddle Stitch** - See To Connect a Straight Pipe via Saddle Stitch.

3. Enter “L” for **Last end** or select the free end of a Pipe entity. The direction of the new straight pipe is determined from selected end point.

**Tip:** Select the ShipConstructor OSNAP mode “Snap to Free pipe ends” to only select unconnected pipe ends.

- The program verifies that a connection between the existing stock’s End Treatment, and the new straight pipes End Treatment, exists before continuing. If no connection exists between the two entities then a log file will be generated, notifying the user that a connection needs to be created between the two listed End Treatments. The user will also have the choice to select a straight pipe from a list of pipes that will connect.

4. Complete the command by entering the **Second point** with the mouse or by specifying the **length** with the keyboard.

```
Specify second point or [Length]:
```

- **Second point** - Enter the second point with the mouse.
- **Length** - With the keyboard, enter the exact length of the pipe.

5. Specify the end point or choose the length option. Straight pipe stocks have a minimum and maximum length associated with them. Should the length that you provide exceed the defined maximum length, the user will be prompted with:

![AutoCAD dialog box](image)

Choosing **Yes** will allow the user to lay as many pipes as necessary to cover the specified length. Choosing **No** will default the pipe length to the maximum length and terminate the command.

**To Connect a Straight pipe via Saddle Stitch**

Saddle stitches are fabricated connections to existing pipe runs. They may be added to straight pipes, bent pipes, and elbows. Any pipe type may connect to a saddle stitch as long as a connection has been created between the saddle stitch End Treatment and the current stocks End Treatment.

What you are really doing in the program when you add a saddle stitch is to create another endpoint on a pipe segment, to which other pipe entities may attach. For example, a straight pipe normally has only two ends to which a pipe may be connected. By adding a saddle stitch connection, you have the ability to tap into that pipe at any intermediate location. The following figure shows a plain-ended pipe connected with a saddle stitch to produce an angled branch of pipe.
The following figure shows two saddle stitches added to a section of straight pipe. One, shown on the left, has another smaller section of pipe attached, whereas the saddle stitch on the right exists only as an endpoint. Later, another pipe/fitting may attach to it as long as a Connection is defined that includes the Saddle Stitch End Treatment and the connecting pipe/fitting's End Treatment.

**Note:** Since a saddle stitch is really an End Treatment, for a stock to connect with a saddle stitch, there needs to be a connection defined that includes: a) The saddle stitch End Treatment. b) The End Treatment of the connecting stock.

1. Start the Straight Pipe function.
2. Enter “a” or “A” for the saddle stitch option.
3. Select the existing pipe entity to which the new straight pipe will attach.
   
   Select a Pipe to add Saddle To:

4. Select the saddle stitch **EndTreatment Type** and the proper size End Treatment for the connecting pipe.
5. Next, the saddle point origin must be determined. Currently ShipConstructor - Pipe does not support offset Saddle Stitches (the axis of the connecting pipe will always intersect the centerline of the pipe it is connecting to).

a) **Change end type** - Allows the user to reselect the saddle stitch End Treatment.

b) **Specify distance from end** - Select this option if you want to place the saddle stitch a known distance from an end.
   i) Select a Pipe end point from which the distance is to be measured.
   
   Please Select Stock End:
   
   ii) Enter a distance via keyboard or mouse.
   
   Specify Distance:
   
   iii) Valid directions are shown as keywords. For example, if the pipes are running fore and aft these options will not be displayed. One can immediately constrain the direction or click to define the direction.
   
   Specify direction or [Port/Stbd/Up/Down/PEn/PErp/Angle]:
   
   iv) Default attachment origin is the pipe center. If another distance is required, it can be entered in the units of the project.
   
   Specify direction from centerline in mm. 0.0000 or [on-Surface]:
   
   c) **Specify first point** - The default option. The first point can be selected with a mouse click.
   
   i) Valid directions are shown as keywords. For example if the pipes are running fore and aft these options will not be displayed. One can immediately constrain the direction or click to define the direction.
   
   Specify direction or [Port/Stbd/Up/Down/PEn/PErp/Angle]:
   
   ii) Default attachment origin is the pipe center. If another distance is required, it can be entered in the units of the project.
   
   Specify direction from centerline in mm. 0.0000 or [on-Surface]:

**Note**: The program verifies that the two stocks can connect with a valid connection. If no connection is defined between the saddle stitch End Treatment and one of the new straight pipes ends, a log file will be generated notifying the user of the situation and then give the user the option of selecting from a list of straight pipes that do connect to the chosen saddle stitch.
6. Specify the end point or choose the length option. Straight pipe stocks have a minimum and maximum length associated with them. Should the length that you provide exceed the defined maximum length, the user will be prompted with:

Choosing **Yes** will allow the user to lay as many pipes as necessary to cover the specified length. Choosing **No** will default the pipe length to the maximum length and terminate the command.

---

**Pipes - Elbow**

**Menu:** SC Pipe / Pipes / Elbow

**Toolbar Button:**

**Command Line:** ARCPipe

**Navigator:** None

Places an elbow pipe segment.

**Placing an Elbow**

The following are the steps to place an elbow:

The placement of an elbow may be broken down into three different categories:

1. **Disconnected** - The elbow’s first point is not connected to any other Pipe entities.
2. **Connected** - The elbow’s first point is connected via a predefined connection to another Pipe entity.
3. **Connected via Saddle Stitch** - The elbow’s first point will be connected to another entity through a fabricated connection.

**To Place a Disconnected Elbow:**

1. Start the Elbow function.
2. The user is shown the name, spec, standards information, and End Treatment types of the current elbow. The user is then prompted to provide one of five options:

   **Command:** `\ARCPipe`  
   `90LLR-4-571, Demo Spec, ASME-B31-160, BW, BW`  
   `Specify first point or [eXchange ends/Stock/sAddle stitch/Intersection of]:`

   - **Specify first point** - Select the start point of the elbow anywhere on the drawing except for on another Pipe entity’s end point.
   - **Exchange ends** - Swaps the ends of the elbow. Use this option when the selected stock has two dissimilar ends and the starting end, End Treatment 1, is not the correct end.
   - **Stock** - Allows the user to change the stock of the current elbow type. The following dialog appears prompting the user to select a type from the current or alternate spec. Select **Default System** to change the system of the pipes.
3. The user can immediately constrain the elbow’s cardinal direction by entering one of the corresponding “key” letter(s) from the list of options. Or, by default, specify the direction with the mouse. The directions correspond to the world UCS.

Outlet orientation or [Fwd/Aft/Up/Down/ANGle]:

*Note: Angle* is an option to reduce the angle of the pipe. (For example: From 90 degrees to 60 degrees. Always less than the original selected stock.)

### To Connect an Elbow to Another Pipe Entity:

*Note:* A logical connection must exist between the End Treatments of the connecting stocks for completion of this command.

The function checks if the current default elbow can form a legal connection with the selected pipe entity. The following cases may occur:

- **Connection is valid** - The elbow is drawn and you can continue as usual.
- **Connection is not valid** - A dialog offers a list of choices to choose from.
- **Connection is not valid, and no legal elbow can be found within the same spec** - An error message is displayed.

1. Start the Elbow function.
2. The user is shown the name, spec, standards information, and End Treatment types of the current elbow. The user is then prompted to provide one of five options:

   Specify first point or [eXchange ends/Stock/sAddle stitch/Intersection of/Last end]:

   **Or** if the **Elbow** command follows the placement of another connected pipe entity, then an additional option is presented: **Last end**.

   Specify First Point or [eXchange ends/Stock/sAddle stitch/Intersection of/Last end]:

- **Exchange ends** - Swaps the ends of the pipe. Use this option when the selected stock has two dissimilar ends and the starting end, End Treatment 1, is not the one you want.
- **Stock** - Allows the user to change the stock of the current pipe type. The following dialog appears prompting the user to select a type from the current or alternate spec. Select **Default System** to change the system of the pipes.

Double click, or select the stock, and click **OK**.

- **Saddle stitch** - See **To Connect an Elbow via Saddle Stitch**.

- **Intersection of** - Allows the user to place an elbow between two pipes only if they are close enough, or overlapping, so that the elbow’s ends can make contact with both pipes. Must still use **SC Pipe / Connect** to actually logically connect the pipes after the elbow has been inserted.

- **Last end** - The first point of the elbow automatically snaps to the end of the last laid pipe. This option only appears if the command follows the placement of a previously connected pipe.

3. Select the free end of a Pipe entity OR enter “AN” for Angle OR enter “L” for Last end. The new elbow will automatically align to the selected Pipe entity.

**Tip:** Select the ShipConstructor OSNAP mode “Snap to Free pipe ends” to only select unconnected pipe ends.

4. Complete the command by entering the Outlet orientation by specifying the direction with the mouse **OR** by entering the corresponding “key” letter(s) of one of the options. Selecting the Angle option is **NOT** an orientation. It is a command to change the angle of formation of the elbow. (For example: From 90 degrees to 60 degrees. Always less than the original selected stock.)

```
90LLR-2+1/2, Demo Spec, ASME-B31-120, PL, PL
Conn = PL
Adding a 90LLR-2+1/2 to a PIP 2-1/2:40
Outlet orientation or [Fwd/Aft/Up/Down/ANgle]:
```

The type of elbow is displayed along with the type of connection between the two entities followed by the actual choices for the Outlet orientation.

**To Connect an Elbow via Saddle Stitch**

Saddle stitches are fabricated connections to existing pipe runs. They may be added to straight pipes, bent pipes, and elbows. Any pipe type may connect to a saddle stitch as long as a connection has been created between the saddle stitch End Treatment and the current stock’s End Treatment.
What you are really doing in the program when you add a saddle stitch is to create another endpoint on a pipe segment, to which other pipe entities may attach. For example, a straight pipe normally has only two ends to which a pipe may be connected. By adding a saddle stitch connection, you have the ability to tap into that pipe at any intermediate location.

**Note:** Since a saddle stitch is really an End Treatment, for a stock to connect with a saddle stitch, there needs to be a connection defined that includes: a) The saddle stitch End Treatment. b) The End Treatment of the connecting stock.

1. Start the Elbow function.
2. Enter “a” or “A” for the saddle stitch option.
3. Select the existing pipe entity to which the new elbow will attach. 
   
   ![](selectpipe.png)

4. If a saddle stitch is not already defined then a dialog will appear. Select the saddle stitch **EndTreatment Type**, the **Nominal Size** end-treatment for the connecting pipe, and **Description** for the type of saddle stitch and click **OK**.

   ![](select_saddle_stitch.png)

5. The next prompt requests the user to select the location (insertion point) for the saddle stitch.
   
   ![](select_saddle_location.png)

**Tip:** If the osnap options are unable to provide the desired insertion point then one option is to create a temporary reference line. This line should be perpendicular to the pipe entity’s centerline - create the line at an end point, or the mid point, and then use the AutoCAD Offset command to make a copy of the line into the correct place. See the figure below.

![](reference_line.png)

Reference line positioned by using the Offset command.
6. The state of the Current Alignment, followed by the saddle direction options, is displayed. If the state is “free” then the elbow’s orientation has no limitations.

```
Current Alignment: Free
Pick Saddle Direction [pErp/taN/Angle/Fwd/AFt/Port/Stbd/Up/Down]:
```

7. The next prompt allows a distance from the center of the pipe entity to be entered for an offset from the saddle stitch point. If no offset is desired then enter “0”.

```
Enter distance from Centerline [Enter for surface]:
```

If no connection is defined between the elbow’s End Treatment & size, and the selected saddle stitch, then a message window will state this fact.

```
AutoCAD

No stocks connect to a
8 in SAD regular
Please create a connection first:

CK
```

It may also offer the option to choose an elbow to use if there exists an elbow that can connect to the saddle stitch. Creating and defining of connections is done in the Catalog Stock Editor by following the steps below.

a) Ensure that the saddle stitch End Treatment exists for the size of the elbow, by going to the End Treatment tab and selecting the box for the saddle stitch type in Show: End Treatments.

   i) To check which End Treatment type is a saddle - select Edit Types and look for the one that has Is Saddle property set to Yes. If none are set to Yes then selecting New will allow you to define one by setting its Is Saddle to Yes.

```
EndTreatment Types

<table>
<thead>
<tr>
<th>Name</th>
<th>LineId</th>
<th>IsSaddle</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bv</td>
<td>BvIcon</td>
<td>No</td>
</tr>
<tr>
<td>Fl</td>
<td>FlIcon</td>
<td>No</td>
</tr>
<tr>
<td>Pl</td>
<td>PlIcon</td>
<td>No</td>
</tr>
<tr>
<td>SAD</td>
<td>SADIcon</td>
<td>No</td>
</tr>
<tr>
<td>SCF</td>
<td>SCIcon</td>
<td>No</td>
</tr>
<tr>
<td>SCM</td>
<td>SCIcon</td>
<td>No</td>
</tr>
<tr>
<td>SWF</td>
<td>SWIcon</td>
<td>No</td>
</tr>
<tr>
<td>SWM</td>
<td>SWIcon</td>
<td>No</td>
</tr>
</tbody>
</table>

   New

   Delete

   Done

   Cancel
```

Note: If defining a saddle stitch End Treatment was necessary then the new saddle stitch type will appear in the Show: End Treatments window after Done has been selected.

   ii) Checking the box for the saddle stitch type will bring up a list of all the sizes that have this End Treatment.

a) If the size of the elbow is not here then select it from Available Norm. Sizes window and then select New End Treatments to define one in the End Treatments window.

```
EndTreatment Types

<table>
<thead>
<tr>
<th>Name</th>
<th>Id</th>
<th>wire</th>
<th>Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>0</td>
<td>0</td>
<td>SAD</td>
</tr>
<tr>
<td>12</td>
<td>0</td>
<td>0</td>
<td>SAD</td>
</tr>
<tr>
<td>1.5</td>
<td>0</td>
<td>0</td>
<td>SAD</td>
</tr>
<tr>
<td>4</td>
<td>0</td>
<td>0</td>
<td>SAD</td>
</tr>
<tr>
<td>6</td>
<td>0</td>
<td>0</td>
<td>SAD</td>
</tr>
</tbody>
</table>

   New

   Delete

   Done
```

Use the drop down list to set the type to a saddle stitch type.
b) Go to the \textbf{Connections} tab and select the type of End Treatment the elbow is using in the \textbf{Show: Connections} window. This will clear the list in the \textbf{Connections} window and bring up a list of defined pairs of End Treatments and sizes, such that at least one of the End Treatments matches the type selected.

i) The End Treatment and size on the elbow must be paired up with a saddle stitch type End Treatment. Select \textbf{Edit} to get the \textbf{New} button to become active. Click \textbf{New} to create a new connection.

ii) Now pair up the End Treatment defined as a saddle stitch and a size, with the End Treatment and size that the elbow uses. This is accomplished by selecting the End Treatment in the drop down list.

\begin{center}
\begin{tabular}{|c|c|c|c|}
\hline
Name & Treatment 1 & Treatment 2 & LookMode Icon \\
\hline
BW & BW 25 m & BW 25 m & BW Icon \\
BW & BW 25 m & PL 25 m & BW Icon \\
BW & BW 4 in & BW 4 in & BW Icon \\
BW & BW 4 in & PL 4 in & BW Icon \\
BW & BW 14 in & BW 14 in & BW Icon \\
BW & BW 10 in & BW 10 in & BW Icon \\
SAW BW & SAD 25 m & BW 25 m & SAD Icon \\
SAW BW & SAD 8 in & SAD 8 in & SAD Icon \\
SAW BW & SAD 10 in & SAD 10 in & SAD Icon \\
SAW BW & SAD 5 in & SAD 5 in & SAD Icon \\
\hline
\end{tabular}
\end{center}

\textbf{Tip}: Matching the size of the saddle stitch and the elbow will prove less confusing later.

iii) \textbf{Save Changes} and then choose \textbf{Done}.

8. The following information is then displayed. Complete the command by entering the Outlet orientation by specifying the direction with the mouse OR by entering the corresponding “key” letter of one of the options.

\begin{center}
90LSR-2+1/2-395, DemoSpec, ASME-B31-160, BW, BW
Conn = SAD-BW
Adding a 90LSR-2+1/2-395 to a PIP 4:160
Outlet orientation or [Pw/Aft/Up/Down]:
\end{center}

\textbf{Pipes - Tee}

Menu: \textbf{SC Pipe / Pipes / Tee}

Toolbar Button: \textbf{TEEPIPE}

Command Line: \textbf{TEEPIPE}

Navigator: \textbf{None}

Places a tee pipe segment.

\textbf{Placing a Tee}

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

1. \textbf{Disconnected} - The tee’s first point is not connected to any other Pipe entities.
2. \textbf{Connected} - The tee’s first point is connected via a predefined connection to another Pipe entity.
3. **Connected via Saddle Stitch** - The tee’s first point will be connected to another entity through a fabricated connection.

**To Place a Disconnected Tee:**

1. Start the Tee function.

2. The user is shown the name, spec, standards information, and End Treatment types of the current tee. The user is then prompted to provide one of four options:

   ```
   Command: '.TEEPIPE
   TEESTR2+1/2-1138, DemoSpec, ASME-B31-80, BW, BW, BW
   Specify first point or [eXchange ends/Stock/sAddle stitch]:
   ```

   - **Specify first point** - Select the start point of the tee anywhere on the drawing except for on another Pipe entity’s end point.
   - **Exchange ends** - Swaps the ends of the tee. Use this option when the selected stock has two dissimilar ends and the starting end, End Treatment 1, is not the correct end.
   - **Stock** - Allows the user to change the stock of the current tee type. The following dialog appears prompting the user to select a type from the current or alternate spec. Select **Default System** to change the system of the pipes.

   ![Pipe Spec and Stocks](image)

   Double click, or select the stock, and click **OK**.
   - **Saddle Stitch** - See **To Connect a Tee via Saddle Stitch**.

3. The user can immediately constrain the tee’s cardinal direction by entering one of the capitalized letters at the command line to activate one of the options. Or, by default, specify the direction with the mouse. The directions correspond to the world UCS.

   ```
   Specify direction or [Fwd/Aft/Port/Stbd/Up/Down]:
   ```

**To Connect a Tee to Another Pipe Entity:**

**Note:** A logical connection must exist between the End Treatments of the connecting stocks for completion of this command.

The function checks if the current default tee can form a legal connection with the connecting pipe. The following cases may occur:

- **Connection is valid** - The tee is drawn and you can continue as usual.
- **Connection is not valid** - A dialog offers a list of choices to choose from.
- **Connection is not valid, and no legal tee can be found within the same spec** - An error message is displayed.
1. Start the Tee function.

2. The user is shown the name, spec, standards information, and End Treatment types of the current tee. The user is then prompted to provide one of four options:

   Specify first point or [eXchange ends/Stock/sAddle stitch]:

or if the Tee command follows the placement of another Pipe entity, then an additional option is presented:

   Last end

   Specify First Point or [eXchange ends/Stock/sAddle stitch/Last end]:

   - **Exchange ends** - Swaps the ends of the pipe. Use this option when the selected stock has two dissimilar ends and the starting end, End Treatment 1, is not the one you want.
   
   - **Stock** - Allows the user to change the stock of the current tee type. The following dialog appears prompting the user to select a type from the current or alternate spec. Select **Default System** to change the system of the pipes.

   Double click, or select the stock, and click OK.

   - **Saddle stitch** - See To Connect a Tee via Saddle Stitch below.

   - **Last end** - The first point of the tee automatically snaps to the end of the last laid pipe. This option only appears if the command follows the placement of a previously connected pipe.

3. Select the free end of a Pipe entity OR enter “L” for Last end. The new tee will automatically align to the selected Pipe entity.

   **Tip**: Select the ShipConstructor OSNAP mode “Snap to Free pipe ends” to only select unconnected pipe ends.

   - The program verifies that a connection between the existing stock’s End Treatment, and the new tee’s End Treatment, exists before continuing. If no connection exists between the two entities then a log file will be generated, notifying the user that a connection needs to be created between the two listed End Treatments. The user will also have the choice to select a tee from a list of tees that will connect.

4. Complete the command by entering the Outlet orientation by specifying the direction with the mouse OR by entering the corresponding “key” letter of one of the options.

   Outlet orientation or [Fwd/Aft/Port/Stbd/Up/Down]:

**To Connect a Tee via Saddle Stitch**

Saddle stitches are fabricated connections to existing pipe runs. They may be added to straight pipes, bent pipes, and elbows. Any pipe type may connect to a saddle stitch as long as a connection has been created between the saddle stitch End Treatment and the current stocks End Treatment.

What you are really doing in the program when you add a saddle stitch is to create another endpoint on a pipe segment, to which other pipe entities may attach. For example, a straight pipe normally has only two ends to which a pipe may be connected. By adding a saddle stitch connection, you have the ability to tap into that pipe at any intermediate location.
Note: Since a saddle stitch is really an End Treatment, for a stock to connect with a saddle stitch, there needs to be a connection defined that includes: a) The saddle stitch End Treatment. b) The End Treatment of the connecting stock.

1. Start the Tee function.
2. Enter “a” or “A” for the saddle stitch option.
3. Select the existing pipe entity to which the new tee will attach.

4. If a saddle stitch is not already defined then a dialog will appear. Select the saddle stitch **EndTreatment Type**, the **Nominal Size** End Treatment for the connecting pipe, and **Description** for the type of saddle stitch and click **OK**.

5. The next prompt requests the user to select the location (insertion point) for the saddle stitch.

   **Tip**: If the osnap options are unable to provide the desired insertion point then one option is to create a temporary reference line. This line should be perpendicular to the pipe entity’s centerline - create the line at an end point, or the mid point, and then use the AutoCAD Offset command to make a copy of the line into the correct place. See the figure below.

6. The state of the Current Alignment, followed by the saddle direction options, is displayed. If the state is “free” then the reducer’s orientation has no limitations.
7. The next prompt allows a distance from the center of the pipe entity to be entered for an offset from the saddle stitch point. If no offset is desired then enter “0”.

Enter distance from Centerline [Enter for surface]:

If no connection is defined between the tee’s End Treatment & size, and the selected saddle stitch, then a message window will state this fact.

It may also offer the option to choose a tee that will connect if there exists a tee that can connect to the saddle stitch. Creating, and defining of connections, is done in the Catalog Stock Editor. For a brief summary on creating and defining a connection see the section To Connect an Elbow via Saddle Stitch.

Note: If no connection is defined between the two stocks via a saddle stitch then a pop up window will state this fact, recommend creating a connection and also offer an option to choose a reducer that will connect to the stock chosen for the saddle stitch point.

8. The following information is then displayed. Complete the command by entering the Outlet orientation by specifying the direction with the mouse OR by entering the corresponding “key” letter of one of the options.

Notes:

Pipes - Flange/Connector
Menu: SC Pipe / Pipes / Flange/Connector

Toolbar Button: 

Command Line: CONNECTORPIPE

Navigator: None

Creates a flanged connector segment.

Placing a Flange/Connector
The following are the steps to create a connector:

Creation of a flange/connector may be broken down into three different categories:

1. **Disconnected** - The flange/connector’s first point is not connected to any other Pipe entities.
2. **Connected** - The flange/connector’s first point is connected via a predefined connection to another Pipe entity.
3. **Connected via Saddle Stitch** - The flange/connector’s first point will be connected to another entity through a fabricated connection.

To Place a Disconnected Flange/Connector:

1. Start the Flange/Connector function.
2. The user is shown the name, spec, standards information, and End Treatment types of the current flange/connector. The user is then prompted to provide one of four options:

- **Specify first point** - Select the start point of the flange/connector anywhere on the drawing except for on another Pipe entity’s end point.
- **Exchange ends** - Swaps the ends of the flange/connector. Use this option when the selected stock has two dissimilar ends and the starting end, End Treatment 1, is not the correct end.
- **Stock** - Allows the user to change the stock of the current flange/connector type. The following dialog appears prompting the user to select a type from the current or alternate spec. Select **Default System** to change the system of the pipes.

3. The user can immediately constrain the flanges/connector’s cardinal direction by entering one of the capitalized letters at the command line to activate one of the options. Or, by default, specify the direction with the mouse. The directions correspond to the world UCS.

**To Connect a Flange/Connector to Another Pipe Entity:**

**Note:** A logical connection must exist between the End Treatments of the connecting stocks for completion of this command.

The function checks if the current default flange/connector can form a legal connection with the connecting pipe. The following cases may occur:

- **Connection is valid** - The flange/connector is drawn and you can continue as usual.
- **Connection is not valid** - A dialog offers a list of choices to choose from.
- **Connection is not valid, and no legal flange/connector can be found within the same spec** - An error message is displayed.

1. Start the Flange/Connector function.
2. The user is shown the name, spec, standards information, and End Treatment types of the current flange or connector. The user is then prompted to provide one of four options:
Specify first point or [eXchange ends/Stock/sAddle stitch]:

Or if the Flange/Connector command follows the placement of another flange or connector, then an additional option is presented: Last end

Specify First Point or [eXchange ends/Stock/sAddle stitch/Last end]:

- **Exchange ends** - Swaps the ends of the pipe. Use this option when the selected stock has two dissimilar ends and the starting end, End Treatment 1, is not the one you want.

- **Stock** - Allows the user to change the stock of the current flange or connector type. The following dialog appears prompting the user to select a type from the current or alternate spec. Select Default System to change the system of the pipes.

- **Last end** - The first point of the flange/connector automatically snaps to the end of the last laid pipe. This option only appears if the command follows the placement of a connected pipe.

3. Select the free end of a Pipe entity OR enter “L” for Last end. The new flange or connector will automatically align to the selected Pipe entity.

**Tip**: Select the ShipConstructor OSNAP mode “Snap to Free pipe ends” to only select unconnected pipe ends.

- The program verifies that a connection between the existing stock’s End Treatment, and the new flange’s End Treatment, exists before continuing. If no connection exists between the two entities then a log file will be generated, notifying the user that a connection needs to be created between the two listed End Treatments. The user will also have the choice to select a flange/connector from a list of flanges and/or connectors that will connect.

4. Complete the command by entering the Outlet orientation by specifying the direction with the mouse OR by entering the corresponding “key” letter of one of the options.

**To Connect a Flange/Connector via Saddle Stitch**

Saddle stitches are fabricated connections to existing pipe runs. They may be added to straight pipes, bent pipes, and elbows. Any pipe type may connect to a saddle stitch as long as a connection has been created between the saddle stitch End Treatment and the current stocks End Treatment.

What you are really doing in the program when you add a saddle stitch is to create another endpoint on a pipe segment, to which, other pipe entities may attach. For example, a straight pipe normally has only two ends to which a pipe may be connected. By adding a saddle stitch connection, you have the ability to tap into that pipe at any intermediate location.
Note: Since a saddle stitch is really an End Treatment, for a stock to connect with a saddle stitch, there needs to be a connection defined that includes: a) The saddle stitch End Treatment. b) The End Treatment of the connecting stock.

1. Start the Flange/Connector function.
2. Enter “a” or “A” for the saddle stitch option.
3. Select the existing pipe entity to which the new flange/connector will attach.

```
Select Pipe to add Saddle To:
```

4. If a saddle stitch is not already defined then a dialog will appear. Select the saddle stitch EndTreatment Type, the Nominal Size End Treatment for the connecting pipe, and Description for the type of saddle stitch and click OK.

```
Select Saddle Stitch End-Treatment for connection

EndTreatment Type: Saddle Stitch
Nominal Size: 2½ in
Description: Regular
```

5. The next prompt requests the user to select the location (insertion point) for the saddle stitch.

```
Choose the Saddle location:
```

**Tip:** If the osnap options are unable to provide the desired insertion point then one option is to create a temporary reference line. This line should be perpendicular to the pipe entity’s centerline - create the line at an end point, or the mid point, and then use the AutoCAD Offset command to make a copy of the line into the correct place. See the figure below.

Reference line positioned by using the Offset command.

6. The state of the Current Alignment, followed by the saddle direction options, is displayed. If the state is “free” then the flange/connector’s orientation has no limitations.
7. The next prompt allows a distance from the center of the pipe entity to be entered for an offset from the saddle stitch point. If no offset is desired then enter “0”.

Enter distance from Centerline [Enter for surface]:

If no connection is defined between the connector’s End Treatment & size, and the selected saddle stitch, then a message window will state this fact.

It may also offer the option to choose a connector if there exists a connector that can connect to the saddle stitch. Creating, and defining of connections, is done in the Catalog Stock Editor. For a brief summary on creating and defining a connection see the section To Connect an Elbow via Saddle Stitch.

Note: If no connection is defined between the two stocks via a saddle stitch then a pop up window will state this fact, recommend creating a connection and also offer an option to choose a reducer that will connect to the stock chosen for the saddle stitch point.

8. The following information is then displayed. Complete the command by entering the Outlet orientation by specifying the direction with the mouse OR by entering the corresponding “key” letter of one of the options.

Adding a FLGSLP4-3121 to a PIP 4:160
Outlet orientation or [Fwd/Aft/Up/Down]:

Pipes - Valve
Menu: SC Pipe / Pipes / Valve
Toolbar Button: ☐
Command Line: VALVEPIPE
Navigator: None
Places a predefined valve.

Placing a Valve
The following are the steps to place a valve:

Placement of a valve may be broken down into three different categories:

1. **Disconnected** - The valve’s first point is not connected to any other Pipe entities.
2. **Connected** - The valve’s first point is connected via a predefined connection to another Pipe entity.
3. **Connected via Saddle Stitch** - The valve’s first point will be connected to another entity through a fabricated connection.
To Place a Disconnected Valve:
1. Start the Valve function.
2. The user is shown the name, spec, standards information, and End Treatment types of the current valve. The user is then prompted to provide one of four options:

- **Specify first point** - Select the start point of the valve anywhere on the drawing except for on another Pipe entity’s end point.
- **Exchange ends** - Swaps the ends of the valve. Use this option when the selected stock has two dissimilar ends and the starting end, End Treatment 1, is not the correct end.
- **Stock** - Allows the user to change the stock of the current valve type. The following dialog appears prompting the user to select a type from the current or alternate spec. Select **Default System** to change the system of the pipes.

<table>
<thead>
<tr>
<th>Name</th>
<th>Catalog</th>
<th>Schedule</th>
<th>End Type 1</th>
<th>End Type 2</th>
<th>Weight</th>
</tr>
</thead>
<tbody>
<tr>
<td>Valve</td>
<td>VALV-BALL-PL4+BM4+ChainH, BALL, DemoSpec, ASME-B31-120, PL, BW</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Specify first point or [exchange ends/Stock/saddle stitch]:

- Double click, or select the stock, and click **OK**.
- **Saddle Stitch** - See To Connect a Valve via Saddle Stitch.

3. The user can immediately constrain the valve’s cardinal direction by entering one of the capitalized letters at the command line to activate one of the options. Or, by default, specify the direction with the mouse. The directions correspond to the world UCS.

Specify direction or [Fwd/Aft/Port/Stbd/Up/Down]:

To Connect a Valve to Another Pipe Entity:

**Note**: A logical connection must exist between the End Treatments of the connecting stocks for completion of this command.

The function checks if the current default valve can form a legal connection with the connecting pipe. The following cases may occur:

- **Connection is valid** - The valve is drawn and you can continue as usual.
- **Connection is not valid** - A dialog offers a list of choices to choose from.
- **Connection is not valid, and no legal valve can be found within the same spec** - An error message is displayed.

1. Start the Valve function.
2. The user is shown the name, spec, standards information, and End Treatment types of the current valve. The user is then prompted to provide one of four options:
Specify first point or [Exchange ends/Stock/Addle stitch]:

Or if the Valve command follows the placement of another pipe entity, then an additional option is presented: Last end.

Specify First Point or [Exchange ends/Stock/Addle stitch/Last end]:

- **Exchange ends** - Swaps the ends of the pipe. Use this option when the selected stock has two dissimilar ends and the starting end, End Treatment 1, is not the one you want.

- **Stock** - Allows the user to change the stock of the current valve type. The following dialog appears prompting the user to select a type from the current or alternate spec. Select Default System to change the system of the pipes.

  Double click, or select the stock, and click OK.

- **Saddle stitch** - See To Connect a Valve via Saddle Stitch.

- **Last end** - The first point of the valve automatically snaps to the end of the last laid pipe. This option only appears if the command follows the placement of a previously connected pipe.

3. Select the free end of a Pipe entity OR enter “L” for Last end. The new valve will automatically align to the selected Pipe entity.

  **Tip**: Select the ShipConstructor OSNAP mode “Snap to Free pipe ends” to only select unconnected pipe ends.

- The program verifies that a connection between the existing stocks End Treatment, and the new valve’s End Treatment, exists before continuing. If no connection exists between the two entities then a log file will be generated, notifying the user that a connection needs to be created between the two listed End Treatments. The user will also have the choice to select a valve from a list valves that will connect.

4. Complete the command by entering the Outlet orientation by specifying the direction with the mouse OR by entering the corresponding “key” letter of one of the options.

   Outlet orientation or [Fwd/Aft/Port/Stbd/Up/Down]:

**To Connect a Valve via Saddle Stitch**

Saddle stitches are fabricated connections to existing pipe runs. They may be added to straight pipes, bent pipes, and elbows. Any pipe type may connect to a saddle stitch as long as a connection has been created between the saddle stitch End Treatment and the current stocks End Treatment.

What you are really doing in the program when you add a saddle stitch is to create another endpoint on a pipe segment, to which, other pipe entities may attach. For example, a straight pipe normally has only two ends to which a pipe may be connected. By adding a saddle stitch connection, you have the ability to tap into that pipe at any intermediate location.

**Note**: Since a saddle stitch is really an End Treatment, for a stock to connect with a saddle stitch, there needs to be a connection defined that includes: a) The saddle stitch End Treatment. b) The End Treatment of the connecting stock.
1. Start the Valve function.

2. Enter “a” or “A” for the saddle stitch option.

3. Select the existing pipe entity to which the new valve will attach.

```
Select Pipe to add Saddle To:
```

4. If a saddle stitch is not already defined then a dialog will appear. Select the saddle stitch End Treatment Type, the Nominal Size End Treatment for the connecting pipe, and Description for the type of saddle stitch and click OK.

5. The next prompt requests the user to select the location (insertion point) for the saddle stitch.

```
Choose the Saddle location:
```

**Tip:** If the osnap options are unable to provide the desired insertion point then one option is to create a temporary reference line. This line should be perpendicular to the pipe entity’s centerline - create the line at an end point, or the mid point, and then use the AutoCAD Offset command to make a copy of the line into the correct place. See the figure below.

Reference line positioned by using the Offset command.

6. The state of the Current Alignment, followed by the saddle direction options, is displayed. If the state is “free” then the reducer’s orientation has no limitations.

```
Current Alignment: Free
Pick Saddle Direction [pERp/taN/Angle/Fwd/AFt/Port/Stbd/Up/Down]:
```

7. The next prompt allows a distance from the center of the pipe entity to be entered for an offset from the saddle stitch point. If no offset is desired then enter “0”.

```
Enter distance from Centerline [Enter for surface]:
```
If no connection is defined between the valve’s End Treatment & size, and the selected saddle stitch, then a message window will state this fact.

It may also offer the option to choose a valve that will connect if there exists a valve that can connect to the saddle stitch. Creating, and defining of connections, is done in the Catalog Stock Editor. For a brief summary on creating and defining a connection see the section To Connect an Elbow via Saddle Stitch.

Note: If no connection is defined between the two stocks via a saddle stitch then a pop up window will state this fact, recommend creating a connection and also offer an option to choose a reducer that will connect to the stock chosen for the saddle stitch point.

8. The following information is then displayed. Complete the command by entering the Outlet orientation by specifying the direction with the mouse OR by entering the corresponding “key” letter of one of the options.

<table>
<thead>
<tr>
<th>VALV-BALL-PL4+BW4+ChainH, BALL, DemoSpec, ASME-B31-120, PL, BW</th>
</tr>
</thead>
<tbody>
<tr>
<td>Conn = SAD-BW</td>
</tr>
<tr>
<td>Adding a VALV-BALL-PL4+BW4+ChainH to a PIP 4:160</td>
</tr>
<tr>
<td>Outlet orientation or [Fwd/Aft/Up/Down]:</td>
</tr>
</tbody>
</table>

**Pipes - Reducer**

**Menu:** SC Pipe / Pipes / Reducer

**Toolbar Button:**

**Command Line:** REDUCERPIPE

**Navigator:** None

Creates a reducer pipe segment.

**Placing a Reducer**

The following are the steps to place a reducer:

Creation of a reducer may be broken down into three different categories:

1. **Disconnected** - The reducer’s first point is not connected to any other Pipe entities.
2. **Connected** - The reducer’s first point is connected via a predefined connection to another Pipe entity.
3. **Connected via Saddle Stitch** - The reducer’s first point will be connected to another entity through a fabricated connection.

**To Place a Disconnected Reducer:**

1. Start the Reducer function.
2. The user is shown the name, spec, standards information, and End Treatment types of the current reducer. The user is then prompted to provide one of four options:
Specify first point - Select the start point of the reducer anywhere on the drawing except for on another Pipe entity’s end point.

Exchange ends - Swaps the ends of the reducer. Use this option when the selected stock has two dissimilar ends and the starting end, End Treatment 1, is not the correct end.

Stock - Allows the user to change the stock of the current reducer type. The following dialog appears prompting the user to select a type from the current or alternate spec. Select Default System to change the system of the pipes.

Double click, or select the stock, and click OK.

Saddle Stitch - See To Connect a Reducer via Saddle Stitch below.

3. The user can immediately constrain the reducer’s cardinal direction by entering one of the capitalized letters at the command line to activate one of the options. Or, by default, specify the direction with the mouse. The directions correspond to the world UCS.

Specify direction or [Fwd/Aft/Port/Stbd/Up/Down]:

To Connect a Reducer to Another Pipe Entity:

Note: A logical connection must exist between the End Treatments of the connecting stocks for completion of this command.

The function checks if the current default reducer can form a legal connection with the connecting pipe. The following cases may occur:

Connection is valid - The reducer is drawn and you can continue as usual.

Connection is not valid - A dialog offers a list of choices to choose from.

Connection is not valid, and no legal reducer can be found within the same spec - An error message is displayed.

1. Start the Reducer function.

2. The user is shown the name, spec, standards information, and End Treatment types of the current reducer. The user is then prompted to provide one of four options:

Specify first point or [exchange ends/stock/saddle stitch]:

Or if the Reducer command follows the placement of another Pipe entity, then an additional option is presented:

Last end.
Specify First Point or [eXchange ends/Stock/sAddle stitch/Last end]:

- **Exchange ends** - Swaps the ends of the reducer. Use this option when the selected stock has two dissimilar ends and the starting end, End Treatment 1, is not the one you want.

- **Stock** - Allows the user to change the stock of the current reducer type. The following dialog appears prompting the user to select a type from the current or alternate spec. Select **Default System** to change the system of the pipes.

- **Saddle stitch** - See To Connect a Reducer via Saddle Stitch.

- **Last end** - The first point of the reducer automatically snaps to the end of the last laid pipe. This option only appears if the command follows the placement of a previously connected pipe.

3. Select the free end of a Pipe entity OR enter “L” for Last end. The new reducer will automatically align to the selected Pipe entity.

   **Tip**: Select the ShipConstructor OSNAP mode “Snap to Free pipe ends” to only select unconnected pipe ends.

   - The program verifies that a connection between the existing stocks End Treatment, and the new reducer’s End Treatment, exists before continuing. If no connection exists between the two entities then a log file will be generated, notifying the user that a connection needs to be created between the two listed End Treatments. The user will also have the choice to select a reducer from a list of reducers that will connect.

### To Connect a Reducer via Saddle Stitch

Saddle stitches are fabricated connections to existing pipe runs. They may be added to straight pipes, bent pipes, and elbows. Any pipe type may connect to a saddle stitch as long as a connection has been created between the saddle stitch End Treatment and the current stocks End Treatment.

What you are really doing in the program when you add a saddle stitch is to create another endpoint on a pipe segment, to which, other pipe entities may attach. For example, a straight pipe normally has only two ends to which a pipe may be connected. By adding a saddle stitch connection, you have the ability to tap into that pipe at any intermediate location.

**Note**: Since a saddle stitch is really an End Treatment, for a stock to connect with a saddle stitch, there needs to be a connection defined that includes: a) The saddle stitch End Treatment. b) The End Treatment of the connecting stock.

1. Start the reducer function.
2. Enter “a” or “A” for the saddle stitch option.
3. Select the existing pipe entity to which the new reducer will attach.
4. If a saddle stitch is not already defined then a dialog will appear. Select the saddle stitch **EndTreatment Type**, the **Nominal Size** End Treatment for the connecting pipe, and **Description** for the type of saddle stitch and click **OK**.
5. The next prompt requests the user to select the location (insertion point) for the saddle stitch.

Choose the Saddle location:

**Tip**: If the osnap options are unable to provide the desired insertion point then one option is to create a temporary reference line. This line should be perpendicular to the pipe entity’s centerline - create the line at an end point, or the mid point, and then use the AutoCAD Offset command to make a copy of the line into the correct place. See the figure below.

Reference line positioned by using the Offset command.

6. The state of the Current Alignment, followed by the saddle direction options, is displayed. If the state is “free” then the reducer’s orientation has no limitations.

Current Alignment: Free
Pick Saddle Direction [pERp/taN/Angle/Fwd/Aft/Port/Stbd/Up/Down]:

7. The next prompt allows a distance from the center of the pipe entity to be entered for an offset from the saddle stitch point. If no offset is desired then enter “0”.

Enter distance from Centerline [Enter for surface]:

If no connection is defined between the reducer’s End Treatment & size, and the selected saddle stitch, then a message window will state this fact.
It may also offer the option to choose a reducer that will connect if there exists a reducer that can connect to the saddle stitch. Creating, and defining of connections, is done in the Catalog Stock Editor. For a brief summary on creating and defining a connection see the section To Connect an Elbow via Saddle Stitch.

**Note:** If no connection is defined between the two stocks via a saddle stitch then a pop up window will state this fact, recommend creating a connection and also offer an option to choose a reducer that will connect to the stock chosen for the saddle stitch point.

8. Complete the command by entering the Outlet orientation by specifying the direction with the mouse OR by entering the corresponding “key” letter of one of the options.

<table>
<thead>
<tr>
<th>Reducer Specifications</th>
<th>Connection</th>
<th>Outlet Orientation</th>
</tr>
</thead>
</table>

### Pipes - Cap

**Menu:** SC Pipe / Pipes / Cap

**Toolbar Button:**

**Command Line:** CAPPipe

**Navigator:** None

Placing a Cap

The placement of a cap may be broken down into three different categories:

1. **Disconnected** - The cap’s first point is not connected to any other Pipe entities.
2. **Connected** - The cap’s first point is connected via a predefined connection to another Pipe entity.
3. **Connected via Saddle Stitch** - The cap’s first point will be connected to another entity through a fabricated connection.

To Place a Disconnected Cap:

1. Start the Cap function.
2. The user is shown the name, spec, standards information, and End Treatment types of the current cap. The user is then prompted to provide one of three options:

   Command: "_CAPPipe"
   
   CAP4:40, DemoSpec, ASME-B31-40, BW

   - Specify first point - Select the start point of the cap anywhere on the drawing except for on another Pipe entity’s end point
- **Stock** - Allows the user to change the stock of the current cap type. The following dialog appears prompting the user to select a type from the current or alternate spec. Select **Default System** to change the system of the pipes.

![Stock dialog]

Double click, or select the stock, and click **OK**.

- **Saddle Stitch** - See **To Connect a Cap via Saddle Stitch**.

3. The user can immediately constrain the cap’s cardinal direction by entering one of the capitalized letters at the command line to activate one of the options. Or, by default, specify the direction with the mouse. The directions correspond to the world UCS.

   **Specify direction or [Fwd/Aft/Port/Stbd/Up/Down]:**

**To Connect a Cap to Another Pipe Entity:**

**Note:** A logical connection must exist between the End Treatments of the connecting stocks for completion of this command.

The function checks if the current default cap can form a legal connection with the connecting pipe. The following cases may occur:

- **Connection is valid** - The cap is drawn and you can continue as usual.
- **Connection is not valid** - A dialog offers a list of choices to choose from.
- **Connection is not valid, and no legal cap can be found within the same spec** - An error message is displayed.

1. Start the Cap function.
2. The user is shown the name, spec, standards information, and End Treatment types of the current cap. The user is then prompted to provide one of three options:

   **Specify first point or [Stock/sAddle stitch]:**

   - **Stock** - Allows the user to change the stock of the current cap type. The following dialog appears prompting the user to select a type from the current or alternate spec. Select **Default System** to change the system of the pipes.
• Saddle stitch - See To Connect a Cap via Saddle Stitch.

3. Select the free end of a Pipe entity. The new cap will automatically align to the selected Pipe entity.

Tip: Select the ShipConstructor OSNAP mode “Snap to Free pipe ends” to only select unconnected pipe ends.

• The program verifies that a connection between the existing stocks End Treatment, and the new cap’s End Treatment, exists before continuing. If no connection exists between the two entities then a log file will be generated, notifying the user that a connection needs to be created between the two listed End Treatments. The user will also have the choice to select a cap from a list of caps that will connect.

To Connect a Cap via Saddle Stitch

Saddle stitches are fabricated connections to existing pipe runs. They may be added to straight pipes, bent pipes, and elbows. Any pipe type may connect to a saddle stitch as long as a connection has been created between the saddle stitch End Treatment and the current stocks End Treatment.

What you are really doing in the program when you add a saddle stitch is to create another endpoint on a pipe segment, to which, other pipe entities may attach. For example, a straight pipe normally has only two ends to which a pipe may be connected. By adding a saddle stitch connection, you have the ability to tap into that pipe at any intermediate location.

Note: Since a saddle stitch is really an End Treatment, for a stock to connect with a saddle stitch, there needs to be a connection defined that includes: a) The saddle stitch End Treatment. b) The End Treatment of the connecting stock.

1. Start the cap function.
2. Enter “a” or “A” for the saddle stitch option.
3. Select the existing pipe entity to which the new cap will attach.
   Select Pipe to add Saddle To:

4. If a saddle stitch is not already defined then a dialog will appear. Select the saddle stitch EndTreatment Type, the Nominal Size End Treatment for the connecting pipe, and Description for the type of saddle stitch and click OK.
5. The next prompt requests the user to select the location (insertion point) for the saddle stitch.

Choose the Saddle location:

**Tip:** If the osnap options are unable to provide the desired insertion point then one option is to create a temporary reference line. This line should be perpendicular to the pipe entity’s centerline - create the line at an end point, or the mid point, and then use the AutoCAD Offset command to make a copy of the line into the correct place. See the figure below.

Reference line positioned by using the Offset command.

5. The state of the Current Alignment, followed by the saddle direction options, is displayed. If the state is “free” then the cap’s orientation has no limitations.

Current Alignment: Free
Pick Saddle Direction [pERp/taN/Angle/Fwd/Aft/Port/Stbd/Up/Down]:

6. The next prompt allows a distance from the center of the pipe entity to be entered for an offset from the saddle stitch point. If no offset is desired then enter “0”.

Enter distance from Centerline [Enter for surface]:

If no connection is defined between the cap’s End Treatment & size, and the selected saddle stitch, then a message window will state this fact.
It may also offer the option to choose a cap that will connect if there exists a cap that can connect to the saddle stitch. Creating, and defining of connections, is done in the Catalog Stock Editor. For a brief summary on creating and defining a connection see the section To Connect an Elbow via Saddle Stitch.

**Note:** If no connection is defined between the two stocks via a saddle stitch then a pop up window will state this fact, recommend creating a connection and also offer an option to choose a reducer that will connect to the stock chosen for the saddle stitch point.

---

### Pipes - Cross

**Menu:** SC Pipe / Pipes / Cross

**Toolbar Button:**

**Command Line:** CROSSPIPE

**Navigator:** None

Places a cross pipe segment.

---

**Placing a Cross**

The following are the steps to create a cross:

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

1. **Disconnected** - The cross’s first point is not connected to any other Pipe entities.
2. **Connected** - The cross’s first point is connected via a predefined connection to another Pipe entity.
3. **Connected via Saddle Stitch** - The cross’s first point will be connected to another entity through a fabricated connection.

---

**To Place a Disconnected Cross:**

1. Start the Cross function.
2. The user is shown the name, spec, standards information, and End Treatment types of the current cross. The user is then prompted to provide one of four options:

   ```
   Command: '_CROSSPIPE
   CRSS:4&2+1/2, DemoSpec, ASME-B31-120, PL, BW, PL, PL
   Specify first point or [eXchange ends/Stock/sAddle stitch/Last end]:
   ```

   - **Specify first point** - Select the start point of the cross anywhere on the drawing except for on another Pipe entity’s end point.
   - **Exchange ends** - Swaps the ends of the cross. Use this option when the selected stock has two dissimilar ends and the starting end, End Treatment 1, is not the correct end.
   - **Stock** - Allows the user to change the stock of the current cross type. The following dialog appears prompting the user to select a type from the current or alternate spec. Select **Default System** to change the system of the pipes.
Double click, or select the stock, and click **OK**.

- **Saddle Stitch** - See **To Connect a Cross via Saddle Stitch**.

3. The user can immediately constrain the cross’s cardinal direction by entering one of the capitalized letters at the command line to activate one of the options. Or, by default, specify the direction with the mouse. The directions correspond to the world UCS.

   \[ \text{Specify direction or [Fwd/Aft/Port/Stbd/Up/Down]}: \]

**To Connect a Cross to Another Pipe Entity:**

**Note:** A logical connection must exist between the End Treatments of the connecting stocks for completion of this command.

The function checks if the current default cross can form a legal connection with the connecting pipe. The following cases may occur:

- **Connection is valid** - The cross is drawn and you can continue as usual.
- **Connection is not valid** - A dialog offers a list of choices to choose from.
- **Connection is not valid, and no legal cross can be found within the same spec** - An error message is displayed.

1. Start the cross function.

2. The user is shown the name, spec, standards information, and End Treatment types of the current cross. The user is then prompted to provide one of four options:

   \[ \text{Specify first point or [eXchange ends/Stock/sAddele stitch]}: \]

   **Or** if the cross command follows the placement of another connected pipe entity, then an additional option is presented: **Last end**

   \[ \text{Specify First Point or [eXchange ends/Stock/sAddele stitch/Last end]}: \]

- **Exchange ends** - Swaps the ends of the pipe. Use this option when the selected stock has two dissimilar ends and the starting end, End Treatment 1, is not the one you want.
- **Stock** - Allows the user to change the stock of the current cross type. The following dialog appears prompting the user to select a type from the current or alternate spec. Select **Default System** to change the system of the pipes.
Double click, or select the stock, and click **OK**.

- **Saddle stitch** - See To Connect a Cross via Saddle Stitch.
- **Last end** - The first point of the cross automatically snaps to the end of the last laid pipe. This option only appears if the command follows the placement of a previously connected pipe.

3. Select the free end of a Pipe entity OR enter “L” for Last end. The new cross will automatically align to the selected Pipe entity.

**Tip:** Select the ShipConstructor OSNAP mode “Snap to Free pipe ends” to only select unconnected pipe ends.

- The program verifies that a connection between the existing stocks End Treatment, and the new cross’s End Treatment, exists before continuing. If no connection exists between the two entities then a log file will be generated, notifying the user that a connection needs to be created between the two listed End Treatments. The user will also have the choice to select a cross from a list of crosses that will connect.

4. Complete the command by entering the Outlet orientation by specifying the direction with the mouse **OR** by entering the corresponding “key” letter of one of the options.

### Adding a CRS:4+CS to a PIP-WD-CS-4_SCH40

Outlet orientation or [Fwd/Aft/Port/Stbd/Up/Down]:

#### To Connect a Cross via Saddle Stitch

Saddle stitches are fabricated connections to existing pipe runs. They may be added to straight pipes, bent pipes, and elbows. Any pipe type may connect to a saddle stitch as long as a connection has been created between the saddle stitch End Treatment and the current stocks End Treatment.

What you are really doing in the program when you add a saddle stitch is to create another endpoint on a pipe segment, to which, other pipe entities may attach. For example, a straight pipe normally has only two ends to which a pipe may be connected. By adding a saddle stitch connection, you have the ability to tap into that pipe at any intermediate location.

**Note:** Since a saddle stitch is really an End Treatment, for a stock to connect with a saddle stitch, there needs to be a connection defined that includes: a) The saddle stitch End Treatment. b) The End Treatment of the connecting stock.

1. Start the Cross function.
2. Enter “a” or “A” for the saddle stitch option.

### Select Pipe to add Saddle To:

3. Select the existing pipe entity to which the new cross will attach.
4. If a saddle stitch is not already defined then a dialog will appear. Select the saddle stitch **EndTreatment Type**, the **Nominal Size** End Treatment for the connecting pipe, and **Description** for the type of saddle stitch and click **OK**.
5. The next prompt requests the user to select the location (insertion point) for the saddle stitch.

   Choose the Saddle location:

   **Tip:** If the osnap options are unable to provide the desired insertion point then one option is to create a temporary reference line. This line should be perpendicular to the pipe entity’s centerline - create the line at an end point, or the mid point, and then use the AutoCAD Offset command to make a copy of the line into the correct place. See the figure below.

   Reference line positioned by using the Offset command.

6. The state of the Current Alignment, followed by the saddle direction options, is displayed. If the state is “free” then the reducer’s orientation has no limitations.

   Current Alignment: Free
   Pick Saddle Direction [pERp/taN/Angle/Fwd/AFt/Port/Stbd/Up/Down]:

7. The next prompt allows a distance from the center of the pipe entity to be entered for an offset from the saddle stitch point. If no offset is desired then enter “0”.

   Enter distance from Centerline [Enter for surface]:

   If no connection is defined between the cross’s End Treatment & size, and the selected saddle stitch, then a message window will state this fact.
It may also offer the option to choose a cross that will connect if there exists a cross that can connect to the saddle stitch. Creating, and defining of connections, is done in the Catalog Stock Editor. For a brief summary on creating and defining a connection see the section **To Connect an Elbow via Saddle Stitch**.

**Note:** If no connection is defined between the two stocks via a saddle stitch then a pop up window will state this fact, recommend creating a connection and also offer an option to choose a cross that will connect to the stock chosen for the saddle stitch point.

8. Complete the command by entering the Outlet orientation by specifying the direction with the mouse OR by entering the corresponding “key” letter of one of the options.

<table>
<thead>
<tr>
<th>Pipe Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CRS:4+CS, Fuel Oil, --MFR, PL, PL, PL, PL: SLEEVE (JOIN), Conn = SAD-PL</td>
<td></td>
</tr>
<tr>
<td>Adding a CRS:4+CS to a PIP-WD-CS-4_SCH40 Outlet orientation or [Fwd/Aft/Up/Down]:</td>
<td></td>
</tr>
</tbody>
</table>

**Pipes - Wye**

Menu: **SC Pipe / Pipes / Wye**

Toolbar Button: ➔

Command Line: **WYEPipe**

Navigator: **None**

Places a wye pipe segment.

**Placing a Wye**

The following are the steps to create a wye:

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

1. **Disconnected** - The wye’s first point is not connected to any other Pipe entities.
2. **Connected** - The wye’s first point is connected via a predefined connection to another Pipe entity.
3. **Connected via Saddle Stitch** - The wye’s first point will be connected to another entity through a fabricated connection.

**To Place a Disconnected Wye:**

1. Start the wye function.
2. The user is shown the name, spec, standards information, and End Treatment types of the current wye. The user is then prompted to provide one of four options:
Command: '_WYEPIPE
WYE4:40, DemoSpec, ASME-B31-40, BW, BW, BW
Specify first point or [eXchange ends/Stock/sAddle stitch/Last end]:

- **Specify first point** - Select the start point of the wye anywhere on the drawing except for on another Pipe entity’s end point

- **Exchange ends** - Swaps the ends of the wye. Use this option when the selected stock has two dissimilar ends and the starting end, End Treatment 1, is not the correct end.

- **Stock** - Allows the user to change the stock of the current wye type. The following dialog appears prompting the user to select a type from the current or alternate spec. Select **Default System** to change the system of the pipes.

  ![Stock Dialog](image)

  Double click, or select the stock, and click **OK**.

- **Saddle Stitch** - See **To Connect a Wye via Saddle Stitch**.

3. The user can immediately constrain the wye’s cardinal direction by entering one of the capitalized letters at the command line to activate one of the options. Or, by default, specify the direction with the mouse. The directions correspond to the world UCS.

   **Specify direction or [Fwd/Aft/Port/Stbd/Up/Down]:**

**To Connect a Wye to Another Pipe Entity:**

**Note:** A logical connection must exist between the End Treatments of the connecting stocks for completion of this command.

The function checks if the current default wye can form a legal connection with the connecting pipe. The following cases may occur:

- **Connection is valid** - The wye is drawn and you can continue as usual.

- **Connection is not valid** - A dialog offers a list of choices to choose from.

- **Connection is not valid, and no legal wye can be found within the same spec** - An error message is displayed.

1. Start the Wye function.
2. The user is shown the name, spec, standards information, and End Treatment types of the current wye. The user is then prompted to provide one of four options:

   **Specify first point or [eXchange ends/Stock/sAddle stitch]:**

   **Or** if the wye command follows the placement of another tee, then an additional option is presented:

   **Last end.**
Specify First Point or [eXchange ends/Stock/sAddle stitch/Last end]:

- **Exchange ends** - Swaps the ends of the pipe. Use this option when the selected stock has two dissimilar ends and the starting end, End Treatment 1, is not the one you want.

- **Stock** - Allows the user to change the stock of the current wye type. The following dialog appears prompting the user to select a type from the current or alternate spec. Select **Default System** to change the system of the pipes.

  - Double click, or select the stock, and click **OK**.

- **Saddle stitch** - See [To Connect a Wye via Saddle Stitch](#).

- **Last end** - The first point of the wye automatically snaps to the end of the last laid pipe. This option only appears if the command follows the placement of a previously connected pipe.

3. Select the free end of a Pipe entity OR enter “L” for Last end. The new wye will automatically align to the selected Pipe entity.

  **Tip:** Select the ShipConstructor OSNAP mode “Snap to Free pipe ends” to only select unconnected pipe ends.

- The program verifies that a connection between the existing stocks End Treatment, and the new wye’s End Treatment, exists before continuing. If no connection exists between the two entities then a log file will be generated, notifying the user that a connection needs to be created between the two listed End Treatments. The user will also have the choice to select a wye from a list of wyes that will connect.

4. Complete the command by entering the Outlet orientation by specifying the direction with the mouse **OR** by entering the corresponding “key” letter of one of the options.

### To Connect a Wye via Saddle Stitch

Saddle stitches are fabricated connections to existing pipe runs. They may be added to straight pipes, bent pipes, and elbows. Any pipe type may connect to a saddle stitch as long as a connection has been created between the saddle stitch End Treatment and the current stocks End Treatment.

What you are really doing in the program when you add a saddle stitch is to create another endpoint on a pipe segment, to which, other pipe entities may attach. For example, a straight pipe normally has only two ends to which a pipe may be connected. By adding a saddle stitch connection, you have the ability to tap into that pipe at any intermediate location.

**Note:** Since a saddle stitch is really an End Treatment, for a stock to connect with a saddle stitch, there needs to be a connection defined that includes: a) The saddle stitch End Treatment. b) The End Treatment of the connecting stock.

1. Start the Wye function.
2. Enter “a” or “A” for the saddle stitch option.
3. Select the existing pipe entity to which the new wye will attach.

4. If a saddle stitch is not already defined then a dialog will appear. Select the saddle stitch **EndTreatment Type**, the **Nominal Size** End Treatment for the connecting pipe, and **Description** for the type of saddle stitch and click **OK**.

5. The next prompt requests the user to select the location (insertion point) for the saddle stitch. Choose the Saddle location:

**Tip:** If the osnap options are unable to provide the desired insertion point then one option is to create a temporary reference line. This line should be perpendicular to the pipe entity’s centerline - create the line at an end point, or the mid point, and then use the AutoCAD Offset command to make a copy of the line into the correct place. See the figure below.

![Reference line positioned by using the Offset command.](image)

6. The state of the Current Alignment, followed by the saddle direction options, is displayed. If the state is “free” then the wye’s orientation has no limitations.

   **Current Alignment:** Free

   **Pick Saddle Direction** \{pERp/taN/Angle/Prd/APt/Port/Stbd/Up/Down\}:

7. The next prompt allows a distance from the center of the pipe entity to be entered for an offset from the saddle stitch point. If no offset is desired then enter “0”.

   **Enter distance from Centerline** \{Enter for surface\}:

   If no connection is defined between the wye’s End Treatment & size, and the selected saddle stitch, then a message window will state this fact.
It may also offer the option to choose a wye that will connect if there exists a wye that can connect to the saddle stitch. Creating, and defining of connections, is done in the Catalog Stock Editor. For a brief summary on creating and defining a connection see the section **To Connect an Elbow via Saddle Stitch**.

**Note:** If no connection is defined between the two stocks via a saddle stitch then a pop up window will state this fact, recommend creating a connection and also offer an option to choose a cross that will connect to the stock chosen for the saddle stitch point.

8. Complete the command by entering the Outlet orientation by specifying the direction with the mouse **OR** by entering the corresponding “key” letter of one of the options.

```
WYE4:40, DemoSpec, ASME-B31-40, BW, BW, BW
Conn = SAD-BW
Adding a WYE4:40 to a PIP 4:120
Outlet orientation or [Port/Stbd/Up/Down]:
```

### Pipes - Bent Pipe

**Menu:** SC Pipe / Pipes / Bent Pipe

**Toolbar Button:**

**Command Line:** BENTPIPE

**Navigator:** None

Creates a bent pipe segment.

**Placing a Bent Pipe**

The following are the steps to create a bent pipe:

Creation of a bent pipe may be broken down into three different categories:

1. **Disconnected** - The bent pipe’s first point is not connected to any other Pipe entities.
2. **Connected** - The bent pipe’s first point is connected via a predefined connection to another Pipe entity.
3. **Connected via Saddle Stitch** - The bent pipe’s first point will be connected to another entity through a fabricated connection.

**To Create a Disconnected Bent Pipe:**

1. Start the Bent Pipe function.
2. The user is shown the name, spec, standards information, and End Treatment types of the current pipe. The user is then prompted to provide one of four options:
Choose first point - Select the start point of the pipe anywhere on the drawing except for on another Pipe entity’s.

Radius - Allows the user to set the maximum radius of the bend.

Stock - Allows the user to change the stock of the current pipe type. The following dialogue appears prompting the user to select a type from the current or alternate spec. Select Default System to change the system of the pipes.

3. The user can immediately constrain the pipes cardinal direction or, by default, specify the next point and direction with the mouse. The directions correspond to the world UCS. The direction may also be specified using the regular AutoCAD options of entering an angle using @ and/or <. The @ is used by entering relative coordinates: @ 100, 50. This would be an angle defined by 100 units in the X direction and 50 units in the Y direction. The < is used by entering the angle in degrees with respect to the standard X-Y plane CCW direction unless AutoCAD is set to something else. Or combine the two by entering a length and a direction. Example: @1000<45. (This also works for 3D as well. @100,100,100 or <45<45 or @100<45<60.)

Choose next point or length or [Radius/Fwd/Aft/Port/Stbd/Up/Down/UNdo last]:

Choose next point - Select the next point of the pipe anywhere on the drawing except for on another Pipe entities.

Length - Allows the user to set the length and takes the direction from the mouse. No “key” letter needs to be entered.

Radius - Allows the user to set the maximum radius of the bend.

Fwd/Aft/Port/Stbd/Up/Down - Standard selection of directions.

Undo last - Undoes the last selected point.

Note: Pressing the “Enter” key at any time ends the Bent Pipe command. If two or more points have been selected during the drawing process then and only then will the pipe be completed.

4. Once two points have been selected a very handy feature becomes available. An alignment feature with the centerline of other pipe entities.

After having selected the first two points simply click on the end point of another pipe entity. This will NOT select the end point as a point to connect to. Unless their centerlines happen to share the same axis in space.
The line associated with this endpoint now forms an “Axis of Travel” for the next point to be chosen along. (See the red dashed line in the figure below. A red line will NOT actually appear in the drawing. It was just inserted into the figure for a clearer explanation.)

**Note:** You cannot directly choose the endpoint of another pipe entity to connect to unless the pipes’ centerlines happen to share the same central axis in space.

- Any point, except the entity’s end point, may now be selected.

**Note:** The radius set on the bent pipe will limit the angle of interception with the axis of the pipe entity.

- Once a point along the “Axis of Travel” has then been selected then the end point of the pipe entity may be chosen to form a connection.

![Axis Of Travel](image)

The red dashed line does NOT actually show up in ShipConstructor.

5. Bent pipes are based on straight pipe stocks that have a minimum and maximum length associated with them. Should the length that you provide exceed the defined maximum length, the user will be prompted with:

![AutoCAD Warning](image)

Choosing **Yes** will allow the user to lay as many pipes as necessary to cover the specified length. Choosing **No** will allow a continuous pipe to be laid down that will have to be broken up into stock length sections by the **SCSPLITBENTTOLEN** function.

Breaking a long pipe into stock length sections is done by:

- First select the pipe. Right click some where near it on the left hand side of the pipe and drag the dashed window over a portion of the pipe.
- Now that the bent pipe has been selected enter **SCSPLITBENTTOLEN** on the command line.
Command: scsplitbenttolen

- **OR** right click some where on the screen and a dialog box of options will show up.
  
  a) The fifth option from the bottom is **Edit Bent Pipe**.
  
  b) Choose the **Split to stock length** from the options to convert the bent pipe into stock length sizes.

Choose end to start from: Start/End:

- Enter the “key” letter of the end of the pipe that the first stock length pipe should start from.

Please enter clamp length:

- Enter the minimum length that the pipe-bending machine requires to clamp onto the pipe. Enter “0” if this is not a consideration that needs to be made.

**To Connect a Bent Pipe to Another Pipe Entity:**

*Note*: A logical connection must exist between the End Treatments of the connecting stocks for completion of this command.

The function checks if the current default pipe can form a legal connection with the connecting pipe. The following cases may occur:

- **Connection is valid** - The pipe is drawn and you can continue as usual.

- **Connection is not valid, but legal pipes can be found within the same spec** - A dialog offers a list of choice to choose from.

- **Connection is not valid, and no legal pipe can be found within the same spec** - An error message is displayed.

1. Start the Bent Pipe function.

2. The user is shown the name, spec, standards information, and End Treatment types of the current pipe. The user is then prompted to provide one of four options:

Choose first point or length or [Radius/Stock/Addle stitch];

**Or** if the pipe command follows the placement of another pipe entity, then an additional option is presented:

**Last end.**

Choose first point or length or [Radius/Stock/Addle stitch/Last end];

- **Radius** - Allows the user to set the maximum radius of the bend.

- **Stock** - Allows the user to change the stock of the current pipe type. The following dialog appears prompting the user to select a type from the current or alternate spec. Select **Default System** to change the system of the pipes.
Double click a stock, or select the stock, and click **OK**.

- **Saddle stitch** - See **To Connect a Bent Pipe via Saddle Stitch**.
- **Last end** - The first point of the pipe automatically snaps to the end of the last laid pipe. This option only appears if the command follows the placement of a previously connected pipe.

3. Select the free end of a Pipe entity OR enter “L” for Last end. The new pipe will automatically align to the selected Pipe entity.

**Tip**: Select the ShipConstructor OSNAP mode “Snap to Free pipe ends” to only select unconnected pipe ends.

- The program verifies that a connection between the existing stocks End Treatment, and the new pipe’s End Treatment, exists before continuing. If no connection exists between the two entities then a log file will be generated, notifying the user that a connection needs to be created between the two listed End Treatments. The user will also have the choice to select a pipe from a list of pipes that will connect.

4. Complete the command by pressing the **Enter** key or continue adding more bent pipe.

Adding a PIP 4:120 to a PIP 4:120

Choose next point or length or [Radius/Fwd/Port/Stbd/Up/Down/UNdo last]:

- **Choose next point** - Select the next point of the pipe anywhere on the drawing.
- **Length** - Allows the user to set the length and takes the direction from the mouse. No “key” letter needs to be entered.
- **Radius** - Allows the user to set the maximum radius of the bend.
- **Fwd/Aft/Port/Stbd/Up/Down** - Standard selection of directions.
- **Undo last** - Undoes the last selected point.

5. Specify the end point or choose the length option. Straight and bent pipe stocks have a minimum and maximum length associated with them. Should the length that you provide exceed the defined maximum length, the user will be prompted with:

Choosing **Yes** will allow the user to lay as many pipes as necessary to cover the specified length. Choosing **No** will allow more pipe to be laid down that will have to be broken up into stock length sections by the **SCSPLITBENTTOLEN** function.

Breaking a long pipe into stock length sections is done by:
• First select the pipe. Right click some where near it on the left hand side of the pipe and drag the dashed window over a portion of the pipe.

• Now that the bent pipe has been selected enter **SCSPLITBENTTOLEN** on the command line.

| Command: sscsplitbenttolen |

• **OR** right click some where on the screen and a dialog box of options will show up.
  c) The fifth option from the bottom is **Edit Bent Pipe**.
  d) Choose the **Split to stock length** from the options to convert the bent pipe into stock length sizes.

| Choose end to start from: Start/End: |

• Enter the “key” letter of the end of the pipe that the first stock length pipe should start from.

| Please enter clamp length: |

• Enter the minimum length that the pipe-bending machine requires to clamp onto the pipe. Enter “0” if this is not a consideration that needs to be made.

**To Connect a Bent Pipe via Saddle Stitch**

Saddle stitches are fabricated connections to existing pipe runs. They may be added to straight pipes, bent pipes, and elbows. Any pipe type may connect to a saddle stitch as long as a connection has been created between the saddle stitch End Treatment and the current stocks End Treatment.

What you are really doing in the program when you add a saddle stitch is to create another endpoint on a pipe segment, to which, other pipe entities may attach. For example, a straight pipe normally has only two ends to which a pipe may be connected. By adding a saddle stitch connection, you have the ability to tap into that pipe at any intermediate location.

The following figure shows a plain-ended pipe connected with a saddle stitch to produce an angled branch of pipe.

![Saddle Stitch Example](image)

**Note:** Since a saddle stitch is really an End Treatment, for a stock to connect with a saddle stitch, there needs to be a connection defined that includes: a) The saddle stitch End Treatment. b) The End Treatment of the connecting stock.

1. Start the Bent Pipe function.
2. Enter “a” or “A” for the saddle stitch option.
3. Select the existing pipe entity to which the new straight pipe will attach.

| Select a Pipe to add Saddle To: |

4. If a saddle stitch is not already defined then a dialog will appear. Select the saddle stitch **EndTreatment Type**, the **Nominal Size** End Treatment for the connecting pipe, and **Description** for the type of saddle stitch and click **OK**.
5. The next prompt requests the user to select the location (insertion point) for the saddle stitch.

Choose the Saddle location:

**Tip**: If the osnap options are unable to provide the desired insertion point then one option is to create a temporary reference line. This line should be perpendicular to the pipe entity’s centerline - create the line at an end point, or the mid point, and then use the AutoCAD Offset command to make a copy of the line into the correct place. See the figure below.

Reference line positioned by using the Offset command.

6. The state of the Current Alignment, followed by the saddle direction options, is displayed. If the state is “free” then the reducer’s orientation has no limitations.

   Current Alignment: Free
   Pick Saddle Direction [pERp/taN/Angle/Fwd/AFt/Port/Stbd/Up/Down]:

7. The next prompt allows a distance from the center of the pipe entity to be entered for an offset from the saddle stitch point. If no offset is desired then enter “0”.

   Enter distance from Centerline [Enter for surface]:

   If no connection is defined between the bent pipe’s End Treatment and size, and the selected saddle stitch, then a message window will state this fact.
It may also offer the option to choose a pipe that will connect if there exists a pipe that can connect to the saddle stitch. Creating, and defining of connections, is done in the Catalog Stock Editor. For a brief summary on creating and defining a connection see the section To Connect an Elbow via Saddle Stitch.

**Note:** If no connection is defined between the two stocks via a saddle stitch then a pop up window will state this fact, recommend creating a connection and also offer an option to choose a reducer that will connect to the stock chosen for the saddle stitch point.

8. Complete the command by pressing **Enter** to finish the pipe **OR** keep adding more pipe by selecting another point **OR** enter a length for another section of pipe and the direction with the mouse **OR** by entering the letter *r* to change the maximum radius of the bend.

```
Radius[100.0000]
```

```
PIP-WD-CS-4_SCH40, Fuel Oil, --MFR, PL, PL:
Conn = SAD-PL
Adding a PIP-WD-CS-4_SCH40 to a PIP-WD-CS-4_SCH40
Choose next point or length or [Radius]:
```

### Pipes - Add Saddle

**Menu:** SC Pipe / Pipes / Add Saddle  
**Toolbar Button:** None  
**Command Line:** ADDSADDLESTITCHCONNECTION  
**Navigator:** None

Adds a Saddle stitch End Treatment to a selected pipe.

Saddle stitches are fabricated connections to existing pipe runs. They may be added to straight pipes, bent pipes, and elbows. Any pipe type may connect to a saddle stitch as long as a connection has been created between the saddle stitch End Treatment and the stocks End Treatment.

What you’re really doing in the program when you add a saddle stitch, is to create an endpoint on a pipe segment, to which other pipe entities may attach. For example, a straight pipe normally has only two ends to which a pipe may be connected. By adding a saddle stitch connection, you have the ability to tap into that pipe at an intermediate location.
 Adding a Saddle Stitch

**Note:** Saddle stitch connections require End Treatments that include an End Treatment type with the IsSaddle flag set to yes. Make sure that you have already created the required saddle stitch End Treatment in the Catalog Stock Editor before you proceed.

**Note:** For best viewing while creating saddle stitches switch to 3D line mode in the Pipe Options / Display tab and switch to AutoCAD’s 2D Wireframe mode.

1. Start the Add Saddle command.
2. Select the pipe to which the saddle stitch will be made.
3. The next step is to select what End Treatment the saddle stitch should have. Only End Treatments that include an End Treatment type with the IsSaddle flag set to yes can be selected, so make sure that you have already created the required End Treatment in the Catalog Stock Editor before you proceed.

Select the **Endtreatment Type**. Once the type has been selected the **Nominal Sizes** defined for this type will be available for selection. Select the size of the saddle stitch. Finally, select the End Treatment of this size out of the Description list box and press OK.
4. Now three options are possible:

Specify first point or [Change end type/Specify distance from end]:

i) **Change end type** - Allows you to reselect the End Treatment type.

ii) **Specify distance from end** - Allows the saddle stitch to be placed a specified distance from the selected stocks end point.

   a. Select the end point from which the distance is to be measured.
   
   Please Select Stock End:

   b. Enter the distance from the end pipe end that the saddle stitch is to be placed.
   
   Specify Distance:

   c. Specify the direction.

   Specify direction or [Port/Stbd/Up/Down/Perp/Angle]:

   • **Port/Stbd/Up/Down** - Refers to the world UCS ship-coordinate system. The connecting pipe entity will only protrude from the selected side.

   • **Perp** - Constrains the saddle stitch so that the connected pipe will protrude in the current x-y plane at right angles to the centerline of the pipe.

iii) **Specify first point** - The default option. You will see a round icon slide up and down the centerline of the pipe entity as you move the mouse. This allows you to visually verify the first point before you select it with a left click of the mouse.

   a. The purpose of adding the saddle stitch is to prepare for the connection of another entity. Therefore, the next step is to constrain the direction of the attaching entity (straight pipes, caps, etc...):

   Specify direction or [Port/Stbd/Up/Down/Perp/Angle]:

   • **Port/Stbd/Up/Down** - refers to the world UCS ship-coordinate system. The connecting pipe entity will only protrude from the selected side.

   • **Perp** - constrains the saddle stitch so that the connected pipe will protrude in the current x-y plane at right angles to the centerline of the pipe.
5. Specify the distance from the centerline for the attaching entities connection point. Press **Enter**.

```
Specify distance from centerline in mm. 0.0000 or [on-Surface]:
```

### Pipes - Remove Saddle

**Menu:** Pipe / Pipes / Remove Saddle  
**Toolbar Button:** None  
**Command Line:** `REMOVESADDLESTITCHCONNECTION`  
**Navigator:** None

Removes a Saddle stitch End Treatment. Use this command to remove unused, in-use, or misplaced saddle stitch End Treatments.

**Note:** For best viewing of Saddle Stitches, do not use one of AutoCAD’s shade modes. Choose the 2D Wireframe mode.

#### To remove a saddle stitch from an entity

1. Start the **Remove Saddle** command.  
2. With the mouse, select the saddle stitch to remove.

```
Pick Custom End To Remove
```

### System Sub-Menu

![Set System]

- **Set System**
- **New / Modify System**

### System - Set System

**Menu:** Pipe / System / Set System  
**Toolbar Button:** None  
**Command Line:** `SCSETSYSTEM`  
**Navigator:** None

Changes the assigned system of selected entities.

#### To Create a New System or Branch

1. Start the **Set System** command.  
2. Select the objects to have their system changed.  
3. From the Set System dialog, select the desired system.
System - Edit System

Menu: **Pipe / System / Edit System**

Toolbar Button: **None**

Command Line: **SCEDITSYSTEM**

Navigator: **None**

Creates and edits Pipe systems. ShipConstructor - Pipe systems, also known as SWBS, can have two levels: system and branch. Currently all systems must be grouped under the main node which, by default, is named Pipe.

**To Create a New System or Branch**

1. Start the Edit System command.

2. To create a new system, select the top-most node in the tree structure (shown here as Pipe) and press **New**. To create a new branch in an existing system, select the system name from within the system tree. Press **New**.

3. Select the color you wish the systems pipes to appear in your Pipe models.

4. You can specify the test method, pressure, and class of system as well as mark the system as galvanized.

5. Press **OK** when done.

**To Delete a System or Branch**

1. Start the Edit System command.

2. Select the system or branch to delete.

3. Press **Delete**.

**Note:** You cannot delete a system or branch if pipes have been assigned to that system or branch.
Catalog Editor

Catalog Basics

All Pipe catalog data is stored in the same project database as the structure and the Pipe itself. An import function allows you to exchange complete catalogs, specs or individual pipes and pipe components from other project databases.

The Pipe Catalog Editor can be accessed through the following methods:

Menu:  SC Pipe / Catalog Editor

Toolbar Button: 

Command Line:  SCCATALOGEDITOR

Navigator:  None
Pipe Entity Overview

ShipConstructor Pipe currently supports ten entities: Straight pipes, Bent pipes, Elbows, Tees, Wyes, Crosses, Flanges, Caps, Reducers, and Valves. The catalog editor is the heart of the creation and maintenance of all the Pipe entities that will be used during your Pipe sessions. The concept is to recreate a physical catalog from, for instance, one of your current suppliers. Once the catalogs have been created you can group items into specs for inclusion in drawings.

All ShipConstructor Pipe entities are originally created by piecing together elements of information as shown in the following figure.
ShipConstructor Pipe is based on a sophisticated underlying database structure and is implemented through object-oriented software. Through these mechanisms, ShipConstructor Pipe introduces “intelligent” behavior. In order to make this work, the Pipe catalogs have to be built in a very specific format and order. This might be confusing initially, but makes your life a lot easier in the long run.

The tabs in the Catalog Editor present a logical order in the development of the Catalog items.

**Before you can define your first pipe or fitting in the catalog you have to define in this order:**

1. **Size Definitions** - Combine a Nominal Size with the outside diameter, wall thickness, and schedule defined by a Pipe standard. Example: ANSI B36.19 Stainless Steel Pipe defines: 3” = OD 3.5” - Schedule 40S - wall thickness 0.216

2. **End Treatments** - Combine a specific End Treatment with one or more Size Definitions and with a 3D visual representation. Example: Threaded Male - NS 3” - Cylinder Diameter 4” - Cylinder Length 5”
After defining the End Treatments, you can start creating pipes. However before the pipes can logically connect to each other you need to create “Connections” between the pipes’ End Treatments.

3. Create Pipes/Fittings - Using the information entered in the two previous steps, you can go ahead and start defining pipes and fittings.
   - The stocks you create must belong to a Catalog. You can create a single Catalog, or multiple Catalogs, during the creation of your first Pipe or alternatively you can use the Catalog tab to manage the Catalogs. Each stock can only belong to one Catalog. Later you can copy the stock to another Catalog.
   - Before a Pipe can be used in a drawing, it must be included in a spec. The same Catalog stock may belong to many specs.

4. Connections - Combine two End Treatments, any gap or overlap, and any accessories (bolts, seals) to a legal connection. The Pipe program will only allow connecting two pipe items that use a legal connection. Example: 3” Threaded Male can connect to 3” threaded female.

After this is done you can assign your pipes and fittings and to catalogs and specs.

**Size Definitions - Nominal Sizes & Standards**

Pipe codes and standards related to Pipe systems and Pipe components are set forward by various organizations (ANSI, ASME, ASTM, DIN, JIS). These standards bodies have each created further standards governing the manufacture and resultant strength of the pipes. Each standards body defines the wall thickness (schedule) accordingly for each Nominal Size.
As you can see from the above tables the Nominal Size is really a general size. It is used to describe a pipe in name only. The governing body’s Geometric Standard - Schedule defines the true geometric size, the OD and Wall thickness. Of course, one Geometric standard may have many schedules for each Nominal Size e.g. schedule 40, 80, 120.

**End Treatments**

The term End Treatments as used in Pipe is used to describe the preparation of the ends of pipes, elbows, tees, etc., as well as the ports in valves. They combine:

- **End Treatment Type** - A general description of the end/port. Ex. Flanged, Buttweld prep, threaded, etc.
- **Nominal Size** - The Nominal Size of the End Treatment.
- **General Properties** - Used to differentiate between End Treatments of the same size and type. For example consider a 3” threaded male pipe. Since the threads may be of many forms, you might have two End Treatments with a Nominal Size of 3” and End Treatment Type of threaded male, one being NPT and the other UNC. You would then use the Extended properties.
Remember: All pipes and fittings are based on their End Treatments.

**Connections**

Connections define which, and how, End Treatments are allowed to connect to each other. The main components of a connection are:

- **End Treatments 1 & 2** - These are the End Treatments that will be allowed to connect and are described in the preceding section.
- **Gap** - This is the physical space or overlap between the two connecting ends of the Pipe entities.
- **Display Properties** - Not implemented yet.
- **Connection Accessories** - Groups accessories such as gaskets and bolts with the connection.
Remember that every pipe or fitting will be based on the defined End Treatments. Thus defining connections, defines which pipes are allowed to connect to each other.

Defining legal connections before attempting to place Pipes will simplify Pipe drafting later on, as only legal entities will be displayed as options when connecting entities in the CAD drawing. The program automatically searches the database for “connectable” items to display.

**Pipe Entities**

After having read the above sections you may be wondering why all the above is necessary? Well, by splitting up the entry of required data, the overall work load is lessened. You can, and will reuse the Size, End Treatment, and Connection information many times in the creation of the Pipe entities that are ultimately used in the Pipe model.

Pipe entities define themselves physically by a combination of the End Treatments and the Geometric Standard assigned to them.
Additionally, there is one extra field that serves to increase the outer diameter of the Pipe entity. This last field can be used for elbows, valves, etc. where the outer diameter is not derived directly from the schedule. As you can see, the information entered in the Size Definitions and End Treatments tab is used in the definition of Pipe entities.

Other physical properties, such as a tee’s branch angle, or an elbow’s radius, are unique to each Pipe entity. These are discussed in detail in Create/Edit Pipe Tab section.

---

**Catalog Editor - Size Definitions**

Size Definitions combine Nominal Sizes (left of dialog) with International Standards (right of dialog). The definitions are later used to describe the ends or ports of any Pipe. The effect of this combination is to reproduce the size tables given as an example in the previous section.

Straight pipes have the same size definition on each side, but may have different End Treatments - reducers have different size definitions on each end - tees and other items use identical or various size definitions. Follow these main steps:

1. Define the Nominal Sizes you require in the left of the dialog.
2. Define the International Standard - Schedule you required in the right of the dialog.
3. Combine Nominal Sizes and the International Standard’s Schedule to complete the Size Definition.
Nominal Sizes

Nominal Sizes are given in inches or millimeters. Make sure to observe the radio buttons for the units in the top left of the dialog.

1. Press **Edit** in the Nominal Size frame to bring up the Edit Nominal Sizes dialog.
**Adding New Nominal Sizes**

1. Enter the new Nominal Size in the top right edit box. Then click **New**.

**Deleting a size**

1. Select the Nominal Size.
2. Click **Delete**.

**Note:** The program checks during saving to the database that there are no conflicts, which disallow deletion. Conditions for these are:

- The Nominal Size is used by a Size Definition.
- The Nominal Size is used by an End Treatment, which in turn might be used by one or more pipe stock items.

**Editing Nominal Sizes**

1. Select the Nominal Size to edit. After a short pause click on it again. You can now change the value of the Nominal Size.

**Note:** A warning dialog will be displayed if the selected Nominal Size is currently used by a Size Definition or by an End Treatment. If you go ahead with the change, all items based on this size will be changed accordingly.

**Standards**

Standards are grouped in three levels:

- International,
- Geometrical,
- Schedule.

To check out existing standards:

1. Select an **International Standard** (i.e. the governing international standards body such as ANSI). The associated **Geometric Standards** are displayed e.g. B36.10.

2. Select a **Geometric Standard**. The associated **Schedules** will be displayed. The schedule can be used for specifying regular schedules, weights (medium, heavy, etc.), or pressure class (3000 psi), so the term “schedule” is used in a loose sense.
To create or edit standards and schedules

Use the usual editing techniques to define standards and schedules.

**Note:** Each Size Definition has to be based on a standard. Define your own standard if the catalog is not clear about which standard has been used. You can later rename the standard or schedule, and all items using this will automatically be updated.
Size Definitions

In the center table of the Size Definitions tab is where the Nominal Size is joined with the Standard to define outer diameter and wall thickness. This information can be found in tables and a sample is included in the database.

<table>
<thead>
<tr>
<th>Nominal Size</th>
<th>Std.</th>
<th>Nominal</th>
<th>Schedule</th>
<th>OD</th>
<th>Wall thickness</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>ANSI</td>
<td>N36.10</td>
<td>STD</td>
<td>10.75</td>
<td>0.307</td>
</tr>
<tr>
<td>8</td>
<td>ANSI</td>
<td>N36.10</td>
<td>STD</td>
<td>8.25</td>
<td>0.277</td>
</tr>
<tr>
<td>6</td>
<td>ANSI</td>
<td>N36.10</td>
<td>STD</td>
<td>6.25</td>
<td>0.257</td>
</tr>
<tr>
<td>4</td>
<td>ANSI</td>
<td>N36.10</td>
<td>STD</td>
<td>4.5</td>
<td>0.237</td>
</tr>
</tbody>
</table>

To add a new Size Definition

1. Press Create / Edit Size Definitions.
2. Select a Nominal Size on the left, and a Schedule on the right of the tab. The >>Combine to New Definition<< button is now enabled.
3. Press >>Combine to New Definition<<. A new entry will be added to the table. The fields for OD and Wall thickness will be empty but editable and need to be entered according to the standard.
4. Press Save Size Definitions when finished to complete the additions.

Note: You cannot leave this tab or do anything else unless you have clicked Cancel or Save to leave the editing mode.

Note: Changing values such as wall thickness or OD will change all pipes using this Size Definition in any drawing attached to this database.

Note: You cannot delete Size definitions that are used by End Treatments or pipe stock. You will get a warning if you try to do so. You have to delete the End Treatments and/or pipe stocks first.

You are now ready to go to the next level - End Treatments.

End Treatments Tab

End Treatments are the physical descriptions of the ends of the pipes, fitting or machinery. End Treatments are later associated with pipes, elbows, and so on.
<table>
<thead>
<tr>
<th>Nominal Size</th>
<th>General Properties</th>
<th>End Treatment Type</th>
<th>3D Display Properties</th>
</tr>
</thead>
<tbody>
<tr>
<td>e.g. 6 inch</td>
<td>Help to differentiate between similar End Treatments e.g. Flange properties, Thread types</td>
<td>e.g. Flanged</td>
<td>e.g. Cylinder Length and Cylinder Radius to represent flange</td>
</tr>
</tbody>
</table>
**Example:**

You plan to buy pre-fabricated 4” straight pipes with flanges on each end. You follow these steps.

A. Define an End Treatment Type FLANGED. This is a generic type with no physical size associated with it.
B. Combine the End Treatment Type FLANGED with the 4” Nominal Size to create a specific End Treatment.
C. Specify the Cylinder Diameter and Length to represent the flange in 3D. **Note:** Currently only cylinders can be shown in the drawing for End Treatments. This will be expanded in future versions.
D. Specify the pipes using the Create / Edit Pipe tab using the just created End Treatment.

**End Treatment Types**

The list on the right of the dialog displays the available End Treatment types. Click the Edit button to modify this list. Use the usual procedures to create new ones, delete or modify them.

<table>
<thead>
<tr>
<th>Name</th>
<th>LineMode Icon</th>
<th>Is Saddle</th>
<th>LineMode Icon</th>
</tr>
</thead>
<tbody>
<tr>
<td>BW</td>
<td>BW Icon</td>
<td>No</td>
<td></td>
</tr>
<tr>
<td>FL</td>
<td>FL Icon</td>
<td>No</td>
<td></td>
</tr>
<tr>
<td>LG</td>
<td>FL Icon</td>
<td>No</td>
<td></td>
</tr>
<tr>
<td>SA</td>
<td>BW Icon</td>
<td>Yes</td>
<td></td>
</tr>
<tr>
<td>SB</td>
<td></td>
<td>No</td>
<td></td>
</tr>
<tr>
<td>SF</td>
<td>SC Icon</td>
<td>No</td>
<td></td>
</tr>
<tr>
<td>SD</td>
<td>SC Icon</td>
<td>No</td>
<td></td>
</tr>
<tr>
<td>SO</td>
<td></td>
<td>No</td>
<td></td>
</tr>
<tr>
<td>SWF</td>
<td>SW Icon</td>
<td>No</td>
<td></td>
</tr>
<tr>
<td>SWM</td>
<td>SW Icon</td>
<td>No</td>
<td></td>
</tr>
<tr>
<td>WAF</td>
<td>FL Icon</td>
<td>No</td>
<td></td>
</tr>
</tbody>
</table>

**Note:** You cannot delete an End Treatment Type if it is in use. Close the EndTreatment Types dialog and view the End Treatments that are based on the type.

**, Line Mode Display Icons**

End Treatments are displayed as icons when viewing them in LineMode or when created in a spool drawing. The icons are displayed at the end of the pipes. You can choose the icon size from a drop down list. The LineMode icon will be drawn in relative size to the OD of the pipe. Using 1.0 will draw the icon the same size as the OD of the pipe. You can choose to fill the icons.

**Note:** You can change the icons at a later time and all drawings will be updated upon opening.

**Note:** For a Flange symbol, select Rectangle as your LineMode Icon.
**Is Saddle flag**

This defines if the End Treatment can be used for a **Saddle stitch**, **O-let**, or **Stub-in** connection. While in practice these types of connections are different, in ShipConstructor - Pipe they are all referred to as Saddle Stitches.

**Saddle stitches, O-lets, and Stub-in** are fabricated connections to pipe headers. In ShipConstructor - Pipe they may be added to straight pipes, bent pipes, and elbows. Any pipe type may connect to a saddle stitch as long as a connection has been created between the saddle stitch End Treatment and the current stock’s End Treatment.

What you are really doing in the program when you add a saddle stitch is to create another endpoint on a pipe segment to which, other pipe entities may attach. For example, a straight pipe normally has only two ends to which a pipe may be connected. By adding a saddle stitch connection, you have the ability to tap into that pipe at any intermediate location. The following figure shows a plain-ended pipe connected with a saddle stitch, to produce an angled branch of pipe.

For all normal End Treatment Type’s this flag should be set to FALSE. If you plan to saddle stitch pipes, you have to define a saddle stitch End Treatment Type, which means the **IsSaddle** field is set to YES. Next, define an End Treatment using that Type for each Nominal Size you want plan to saddle stitch.

---

**Note:** Since a saddle stitch is really an End Treatment, for a stock to connect with a saddle stitch, there needs to be a connection defined that includes: a) The saddle stitch End Treatment. b) The End Treatment of the connecting stock.

---

**End Treatments**

**To Create End Treatments:**

1. Click the End Treatments **Edit** button.
2. Select a Nominal Size in the list on the left.

3. Click the New End Treatment button.

4. Select the proper End Treatment Type by clicking the Type cell. A drop list of all current End Treatment Types is displayed.

5. Specify a Cylinder Diameter and a Cylinder Type if desired. The Cylinder will be displayed in PipeMode not LineMode. The cylinder can represent the flange plate or a socket. If no display is required, enter “0” for the cylinder length and diameter.

6. To enter extended data such as flange type, class, and facing property make use of the General Property, Desc-Prop, and Facing-Prop fields. These fields can also be used to specify thread type, pitch, class, and any other information that is required but not contained in other fields.

**Note:** Do not worry that Cylinder length will affect the length of your pipe, elbow, etc. as specified later in the Create/Edit Pipe tab. The cylinder is drawn inward from the end of the Pipe entity.

### To Delete End Treatments:

1. Click the End Treatments **Edit** button.

2. Select the End Treatments to be deleted.

3. Click **Delete**.

**Note:** Two conditions must be met for an End Treatment to be deleted:

1. The End Treatment cannot be included in a connection. If it is then a message box opens informing you that the request violates this rule.

2. The End Treatment cannot be used by a Stock item. If the End Treatment is in use by a Stock item then an error message will appear indicating this.

Following this message a log file is generated and opened for viewing. The log will indicate:

- The Connection using the End Treatment.
- The Pipe drawings that include pipes that are connected using that End Treatment.

Following this message a log file is generated and opened for viewing. The log will indicate:

- The Stock type and the Stock name.
Create / Edit Pipe Tab

This tab provides all the functionality needed to create pipes and fittings as well as modify all associated data. The sections below describe the steps involved in creating each type of pipe or fitting.

**Note:** Before creating pipe stocks make sure to create at least one catalog name using the Catalog tab.

**Tip:** Make sure that after creating the pipes and fittings, there are connections defined that allow your pipes and fittings to connect.
Filter By

The left of the tab lets you filter the items displayed in the main list. The main list would display thousands of items if all items in the database would be displayed all the time. To reduce the number of listed items, use the provided filters.

Creating Pipes and Fittings

Several options exist when creating a new entity:

- Create a new stock that is not based on another stock items data.
- Create a new stock that copies information from a selected stock.
- Create a new stock that copies all information, but increments the size of the End Treatment to the next available size.

Whether creating or editing a stock item, you will make your changes through a common dialog that changes slightly to reflect the type of entity being created or edited. For instance, for the Tee, the dialog looks like:
**Common Properties Between Pipe Entities**

Each type of Pipe entity has certain, unique properties associated with it. Other properties are common among all entity types:

**Common Properties:**

- **Name** - The name of the entity. You can type any name you want.
- **Desc.** - A description for the entity.
- **Weight** - The weight in pounds if imperial, or in Kg if metric. For straight pipes the units are pounds per foot and Kg per meter respectively.
- **Model#** - The model number of the entity.
- **Order#** - The catalog order number.
- **User#** - The internal company reference number.
- **End n-Type** - The End Treatment type for end n. The available selections are based on the previously defined End Treatments that were created in the End Treatments tab.
- **Additional Thickness** - Adds external thickness to the stock. Pipes and fittings use the basic OD specified by the Nominal Size and schedule. In some cases this is not correct. For example a coupling generally has an OD greater than that of either pipe it is connecting. To make the coupling realistic, enter the extra material thickness to be added.
- **Standard** - The standards drop list contains a listing of all the schedules along with their Geometric and International standards. These entries were previously created from the Size Definitions tab. So if you don’t see the required schedule listed, go to the Size Definitions tab and create the required schedule.
- **Catalog** - Every item must belong to a catalog just as the items you purchase are found in a catalog. Items may be copied to other catalogs later using functions in the Catalog tab. Select the catalog you wish to the stock to belong to.

Item 1 in the catalog list, **Edit/Add Catalog(s)...**
lets you create, edit or delete catalog names.

Edit these values with the normal editing techniques.

- **Material** - The materials drop list shows all materials previously entered. If you do not see the material of the pipe you are creating, click item 1 in the list:

Clicking item 1, **Edit/Add Material(s)**... brings up the materials dialog from which the material properties may be set:
• Manufacturer - The Manufacturers drop list shows all the manufacturers previously entered. If you do not see the Manufacturer listed, select Item 1 in the Manufacturer list:

This lets you create, edit or delete manufacturer names.
To Copy / Create a New Straight Pipe:

1. In the Filter list, select the Pipe type (you do not have to check the box). The New button changes to reflect the type of entity that can be created. If you want to copy a previous item, select the item to copy the information from.
   - To create a new stock that is not based on another stock items data, make sure no items in the stock list are selected. Hint: if an item is selected, hold down the Ctrl key and click on the item.

2. Click New Pipe and the Creating New Pipe - Pipe dialog will open.

3. Edit the properties accordingly. See the section titled Common Properties Between Pipe Entities above for properties not specific to Straight Pipes. Properties that are specific to straight pipes are:
   - **End Treatment Size** - Straight pipes must have the same sized End Treatments. The outer diameter of the pipe is determined by the combination of schedule, selected in the Standard drop list, the Nominal Size of the End Treatment and the Add. Thick field. ShipConstructor - Pipe cross references the Nominal Size with the International Standard’s Schedule to determine the basic OD and then adds the additional thickness.
- **Min. Length** - The minimum length that the stock can exist. For the straight pipe, the minimum length must be less than the maximum length.

- **Max Length** - The maximum length that the stock can be purchased or handled.
  - When placing straight pipes, the program takes these lengths into account.

4. Use the preview window to verify the appearance of the pipe.

5. Press **Done** when finished.

**To Copy / Create a New Elbow:**

1. In the Filter list, select the **Elbow** type (you do not have to check the box). The **New** button changes to reflect the type of entity that can be created. If you want to **copy** a previous item, select the item to copy the information from.
   - **Hint:** if an item is selected, hold down the Ctrl key and click on the item.

2. Click **New Elbow** and the Creating New Pipe - Elbow dialog will open.
3. Edit the properties accordingly. See the section titled **Common Properties Between Pipe Entities** above for properties not specific to Elbows. Properties that are specific to Elbows are:

- **End Treatment Size** - Elbows may have different sized End Treatments, thus making them reducing elbows. The outer diameter of the Elbow is determined by the combination of schedule, selected in the **Standard** drop list, and Nominal Size of the End Treatment. ShipConstructor - Pipe cross-references the Nominal Size with the International Standard’s Schedule to determine the basic OD and then adds the additional thickness.

- **End 1, 2 - Length** - As shown in the diagram, these lengths are from the intersection of the centerline axes, to the end faces. Either length cannot be less than the radius as this would make for an incomplete elbow.

- **Angle** - Total swept angle of the elbow. Allowed angles are: $0^\circ \leq \text{Angle} < 360^\circ$.

- **Radius** - The radius can be entered as a function of the diameter, or entered explicitly. The radius cannot exceed the value of either of the End Lengths.

4. Use the preview window to verify the appearance of the Elbow.

5. Press **Done** when finished.

**To Copy / Create a Tee:**

1. In the Filter list, select the **Tee** type (you do not have to check the box). The **New** button changes to reflect the type of entity that can be created. If you want to copy a previous item, select the item to copy the information from.

   - To create a new stock that is not based on another stock items data, make sure no items in the stock list are selected. **Hint**: if an item is selected, hold down the Ctrl key and click on the item.
2. Click **New Tee** and the Creating New Pipe - Tee dialog will open.

3. Edit the properties accordingly. See the section titled **Common Properties Between Pipe Entities** above for properties not specific to Tees. Properties that are specific to Tees are:
• **End Treatment Size** - Tees may have different sized End Treatments, thus making them reducing Tees. The outer diameter of the Tee is determined by the combination of schedule, selected in the **Standard** drop list, and Nominal Size of the End Treatment. ShipConstructor - Pipe cross-references the Nominal Size with the International Standard’s Schedule to determine the basic OD and then adds the additional thickness.

• **End [n] - Length** - As shown in the diagram, these lengths are from the intersection of the centerline axes, to the end faces.

• **Branch Angle** - Angle of the Tee branch. Allowed angles are: \( 0^\circ \leq \text{Angle} < 360^\circ \).

4. Use the preview window to verify the appearance of the Tee.

5. Press **Done** when finished.

**To Copy / Create a Flange/Connector:**

Flange/Connectors are a versatile type that can be used to represent any item that you wish, with up to three cylinders. For instance, flanges, couplings, nipples, and O-lets may all be defined as this basic type.

Flange/Connectors may be thought of as special pipes, i.e., straight with two End Treatments. Unlike pipes, however, Flange/Connectors are not constrained to having the same sized End Treatments and a fixed length. Visually, you may wish to have their appearance derived solely from the assigned End Treatments or include the center straight section with a combination of Treatments. Use your creativity in creating parts.

**Note:** Flange/Connectors include two End Treatments and therefore both ends require a Connection before attempting to connect the object to another Pipe object.

1. In the Filter list, select the **Flange/Connector** type (you do not have to check the box). The **New** button changes to reflect the type of entity that can be created. If you want to **copy** information from a previous item, select the item to copy the information from and then press **New**.

   ➢ To create a new stock that is not based on another stock items data, make sure no items in the stock list are selected. Hint: if an item is selected, hold down the Ctrl key and click on the item.

2. Click **New Flange/Connector** and the Creating New Pipe - Flange/Connector dialog will open.
3. Edit the properties accordingly. See the section titled **Common Properties Between Pipe Entities** above for properties not specific to Flange/Connectors. Properties that are specific to Flange/Connectors are:

- **End Treatment Size** - Flange/Connectors may have different sized End Treatments. The outer diameter of the center portion (portion between End Treatments) is determined by the combination of schedule, selected in the **Standard** drop list, and Nominal Size of End Treatment 1. ShipConstructor - Pipe cross-references the Nominal Size with the International Standard’s Schedule to determine the basic OD and then adds the additional thickness.

  If either of the End Treatment lengths exceeds the overall length, the End Treatment will protrude from the opposite side.

- **Length** - As shown in the diagram, the length is the overall length of the fitting. The End Treatments draw themselves inwards from the end points.
Flange/Connector

- **Flange / Connector Types** - Allows the part to be identified as a specific type. The Types are user defined.

  To define a Flange/Connector Type:
  
  I. Click **Edit Flange/Connector Types >>**
  
  II. In the Edit Flange/Connector Types dialog, create your Type with the usual techniques.
III. Press **OK** when done.

4. Use the preview window to verify the appearance of the Flange/Connector.

5. Press **Done** when finished.

**Example 1: Representing a slip-on flange**

1. Since a flange can be represented as two cylinders, define two End Treatments:
   a) **End Treatment 1** - Use to represent the actual flange portion. The End Treatment has a cylinder size associated with it that will be used to represent the flange portion.
   b) **End Treatment 2** - Use to represent the slip-on portion of the flange. Again use the End Treatment’s diameter and length properties to represent the base portion of the flange.

2. In the Catalog Editor, go to the Create/Edit Pipe tab.
3. Create a new Flange/Connector Type called Flange.
4. Enter the length and other properties defining the flange.
5. Ensure that Connections exist between the End Treatments of the new flange you just created, and the End Treatments of the other pipes or fittings you will be attaching to.
6. Add the flange to the required specs.
7. The flange can now placed in a drawing.

**Example 2: Representing a Threaded, Female Coupling**

A female coupling can be represented simply by one cylinder.

1. Define End Treatments for the threads. Create one End Treatment, if the threads are identical, or two End Treatments if the threads are distinct. The End Treatments need not have a value entered for the cylinder length/diameter as the entire portion of the coupling can be obtained from the Nominal Size, schedule, additional thickness, and length.
2. In the Catalog Editor, go to the Create/Edit Pipe tab. Create a new Flange/Connector.
3. Create a new Flange/Connector Type called Coupling. This will help you distinguish the stocks later on.
4. Enter the length and other properties that define the flange. Specifically, pay attention to the Nominal Size of End Treatment 1 and the schedule you pick. The base OD of the coupling is derived from the cross reference of the Nominal Size with the Schedule in the Size Definitions tab.
5. Ensure that Connections exist between the End Treatments of the new coupling you just created and the End Treatments of the other pipes or fittings you will be attaching to.
   - For a threaded female coupling, enter a negative value for the gap. This way the male will be inserted into the coupling.
6. Add the coupling to the required specs.
7. The coupling can now placed in a drawing.

**To Copy / Create a Valve:**

Each valve may be customized through user-defined handles. For instance, a gear-actuated valve may have the gearbox, hand crank, and other dimensions customized to produce a realistic representation.

1. In the Filter list, select the Valve type (you do not have to check the box). The New button changes to reflect the type of entity that can be created. If you want to copy information from a previous item, select the item to copy the information from and then press New.
   - To create a new stock that is not based on another stock items data, make sure no items in the stock list are selected. **Hint**: if an item is selected, hold down the Ctrl key and click on the item.
2. Click **New Valve** and the Creating New Pipe - Valve dialog will open.

3. Edit the properties accordingly. See the section titled **Common Properties Between Pipe Entities** above for properties not specific to Valves. Properties that are specific to Valves are:

   - **End Treatment Size** - Valves may have different sized End Treatments. The diameter of the end cones (ports) are determined by the combination of schedule, selected in the **Standard** drop list, and Nominal Size of End Treatments 1 and 2. ShipConstructor - Pipe cross-references the Nominal Size with the International Standard’s Schedule to determine the basic OD and then adds the additional thickness.

   - **End [n] Length** - As shown in the diagram, these lengths represent the length of the end cones.

   - **Handle** - Currently there are nine different basic handle types. An unlimited number of Valve Handles may be created by customizing the dimensions of the base Handle.
To Create a Valve Handle

1. Click Edit Valve Handles>> to open the Edit Valve Handles dialog.

2. Choose whether the dimensions are in inches or millimeters by selecting the Metric or Imperial radio button. You cannot change the Units after addition. To change the units, press OK. This saves the information to the database. Then reopen the dialog and change the units.

3. Click New. A new entry will appear in the handle list.

4. Change the Handle Type of the new entry by clicking on it. From the drop down list select the handle type.
5. Enter the dimensions, A - H, as appropriate for the Handle Type.
6. Verify the handle type and relative dimensions in the preview window.
7. Press OK to save the information to the database. The Handle can now be added to a valve.

- **Handle Position** - The distance from port 1 to the center axis of the handle.
- **Rel. Port Angle** - Rotates the axis of port 2 clockwise by the amount in degrees. This effectively reduces the angle between port 1 and port 2 from 180°.
- **Handle angle rel. Port 1 face** - The tilt angle of the handle towards port 1. Measured clockwise in degrees.
- **Handle rot. Angle about Port 1 axis** - The angle the handle is rotated clockwise about port 1’s axis as viewed from end 1.
- **Valve Types** - Allows the valve to be identified as a specific type. An unlimited number of types may be defined, each having one of four different visual representations: ball, arrow (for directional flow), cylinder, or none.

  - Valve Types are user defined, abstract names with no visual representation. The visual representation of the valve is determined once the **icon** is applied to that type.

To define a Valve Type:

I. Click **Edit Valve Types >>**

II. In the Edit Valve Types dialog, create your Type with the usual techniques.
III. Select the **Valve Icon** to be used in the center of your valves. Valve Icons are defined internally to Pipe and allow for a visual representation of the type of valve i.e. a sphere could be used for a ball valve, the arrow for a check valve. Following are examples of the four Valve Icons:

<table>
<thead>
<tr>
<th>Valve Type</th>
<th>Valve Icon</th>
</tr>
</thead>
<tbody>
<tr>
<td>DVL</td>
<td>Sphere</td>
</tr>
<tr>
<td>Arrow</td>
<td>Cylinder</td>
</tr>
<tr>
<td>None</td>
<td>None</td>
</tr>
</tbody>
</table>

IV. Press **OK** when done.

4. Use the preview window to verify the appearance of the Valve.

5. Press **Done** when finished.

**To Copy / Create a Reducer:**

Reducers may be concentric or eccentric.
1. In the Filter list, select the **Reducer** type (you do not have to check the box). The **New** button changes to reflect the type of entity that can be created. If you want to **copy** information from a previous item, select the item to copy the information from and then press **New**.

   ➢ To create a new stock that is not based on another stock items data, make sure no items in the stock list are selected. **Hint**: if an item is selected, hold down the Ctrl key and click on the item.

2. Click **New Reducer** and the Creating New Pipe - Reducer dialog will open.

3. Edit the properties accordingly. See the section titled **Common Properties Between Pipe Entities** above for properties not specific to Reducers. Properties that are specific to Reducers are:

   • **End Treatment Size** - Reducers may have different sized End Treatments. The diameter of the end cones (ports) are determined by the combination of schedule, selected in the **Standard** drop list, and Nominal Size of End Treatments 1 and 2. ShipConstructor - Pipe cross-references the Nominal Size with the International Standard’s Schedule to determine the basic OD and then adds the additional thickness.

     ➢ **Remember**: End Treatments draw themselves over top of the ends. i.e. for flanged End Treatments you would not see the ends of the cones as they would be obscured by the flanges.
• **End [n] Length** - As shown in the diagram, these lengths represent the length of the straight portions of the reducer.

• **Length** - The length of the reduction cone.

• **Concentric / Eccentric** - Concentric reducers share a common centerline whereas eccentric reducers have offset centerlines.

![Eccentric Concentric Reducers](image)

• **Offset** - If the reducer is of the eccentric type, this value specifies the offset distance of the axes.

4. Press **Done** when finished.

**To Copy / Create a Cross:**

1. In the Filter list, select the **Cross** type (you do not have to check the box). The **New** button changes to reflect the type of entity that can be created. If you want to **copy** information from a previous item, select the item to copy the information from and then press **New**.

   - To create a new stock that is not based on another stock items data, make sure no items in the stock list are selected. **Hint**: if an item is selected, hold down the Ctrl key and click on the item.

2. Click **New Cross** and the Creating New Pipe - Cross dialog will open.
3. Edit the properties accordingly. See the section titled **Common Properties Between Pipe Entities** above for properties not specific to Crosses. Properties that are specific to Crosses are:

- **End Treatment Size** - Crosses may have different sized End Treatments. The diameter of the ends are determined by the combination of schedule, selected in the **Standard** drop list, and Nominal Size of End Treatments **1 - 4**. ShipConstructor - Pipe cross-references the Nominal Size with the International Standard’s Schedule to determine the basic OD and then adds the additional thickness.

- **End [n] Length** - As shown in the diagram, these lengths represent the length of each straight portion of the Cross, from the center outward.

4. Press **Done** when finished.

**To Copy / Create a Wye:**

1. In the Filter list, select the **Wye** type (you do not have to check the box). The **New** button changes to reflect the type of entity that can be created. If you want to **copy** information from a previous item, select the item to copy the information from and then press **New**.

   - To create a new stock that is not based on another stock items data, make sure no items in the stock list are selected. **Hint**: if an item is selected, hold down the Ctrl key and click on the item.
2. Click **New Wye** and the Creating New Pipe - Wye dialog will open.

3. Edit the properties accordingly. See the section titled **Common Properties Between Pipe Entities** above for properties not specific to Wyes. Properties that are specific to Wyes are:

   - **End Treatment Size** - Wyes may have different sized End Treatments. The diameters of the ends are determined by the combination of schedule, selected in the **Standard** drop list, and Nominal Size of End Treatments 1 - 3. ShipConstructor - Pipe cross-references the Nominal Size with the International Standard’s Schedule to determine the basic OD and then adds the additional thickness.

   - **End [n] Length** - As shown in the diagram, these lengths represent the length of each straight portion of the Wye, from the branch origin outward.

   - **Branch Angle** - The angle between branch 2 and branch 3. The branch angles are symmetric about the branch 1 axis and lie in the same plane.
4. Press **Done** when finished.

**To Copy / Create a Cap:**

Caps may also be used to represent plugs by selecting a connection with a negative gap value and selecting the correct cap type.

1. In the Filter list, select the **Cap** type (you do not have to check the box). The **New** button changes to reflect the type of entity that can be created. If you want to **copy** information from a previous item, select the item to copy the information from and then press **New**.
   - To create a new stock that is not based on another stock items data, make sure no items in the stock list are selected. **Hint**: if an item is selected, hold down the Ctrl key and click on the item.

2. Click **New Cap** and the Creating New Pipe - Cap dialog will open.

3. Edit the properties accordingly. See the section titled **Common Properties Between Pipe Entities** above for properties not specific to Caps. Properties that are specific to Caps are:
- **End Treatment Size** - Caps only have one End Treatment. The diameter of the Cap is determined by the combination of schedule, selected in the **Standard** drop list, and Nominal Size of End Treatments. ShipConstructor - Pipe cross-references the Nominal Size with the International Standard's Schedule to determine the basic OD and then adds the additional thickness.

- **Length** - As shown in the diagram, the length represents the length of main portion of the Cap.

- **Wrench-Flat Length** - If **Cap Type** is set to square or recessed square then you may enter a value for the Flat length. If no value is entered, no flat will be drawn.

- **Cap Type** - Caps may be drawn in one of four different styles:

  - Plain Cylinder
  - Round
  - Recessed Square
  - Square

4. Press **Done** when finished.

**Edit**

Opens a separate dialog that allows you to edit all properties of a specific pipe or fitting. All items use the same type of dialog, with all data values displayed in the list on the left. Just click on any item to edit it.

- Black - Values that have not been changed.
- Red - Values that have been changed.

**To Edit a Stock Item**

1. Select a pipe or fitting in the main list.
2. Click **Edit**.
3. In the dialog, edit any data. See the section above - **Creating Pipes and Fittings**.
4. Press **Done** when finished.

**Penetrations**

In ShipConstructor - Pipe, penetration objects are cutouts that allow pipes to pass through a part of the structure. They are associated with a particular stock item, and their name, size, and shape is defined in the Catalog Editor. Since a particular pipe stock item may be used in many instances, many different penetration objects will likely be defined for each stock.

Penetrations are added to the drawing after running an interference check. Using ShipConstructor’s Interference checking, you first find the locations where components of the ship model interfere with each other - in the case of Pipe, the location where Pipe entities penetrate the ship structure. Rather than have the user manually find the location and then create the necessary clearance holes, ShipConstructor automatically places them for you. Before these clearance holes are created however, the penetration object must be added to the associated interference and then applied. Applying the penetrations is the step that actually creates the clearance hole.
Penetrations Dialog

The Pipe Penetrations dialog is where penetration objects are defined and associated with a particular pipe stock. Currently only straight pipes and bent pipes can have penetration objects assigned to them (bent pipes are based on straight pipes).

The Penetrations Dialog can be broken down into four areas:

- **Filter Options** - Used to include or exclude pipe stocks from the display
- **Create Pipe Penetrations** - Used to create the cutout information such as: name, shape, and size.
- **Pipe Stock List** - Displays the filtered pipe stocks.
• **Pipe Penetrations** - The Pipe Penetrations list reflects the penetration objects already associated with the selected pipe stock. For instance, in the above figure, the penetrations shown in the Pipe Penetrations list have all been assigned to the pipe, PIP-SM-CS-3_SCH40.

**To Create a Penetration Object and Assign it to Pipe Stock:**

1. Choose the pipe stock[s] that the penetration will be created for.
2. From the **Type** drop list, choose the type of penetration.

   ![Penetration Types](image)

   If the Type of penetration required is not displayed, then select **Edit Types** and create it in the Penetrations Type dialog using the usual editing techniques.

3. Select the cutout shape, either round or rectangular. The dimension map and text boxes will change to reflect the information required.
4. Select the units of the cutout, either Imperial (in) or Metric (mm).

5. Enter the dimensions required of the cutout.

6. Once you’re satisfied with the cutout press the Create button. The cutout will be added to the selected pipes.

Deleting Penetration Objects

1. Select the stock with the penetration objects that are to be deleted.

2. In the penetrations list, select the penetrations you wish to delete.

3. Press the Delete button.

**Note:** You cannot delete any penetrations that are used in drawings. Upon attempting to delete a penetration that is used in a drawing, a report will be generated, informing the user of the drawings where the penetrations are used.

Connections Tab

Connections define which, and how, End Treatments connect to each other. Remember that every pipe or fitting will be based on the defined End Treatments. Thus defining connections, defines which pipes are allowed to connect to each other.

Defining legal connections will simplify pipe drafting later on, as only legal entities will be displayed as options when connecting in the CAD drawing. The program automatically searches the database for allowed items to display.
Connections are defined by:

- Specifying an End Treatment for both connecting items.
- The type of connection that shall be displayed.
- A gap or overlap. A gap is specified by a positive value, and an overlap is specified by a negative value.
To Create a New Connection

<table>
<thead>
<tr>
<th>Name</th>
<th>Treatment 1</th>
<th>Treatment 2</th>
<th>Conn Display</th>
<th>Gap (mm)</th>
<th>Disp. Treat</th>
</tr>
</thead>
<tbody>
<tr>
<td>FLG</td>
<td>Plain - 6 in</td>
<td>Plain - 6 in</td>
<td>Flanged</td>
<td>0.125</td>
<td>False</td>
</tr>
</tbody>
</table>

1. Click **Connections / Edit**.
2. Select the units of the Connection. Selecting Imperial or Metric will only show End Treatments of those units. Select Both if you want to create a connection between a Metric End Treatment and an Imperial End Treatment.
3. Click **Copy** to copy a selected connection, and then change any of the values.
4. Or click **New** to create a new connection with default values.
5. **Name** - The name for the connection, usually common Pipe terms such as FLG for flanged, BTW for butt weld are used. You can use any term you like.
6. **Treatment 1 & Treatment 2** - The treatments used in the connection. This is selected from a drop down list of defined End Treatments. The treatments may be the same or different. A Flanged connection would use a flange of...
the same size on both sides of the connection. A Screwed connection would be a combination of the male and female End Treatment.

End Treatments without extended data have the entry `<Regular>` otherwise the information in the three extended data fields is concatenated and presented as an End Treatment.

7. **Connection Display LineMode Icon** - Currently not implemented.

8. **Gap** - Specifies the distance between the ends of the two pipes or fittings. For example a butt weld connection might require a gap of 0.125” for welding purposes; a flanged connection might use a gasket of 0.25” thickness; a slip on flange has to slipped over the plain end of a pipe by -1.25”. Negative values indicate an insertion of one item into the other.

9. **Disp. Treats** - Allows the user to turn off the drawing of a Pipe entity’s End Treatments at the connection point.
Accessories

Accessories are additional items that are required for a connection. In the case of flange connections, these can be bolts, nuts washers, gaskets and more. It would be feasible to add into this also items such as weld length, assembly hours and similar. Please let us know how you would like to see this implemented.

On the left of the Dialog you can define new Accessory Types. On the right you can define new Accessory items. Accessory Types allow the grouping of accessories such as bolts, nuts, and flanges.

To add Accessories to a connection:

1. Make sure the desired connection is selected in the connections list.
2. Select the Type.
3. From the list select an accessory and click <<Add New. Add any other Accessories you want. Pressing <<Add New more than once for the same accessory will increment the quantity.
4. Select the Accessory in the Accessories list and specify the Quantity for each one.

Note: Accessories are not displayed in the Pipe drawing, but will be listed in BOMs and reports.

Catalog

The Catalog tab allows users to create and/or delete stock catalogs. Stocks (pipes/fittings), when initially created must belong to a “Catalog” just as the items purchased from a supplier are found in a catalog. By creating Catalogs you can reproduce your supplier’s catalogs or group them into a logical grouping such as Carbon Steel.

Each stock item can only belong to one Catalog. But, since similar stock items may be found in many catalogs, the Catalog tab allows stocks to be copied and moved to other Catalogs rather than needlessly reentering data. Correspondingly, stocks may also be deleted from Catalogs.
The catalog tab is divided into three main areas, Catalog functions, Catalog filter options, and Catalog stock list.

**Catalog Functions**

**Edit Catalogs...** - Opens the Edit Catalogs dialog from where Catalogs are created or deleted. Edit the Catalog names in the usual manner.

**Note:** You cannot delete catalogs which have stocks assigned to them. Delete the stocks first and then delete the Catalog.
**Copy to Catalog**… - Allows you to copy selected stocks to another Catalog.  
To copy stocks to a Catalog:  
1. Select the stocks to be copied.  
2. Click **Copy to Catalog**…  
3. In the drop list select the Catalog to which the stocks will be copied.  
4. Click **OK**.

**Move to Catalog**… - Allows you to move selected stocks to another Catalog.  
To move stocks to another Catalog:  
1. Select the stocks to be moved.  
2. Click **Move to Catalog**…  
3. In the drop list select the Catalog to which the stocks will be moved.  
4. Click **OK**.

**Delete from Catalog** - Deletes selected stocks.  

**Warning:** Deleting a stock from a Catalog deletes it permanently. If you’re not sure if the stock should be deleted, create a temporary Catalog first and then move the stock to the temporary Catalog rather than deleting it.
To Delete stocks from a Catalog:
1. Select the stocks to be deleted.
2. Click **Delete from Catalog…**

**Note:** If a stock is in use in a drawing, or included in a spec, it will not be deleted from the Catalog. A log file will be generated informing the user of which drawings or specs the stock is included.

---

**Catalog filter options**

The filter options allow for total customization of the stock list display. Through the filter options, stocks can be filtered by type and Catalog, and fields can be added or deleted from the display.

**To Filter by Catalog:**
1. In the Catalog drop-list, select the Catalog from which the stocks are to be displayed. You can choose to display stocks from all Catalogs or a specific one.

**To Filter by Stock Type:**
1. Select the **Filter by Pipe Types** radio button.
2. Check the box next to the stock type you wish to display. This may take a few seconds depending on the number of pipe stocks that exist.

**To Configure Columns:**
1. Select the **Configure Columns** radio button.
2. The list box shows all the properties available for all pipe types. Each entry in the list represents a column in the Stock List.
By default, all columns are selected regardless of the type selected. This means that if you only want to view all straight pipes, you likely don’t want to see columns such as Angle or End Type 3 and 4. Therefore select the fields you wish to display and the Stock List will be updated accordingly.

3. Columns can be ordered by dragging the column header into a new position. The blue mark indicates the position the column will be placed.

4. The contents of the Stock list can be sorted simply by clicking the header of the column containing the information you wish to sort by.

.Spec

Specs, or specifications, stipulate specific details for design, fabrication, and construction. In ShipConstructor - Pipe, Specs allow for grouping of catalog stocks that conform to a particular design specification (spec).
Use the specs to the advantage of your company. An example scenario is to create a spec for each system. Grouping pipes in this manner allows the pipe designers to easily select pipes and fittings for placement when working in a particular system. The following figure shows the such a scenario.
The Spec tab is very similar to the Catalog tab and can be broken down into three main sections:

1. Spec functions
2. Filter Options
3. Stock List

Additionally, there are some useful reporting functions such as:

- Listing the specs that a Catalog stock is included in.
- Listing the drawings that a Catalog stock is used in.

**Spec Functions**

**Edit Specs...** - Opens the Edit Specs dialog from where Specs are created or deleted. Edit the Spec names in the usual manner.

**Note:** You cannot delete a Spec that has Catalog Stocks assigned to it. Use the **Show Specs** reporting function to list all the specs the stock is included in.
Remove from Spec… - Allows you to remove selected stocks from a spec.

To remove stocks from a spec:
1. Select the Spec from which the stocks are to be removed.
2. Select the stocks to be removed from the spec.
3. Click Remove from Spec.

**Note**: Stocks cannot be removed from a spec if they are used in a drawing.

Assign to Spec… - Allows you assign stocks to a spec.

To assign stocks to a spec:
1. Select the stocks to be assigned to specs.
2. Click **Assign to Spec**…
3. In the list, select the Spec that the stocks will be assigned to.
Spec Filter Options

The filter options allow for total customization of the stock list display. Through the filter options, stocks can be filtered by type and Catalog, and fields can be added or deleted from the display.

To Filter by Catalog:
1. In the Catalog drop-list, select the Catalog from which the stocks are to be displayed. You can choose to display stocks from all Catalogs or a specific one.

To Filter by Spec:
1. In the Spec drop-list, select the Spec from which the stocks are to be displayed. You can choose to display stocks from all Specs or a specific one.

To Filter by Stock Type:
1. Select the Filter by Pipe Types radio button.
2. Check the box next to the stock type you wish to display. This may take a few seconds depending on the number of pipe stocks that exist.
To Configure Columns:

1. Select the **Configure Columns** radio button.

2. The list box shows all the properties available for all pipe types. Each entry in the list represents a column in the Stock List.

   By default, all columns are selected regardless of the type selected. This means that if you only want to view all straight pipes, you likely don’t want to see columns such as Angle or End Type 3 and 4. Therefore select the fields you wish to display and the Stock List will be updated accordingly.

3. Columns can be ordered by dragging the column header into a new position. The blue mark indicates the position the column will be placed.

4. The contents of the Stock list can be sorted simply by clicking the header of the column containing the information you wish to sort by.

Stock Report Functions

Two functions are available to help you investigate where a catalog stock is used. The first function, Show Specs, lists all the specs to which a stock belongs and the second function, Show Drawings, lists the drawings that use a spec’s stock.

To Create a Stock-Spec Report:

1. Select the stocks to perform the spec search on.
2. Click the Show Specs radio button in the Where Used frame.

3. Click the **Generate report** button and a report will be generated and output in Notepad.

---

**Import**

The Import tab contains functions that allow you to import pipe stocks from other databases.

Schematically, the import process works as shown below. All information required to define a pipe, including: Size Definitions, End Treatments, Penetrations, and Catalogs are all imported automatically.
As can be seen in the diagram above, Connections are separate objects from Pipes / Fittings. Importing pipes does not import the connections that are required to join the imported pipes and fittings. Therefore you must use the **Import Connections** function to add the Connections between pipes. Importing Connections will also import the connection’s accessories.

**Import Filter Options**

The filter options allow for total customization of the stock list display. Through the filter options, stocks can be filtered by type and Catalog, and fields can be added or deleted from the display.

**To Filter by Catalog:**

The Catalog list displays the Catalogs available in the source database.

1. In the Catalog drop-list, select the Catalog from which the stocks are to be displayed. You can choose to display stocks from all Catalogs or a specific one.
To Filter by Spec:
The Spec list displays the Catalogs available in the source database.

1. In the Spec drop-list, select the Spec from which the stocks are to be displayed. You can choose to display stocks from all Specs or a specific one.

   Spec:
   Balast

To Filter by Stock Type:

1. Select the Filter by Pipe Types radio button.

2. Check the box next to the stock type you wish to display. This may take a few seconds depending on the number of pipe stocks that exist.

To Configure Columns:

1. Select the Configure Columns radio button.

2. The list box shows all the properties available for all pipe types. Each entry in the list represents a column in the Stock List.

   By default, all columns are selected regardless of the type selected. This means that if you only want to view all straight pipes, you likely don’t want to see columns such as Angle or End Type 3 and 4. Therefore select the fields you wish to display and the Stock List will be updated accordingly.

3. Columns can be ordered by dragging the column header into a new position. The blue mark indicates the position the column will be placed.
4. The contents of the Stock list can be sorted simply by clicking the header of the column containing the information you wish to sort by.

To Import Pipe Stocks
1. Make sure the Catalog Editor is accessing the target database i.e. 1) You are registered to the project whose underlying database is to receive the stocks or 2) You are connected directly to the database.
2. Click the Select button in the Import from Database frame.
3. The Select Database dialog appears from which you can select the database with the pipe stocks to copy.

a) From the server drop down list, select the server hosting the source database.

Note: If you are running ShipConstructor on a machine with the Windows 98 operating system, you will have to manually enter the server name.

b) Once the server has been selected, the list of available databases within the server group will be available for selection in the Database drop list. Select the source Database from the drop list.
4. Select the pipes and fittings you wish to copy. Use the filter options to make it easier to select the stocks.
5. Once you have selected the stocks to import press the Import! Button. The process may take a few seconds depending on the number of stocks being imported.

To Import Connections
1. Make sure the Catalog Editor is accessing the target database i.e. 1) You are registered to the project whose underlying database is to receive the stocks, or 2) You are connected directly to the database.
2. Click the Select button in the Import from Database frame.
3. The Select Database dialog appears from which you can select the database with the pipe stocks to copy.
a)  From the server drop down list, select the server hosting the source database.

Note: If you are running ShipConstructor on a machine with the Windows 98 operating system, you will have to manually enter the server name.

b)  Once the server has been selected, the list of available databases within the server group will be available for selection in the Database drop list. Select the source Database from the drop list.

4.  Select the pipes and fittings you wish to copy. Use the filter options to make it easier to select the stocks.

5.  Once you have selected the stocks to import press the **Import!** Button. The process may take a few seconds depending on the number of stocks being imported.
Outfitting Reference

Overview

Outfit drawings are the first step to move from a 3D structural model to a 3D Product Model. A product model consists of structure, outfitting, Pipe, electrical, and any other type of item. To make a product model usable you require (besides the CAD elements) a relational database to keep track of all parts, changes and interrelations. Manager plays an integral role in moving you from 3D structural modeling to product modeling.

Outfit drawings are created and accessed using the Navigator.

Outfit Arrangement - New

This function creates a new outfit drawing. In the dialog enter the name of the new outfit drawing.
Outfit menu

The outfit menu consists of only one menu item. However, many other ShipConstructor functions can be used in the same manner as in other drawings. Many functions have been modified to achieve the desired results when using them while inside an outfit drawing.

Outfit - Insert Outfit

Menu: SC Outfit / Insert Outfit
Toolbar: None
Command line: SCINSOUTFIT
Navigator: None

Inserts an outfit part into the outfit drawing.

1. Select Insert Outfit.
2. In the dialog select the outfit standard you want to use. Click OK.
3. Select an insertion point for the outfit part.
4. Select a rotation for the outfit part.

5. The Part dialog is displayed. Assign the outfit part to an assembly, and assign paint if required. Click OK.

6. The part is added to the database and will show up in the reports, assembly drawings and will be considered in all weight calculations.

**Outfit - Check Outfits**

- **Menu:** SC Outfit / Check Outfits
- **Toolbar:** None
- **Command line:** SCOUTFITCHK
- **Navigator:** None

Checks Outfit parts to ensure the drawing they are in still exists.

**Outfit - OPM**

- **Menu:** right-click - Properties
- **Toolbar:** None
- **Command line:** None
- **Navigator:** None

The AutoCAD Object Property Manager (OPM) provides an easy to use and easy to access way of editing an object’s properties. For the ShipConstructor custom Outfit it displays some of the outfit’s physical properties such as Volume, CG and Name, and some of its editable properties such Hierarchy and Paint. Clicking on the Paint and Hierarchy buttons will bring up the Paint dialog and the Hierarchy dialog respectively allowing you to choose new values for these properties.
General Procedures

Outfitting Arrangement drawings can be done for individual assemblies, units or the complete ship. Each outfit item can be assigned to any structural assembly. Thus all reporting and assembly functions will consider the outfit items. It is now easy to generate 3D assembly drawings for structure and outfit, or generate purchase lists. You can have as many outfit drawings as you like.

You will need to display the structural group drawing to which the outfit item is attached in the outfit drawing. Use ShipConstructor’s XREF dialog for this.
Check the group drawings that you require. In this example two frames and the tanktop are loaded to place an outfit part.

In general you will activate a UCS onto which the outfit item will be placed. This can be a UCS within any one of the loaded drawings or one that you generate yourself, using ShipConstructor functions or AutoCAD functions.

Next, activate the layer(s) that you need on while inserting the outfit item.

You may want to draw reference lines identifying the position of the item to insert.

The outfit item is inserted as a custom ShipConstructor object, which looks up its information from the Outfit Standard drawing (similar to an XREF), thus keeping the outfit drawing small. Most outfit drawings will be just a few kilobytes in size, even when consisting of many outfit items.

You can use several standard AutoCAD commands to modify outfit items. These include:

- **ERASE** - Erases the outfit part and updates Manager.
- **ROTATE** - Rotates the outfit part and updates Manager.
- **MOVE** - Moves the outfit part and updates Manager.

Other AutoCAD functions will be intercepted and denied if they violate the Manager integrity.
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