



# Standards Manager User Guide

Standard Manager is a product of PROCAD software. It is designed for use with PROCAD SPOOLCAD's products.

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# About Standards Manager

PROCAD SPOOLCAD's Standards Manager allows you to create standards and organize shared standard files.

Using Standards Manager you can assign standard parameters including:

- Standard units
- Border settings
- Line numbers
- Active specifications
- Application settings
- User access

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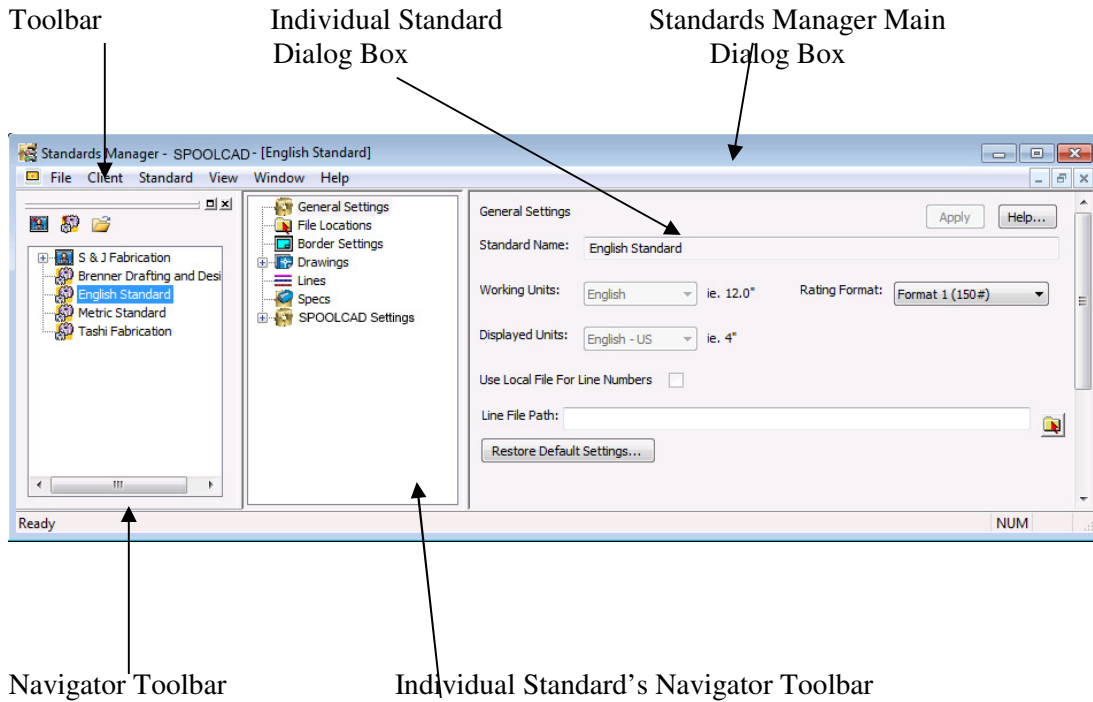
## Navigating the Standards Manager

The **Standards Manager Application** window is the main multi-document. You can open and view several standards in the child's window at the same time.

The Navigator Bar is a movable toolbar that displays clients and standards in a tree **structure**. Use this toolbar to open a desired standard.

The Standards Manager window (right side of the image below) displays information for the selected standard, which has its own navigation tool bar. You can view information for multiple standards or sub-projects in the data pane.

The toolbar, main menus, and context windows provide access to all software functions.



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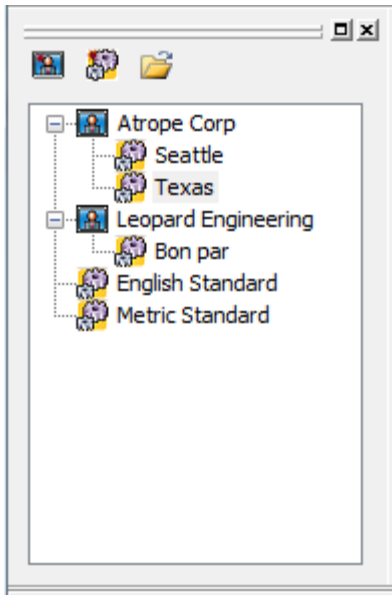
## Definitions

To effectively use the Standards Manager, it is important to understand the concepts of a Client and a unique Standard.

**Standard** One standard controls the appearance of one or more drawings. Each standard may have the same settings globally applied to all of its drawings through the standard manager.

**Client** One client consists of one or more standards.

This configuration of drawings, standards and clients can be seen in the example below. In this example, work is being performed on behalf of two companies, Atrope Corp and Leopard Engineering. At Atrope Corp, drawings are being produced for two plants while Leopard Engineering has only one location.



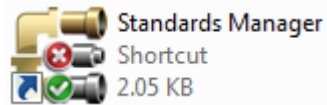
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## Starting Standards Manager

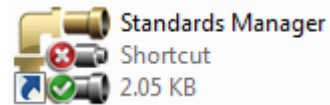
### *To start Standards Manager...*

In the PROCAD SPOOLCAD or PROCAD SPOOLCAD+ folder, double-click the Standards Manager icon.

#### SPOOLCAD



#### SPOOLCAD+



The **Standards Manager** dialog box opens.

Continue by opening an existing standard or creating a new Standard.

# Standards Manager Procedures

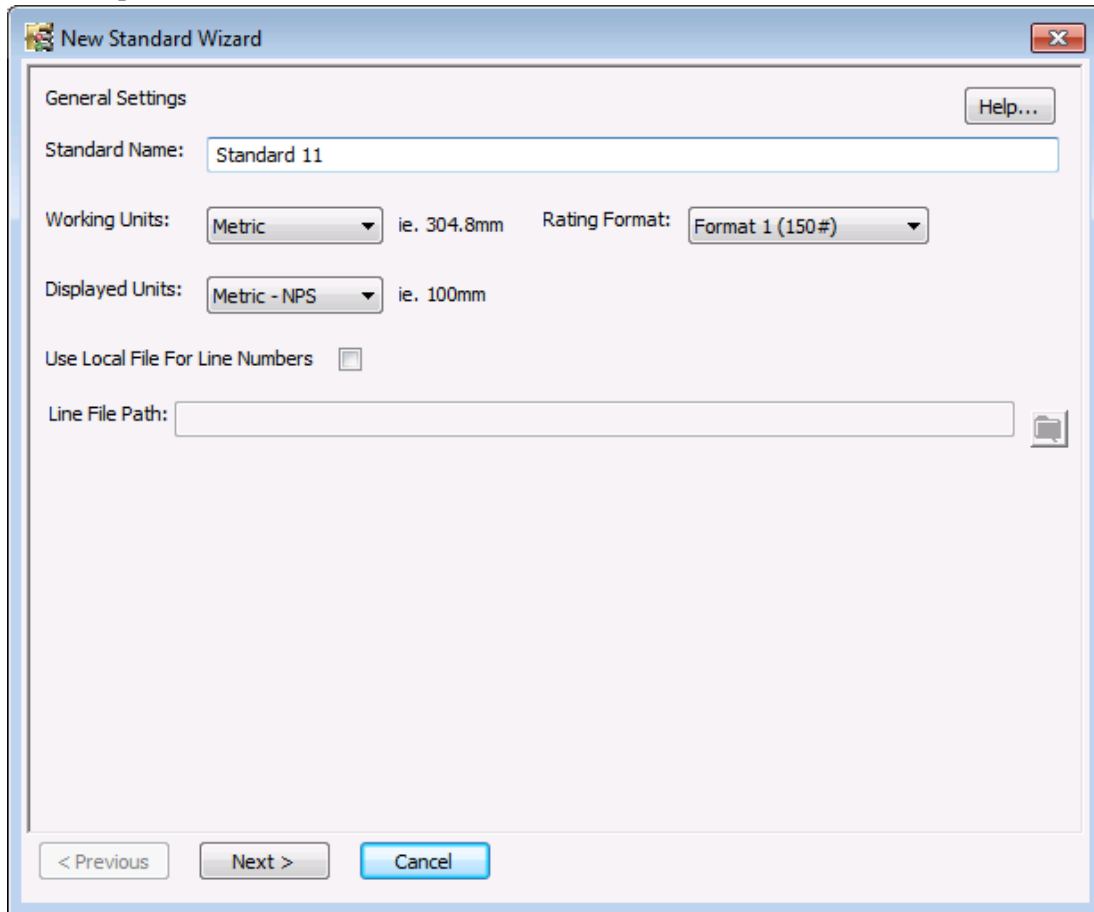
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## Creating a New PROCAD SPOOLCAD Standard

Use this procedure to create a new standard.

*To create a new standard....*

1. Click the New Standard button.  The **New Standard Wizard** dialog box opens.



The **New Standard Wizard** dialog box is shown. It has a title bar with a close button. The main area is titled "General Settings" and contains the following fields and controls:

- Standard Name:** A text box containing "Standard 11".
- Working Units:** A dropdown menu set to "Metric", with "ie. 304.8mm" displayed next to it.
- Rating Format:** A dropdown menu set to "Format 1 (150#)".
- Displayed Units:** A dropdown menu set to "Metric - NPS", with "ie. 100mm" displayed next to it.
- Use Local File For Line Numbers:** A checkbox that is currently unchecked.
- Line File Path:** A text box with a file explorer icon to its right.

At the bottom of the dialog box are three buttons: "< Previous", "Next >", and "Cancel". A "Help..." button is located in the top right corner of the main content area.

2. In the Name field, type a name for your drawing standard.
3. In the Working Units field, select the units you want the data to appear in, either English or Metric, from the drop down list.



- *Note:* Working units refers to the length and coordinates information used to place components.

**English:** to have length and coordinates information displayed in English units.

**Metric:** to have length and coordinates information displayed in metric units.

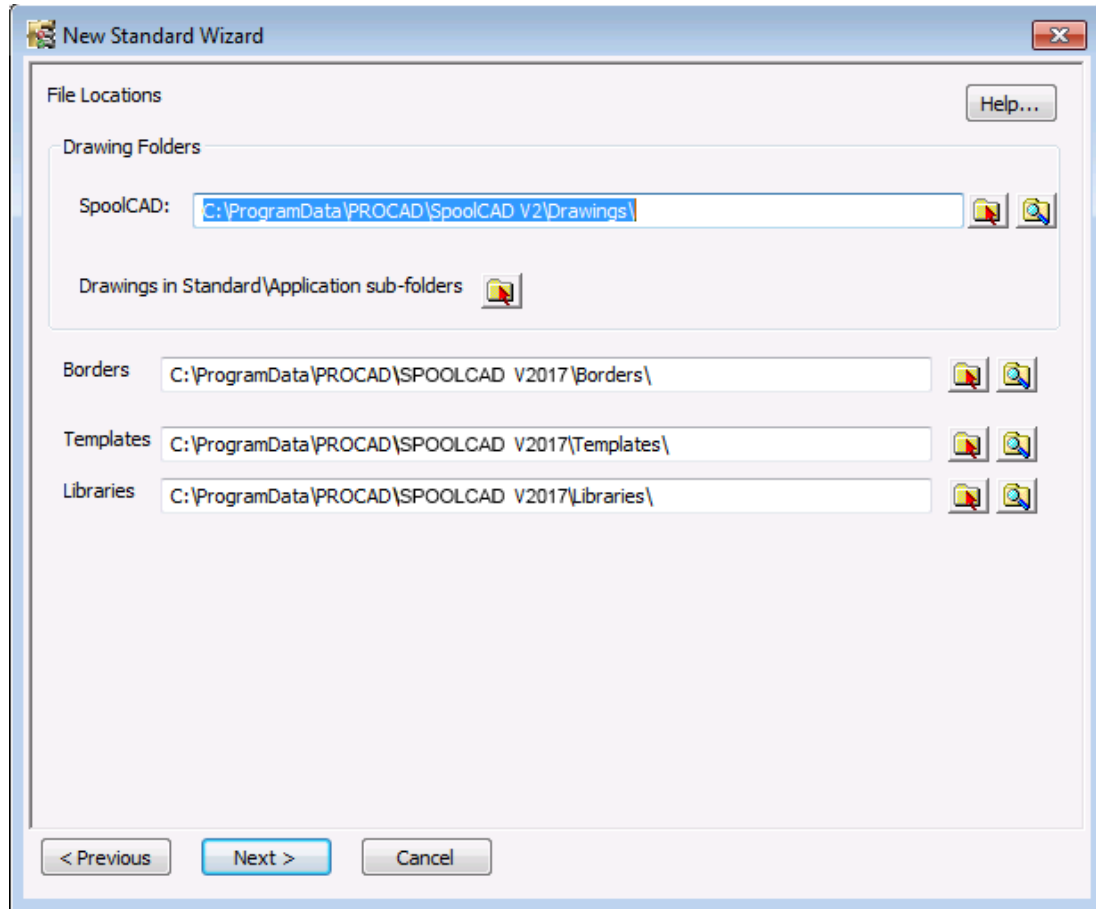
4. In the **Displayed Units** field, select the units you want the data to be displayed as from the drop down list: English US, English NPS, Metric OD or Metric NPS.

- **Note:** Displayed units refer to units that appear when selecting component sizes.
  - **English - US:** to have component sizes displayed in US English units (example, 4")
  - **English - NPS:** to have component sizes displayed in English units using standard nominal pipe size designations (example, NPS 4)
  - **Metric - OD:** to have component sizes displayed in metric units using standard pipe OD size designations (example, 114.3mm)
  - **Metric - NPS:** to have component sizes displayed in metric units using standard nominal pipe size designation (example, 100mm)

5. In the **Rating Format** field, select the display format for the ratings within the drawing standard. This format will affect the BOM output. For more information, please refer to the *Spec Generator User Guide*.
6. Some organizations may choose to restrict access to the datafiles, where the specs and standards settings are stored, to read-only for their users. In this case, the **Use Local File For Line Number** check box must be selected. This option also allows the line numbers to be accumulated on a per drawing basis, as opposed to an overall list for the entire Standard.

If this checkbox is active, the path of the line number \*.ini files may be defined. By default, this path is on the local users' machine in the c:\programdata\PROCAD\SPOOLCAD V2017\Lines folder.

7. Click Next.



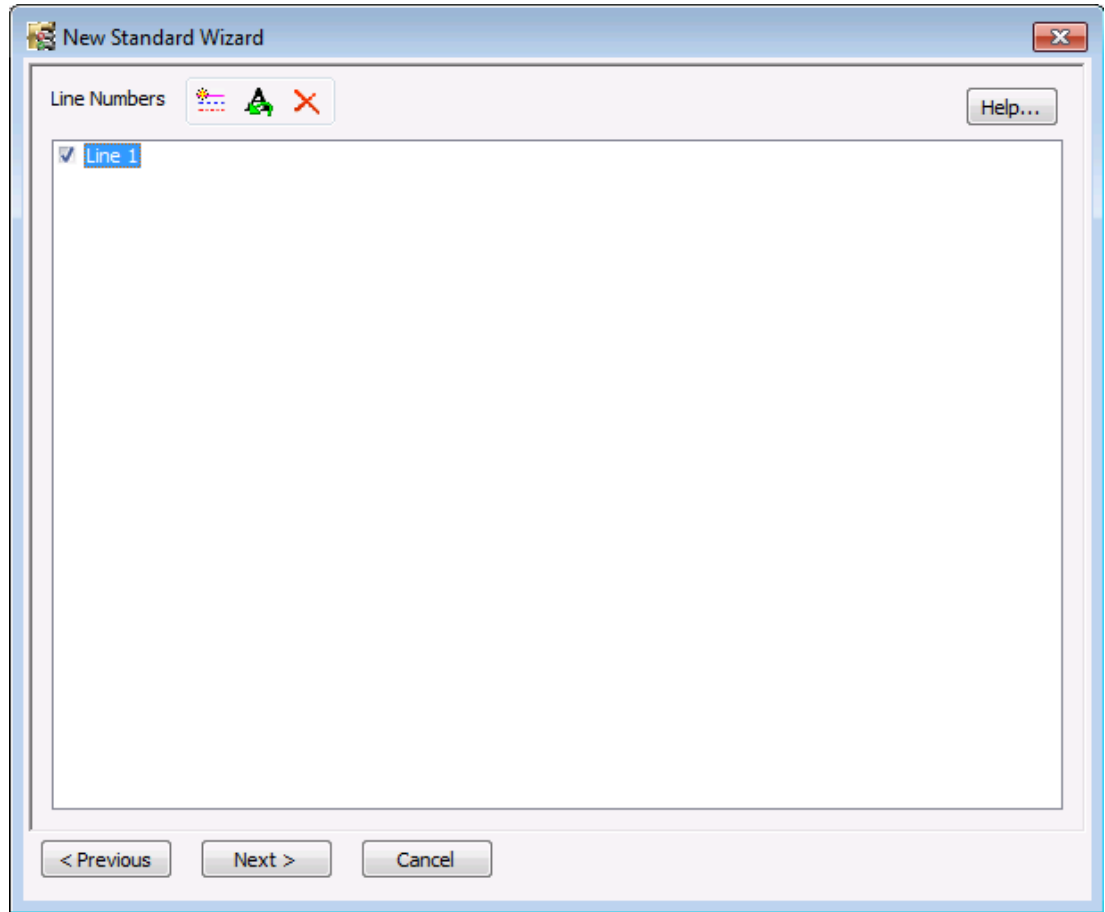
8. If you wish to open the drawings through the Drawing Manager in the applications, you need to define the drawing location.
- **Note:** The Drawing Manager will allow you to explore in any subfolders that exist below the folder location defined in this dialog box. Therefore, it is recommended that you select the highest level folder where all drawings that will use the drawing standard are located. For example, you may have a drawing folder structure such as:

- Client A
  - Project #A
    - SPOOLCAD drawings
  - Project #B
    - SPOOLCAD drawings

If all drawings under the Client A folder are to use the same drawing standard, then this folder should be used as the defined drawing location.

9. Define the location of your borders, if you intend to use the **Insert Border** command within the drafting applications.

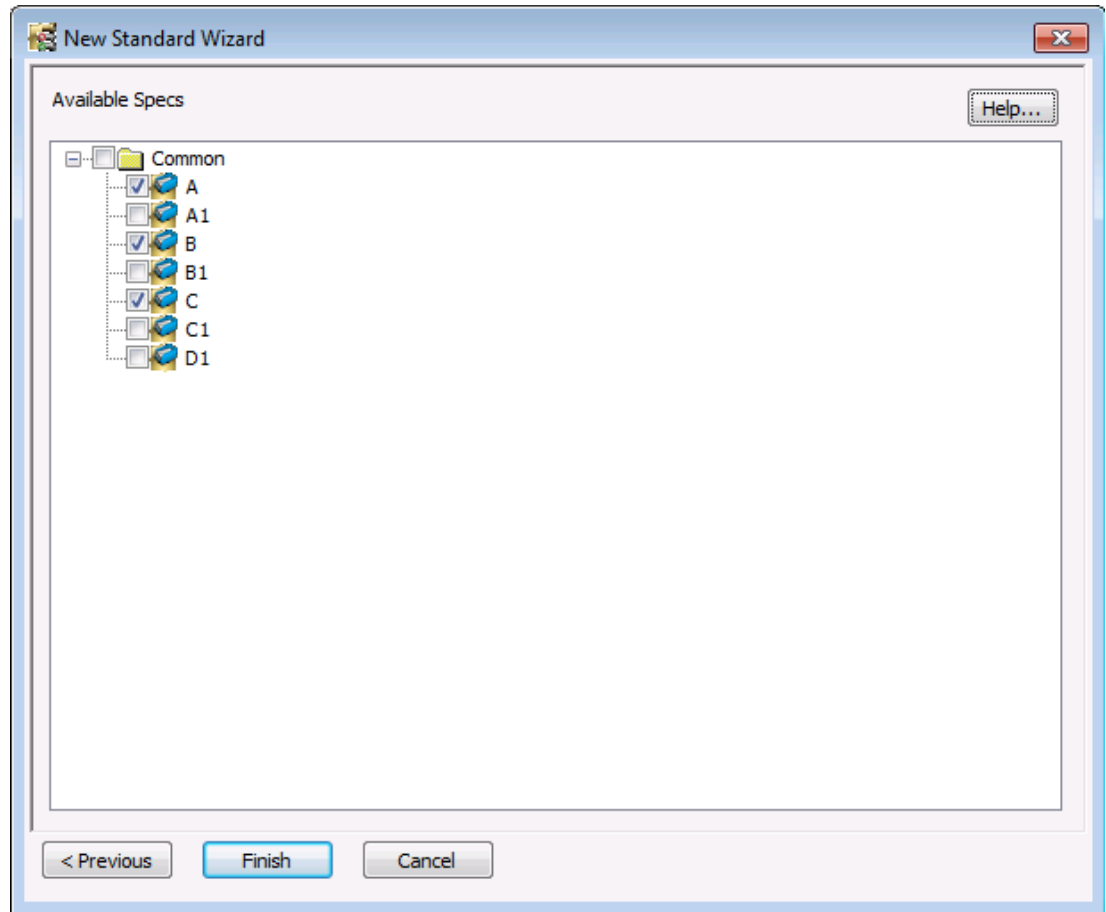
10. Define the location of any customized templates.
11. Click the **Next** button.
12. Click the **Next** button.



13. Create and check in line numbers required for the project. For more information, refer to *Line Numbers*.

Please note that this dialog will not appear if the **Use Local File For Line Number** checkbox is activated.

14. Click **Next**.



15. Check in any specifications that are required for the standard. For more information, see *Available Specs* and the *Spec Generator User Guide*.

16. Click **Finished**.

The new Standard will now appear in the Navigator bar to the left hand side

---

## Opening an Existing Standard

Use this procedure to open a previously created drawing standard.

### *To open a standard...*

1. Click to highlight the drawing standard you want to open.

2. Click the **Open Standard** button. 

-or-

Select **Open** from the **Standard** menu

-or-

Right-click and select **Open** from the context menu.

The standard opens in the Standard Manager window.

---

## Deleting a Standard

### *To delete a standard...*

1. In the Standard Manager window, click to highlight the drawing standard you want to delete.
2. Right-click and select **Delete** from the context menu.

-or-

Select **Delete** from the **Standard** menu.

- **Note:** Clicking **Delete** only removes the standard from the directory tree. Once you have deleted the standard, the **Purge** option becomes enabled, which removes the standard from the database entirely.

---

## Restoring a Standard

### *To restore a standard...*

1. Right-click in the Standard Manager's window and select **Show Deleted** from the right-click context menu.

-or-

Select **Show Deleted Standards** from the **View** menu

All of the standards that have been deleted appear in the standard directory. They are distinguished by having an icon with a grey folder and a red "x".

2. Right-click again on the standard that you want to restore. Select **Recover** from the right-click context menu.

-or-

Select **Recover** from the **Standards** menu.

The standard is restored to the standard directory tree.

---

## Purging a Standard

### *To restore a standard...*

1. Right-click in the **Standard Manager's** window and select **Show Deleted** from the right-click context menu.

-or-

Select **Show Deleted Standards** from the **View** menu

All of the standards that have been deleted appear in the standard directory. They are distinguished by having an icon with a grey folder and a red “x”.

2. Right-click again on the standard that you want to restore. Select **Purge** from the right-click context menu.

-or-

Select **Purge** from the **Standards** menu.

The standard is restored to the standard directory tree.

---

## Renaming a Standard

### *To rename a standard...*

1. In the Standard Manager window, click to highlight the drawing standard you want to rename. Right-click on the Standard and select **Rename** from the context menu.

-or-

Select **Delete** from the **Standard** menu.

The standard is renamed in the standard directory tree.

---

## Copying a Standard

When copying a standard, only the settings for the particular standard are saved, not the drawings. This is a quick way to start a new standard using another standard's outline.

➤ **Note:** After a copy of a standard has been made, the user cannot change the units in the copied standard.

### *To copy a standard...*

In the standard directory tree, either click to highlight the standard you want to copy and click the **Copy** button or right-click on the standard and choose **Copy** from the right-click menu.

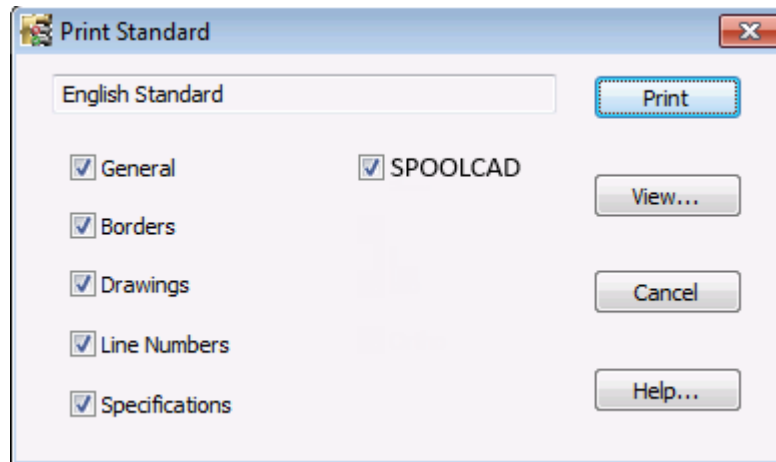
The new copy of the standard will appear in the standard directory tree.

## Printing a Standard

### *To print a standard...*

1. In the standard directory tree, either click to highlight the standard you want to print and click the **Print** button or right-click on the standard and choose **Print** from the right-click menu.

The **Print Standard** dialog box appears.



2. Select the components of the standard you want to print.

Click **Print** to print the standard.

Click **View** to see what the standard will look like before it's printed.

Click **Cancel** to exit the **Print Standard** dialog box without printing.

Click **Help** to access the help files.

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## Comparing Two Standards

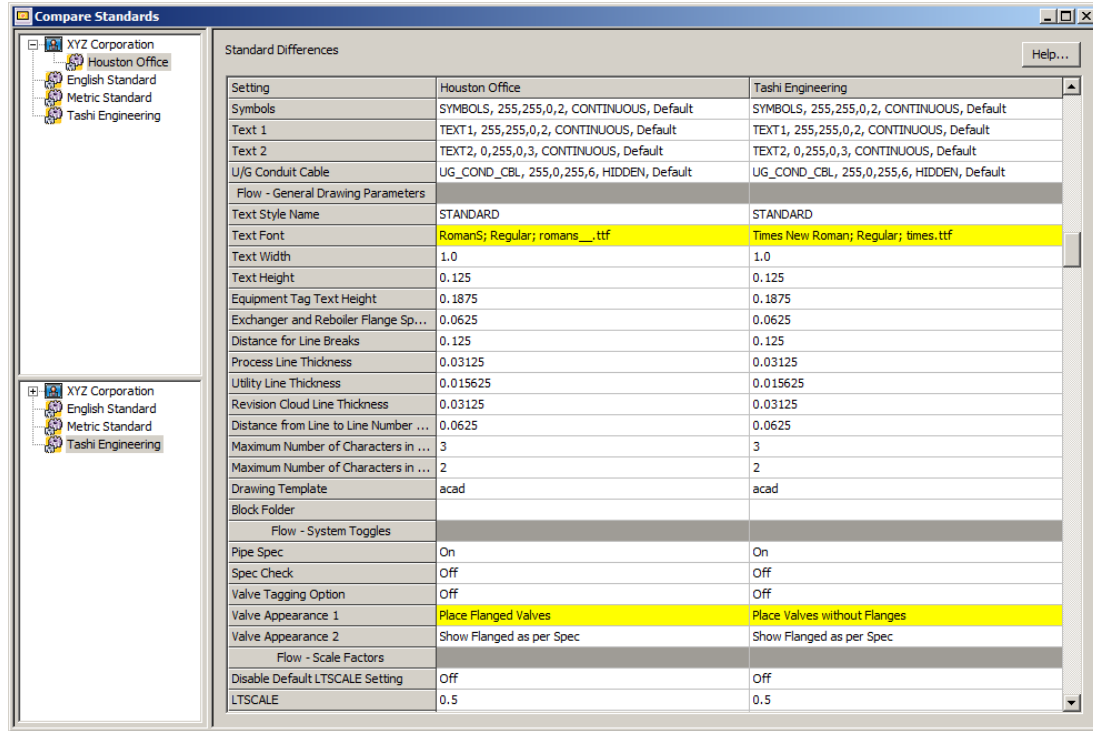
### *To compare two Standards....*

1. From the **Standard** menu select **Compare**.

-or-

Right click on the Standard name and select **Compare** from the context menu

2. The **Compare Standards** dialog appears.



3. In the panes on the left, select two different Standards that you wish to compare.
4. In the pane on the right, the differences between the two Standards will be highlighted in yellow. Use the scroll bar on the far right to examine all settings in the Standards.

## Exporting a Standard

### *To export a Standard....*

1. From the **Standard** menu select **Export**.
- OR-
- Right click on the Standard name and select **Export** from the context menu
2. In the **Save Standard As** dialog, navigate to the location that you wish to save the exported file.
3. Enter the name of the exported file.
4. Click the **Save** button.



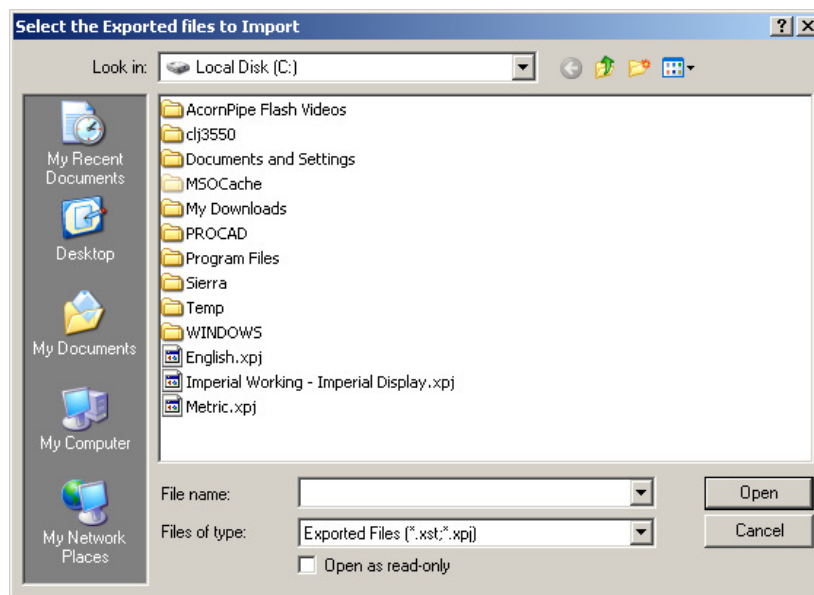
## Importing a 2D DESIGNER 2010 or PapriCAD Standard

*To import 2D DESIGNER 2010 or PapriCAD Standard....*

1. From the **Standard** menu select **Import**.

-or-

Right click on the Standard name and select **Import** from the context menu.




2. In the **Select the Exported files to Import** dialog, navigate to the location of the \*.xst or \*.xpj file.
3. Highlight the file name.
4. Click the **Open** button. The Standard will now appear in the navigation toolbox.

---

## Creating a New Client

*To create a new client....*

1. Click the **New Client** button  or use the **New** command from the **Client** menu.
2. In the **Name** field, type a name for your standard.
3. Add standards to your client.

## Deleting a Client

*To delete a client...*

1. Right click on the client you wish to delete and select **Delete** from the dialog box or use the **Delete** command from the **Client** menu.
2. Confirm the deletion by clicking **Yes**.
  - **Note:** This will not delete any standards that are contained within the client.


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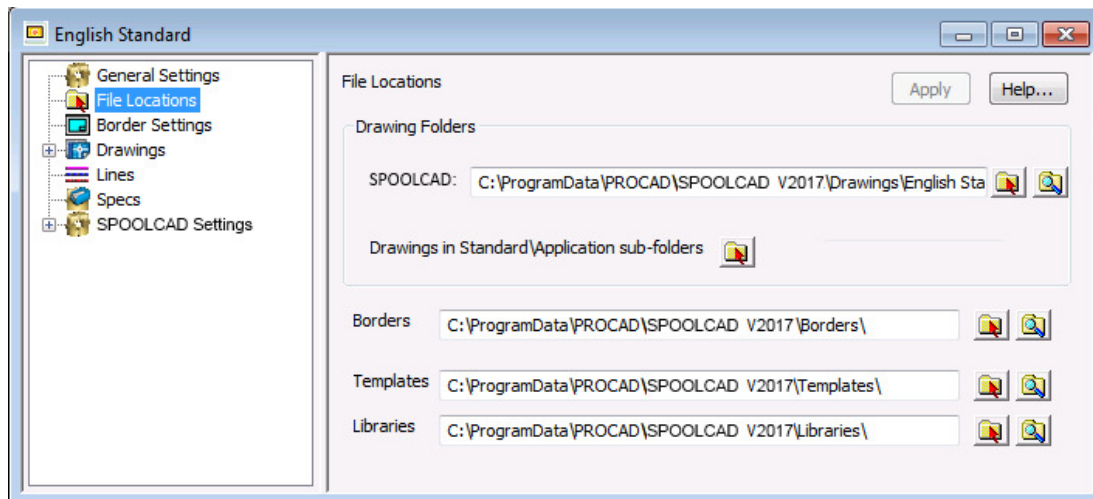
## Customizing Dimensional Files

Some customers may wish to customize the dimensional data that is used by the applications. These files are by default located in the Data Files directory in the Libraries directory. A complete copy of this directory must be made before any data is customized.

*To edit the directory to use custom dimensional data...*

1. Open the standard where you want to edit the border.
  2. In the Standards Manager window, click the File Locations icon.

The **File Locations** dialog box opens.
  3. Click the  icon to open a Windows Explorer dialog box to navigate to the custom dimensional file location.
- OR-
- Enter in the file path of the custom dimensional file location.



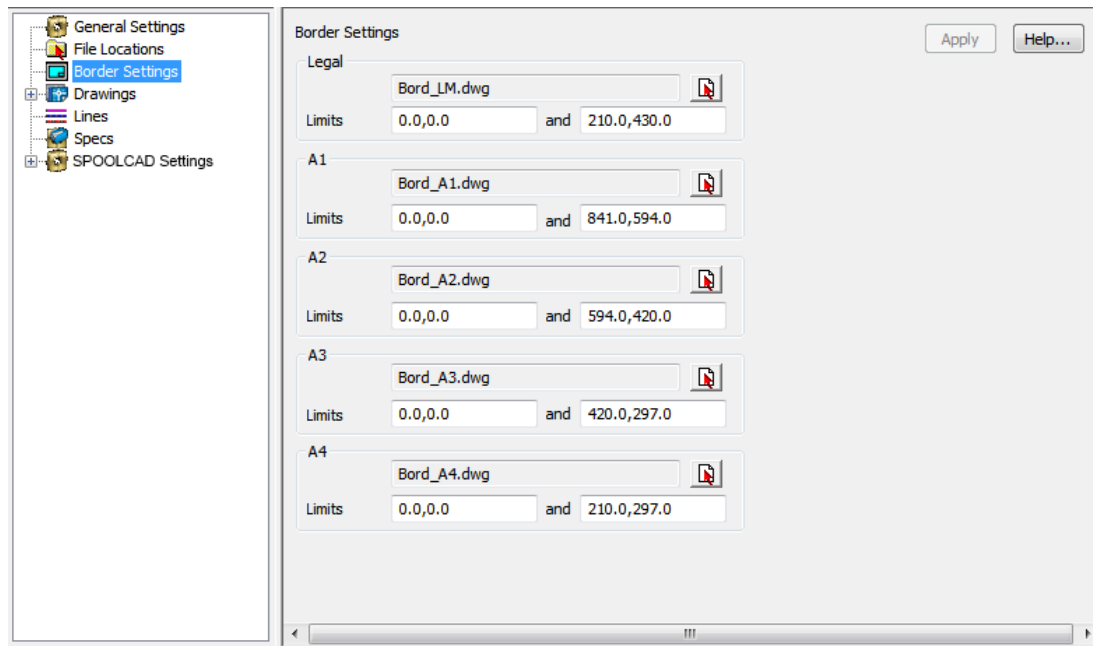
## Drawing Borders and Limits

You can change the border values by opening a standard and double clicking on the **Borders** icon.

*To edit the borders and limits used...*

1. Open the standard where you want to edit the border.
2. In the Standards Manager window, click the Borders icon.

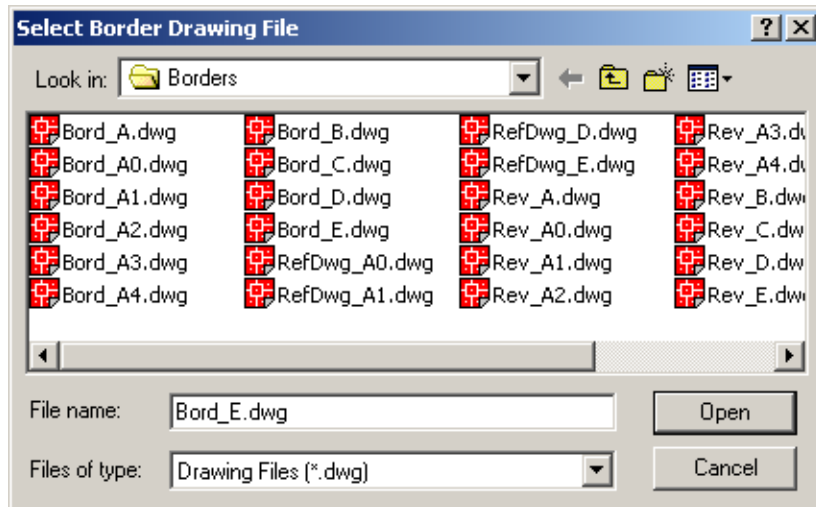
The **Borders Settings** dialog box opens. It displays all paper sizes, the corresponding border drawing and two limits fields.



*To assign border block drawing files...*


1. Ensure that the border drawing files are correct.
  - **Important:** You must save the border drawing files in the same folder you select for the border file.

To choose a different file, click the **Select Border File** icon and the **Select Border Drawing File** dialog box will appear.



2. Select the appropriate file and do one of the following:
  - Click **Open** to insert the selected border file and close the dialog box.
  - Click **Cancel** to close the dialog box without saving the changes.
  - **Note:** Do not browse to a different folder to select a border.

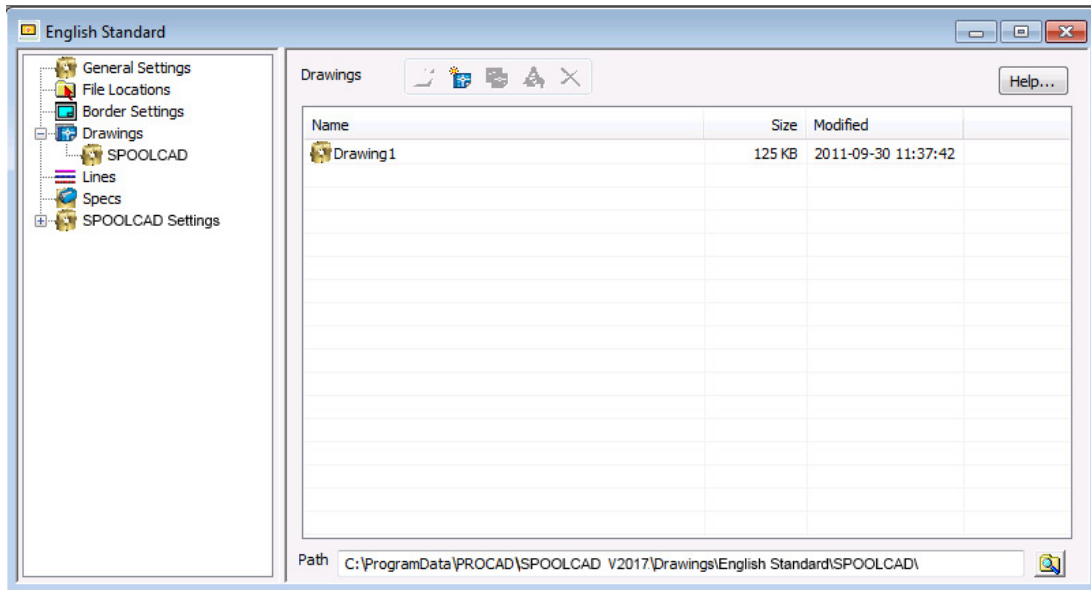
### *To change limits...*

1. In the Borders dialog box, ensure the limit values for each paper size are correct.  
To change limit values, select the data in the cell, then type in a new value.
  - **Important:** The system does not validate data entered in these cells.  
Ensure you enter the correct numerical value.
2. When all limits information is complete, do one of the following:
  - Click **Apply** to save your changes, and keep the dialog box open.
  - Click the **Close** button  to exit the dialog box without saving changes.

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## Drawing Management

In the **Drawings** dialog box, the user can view information about a specific drawing.



A few things to note about the **Drawings** dialog box:

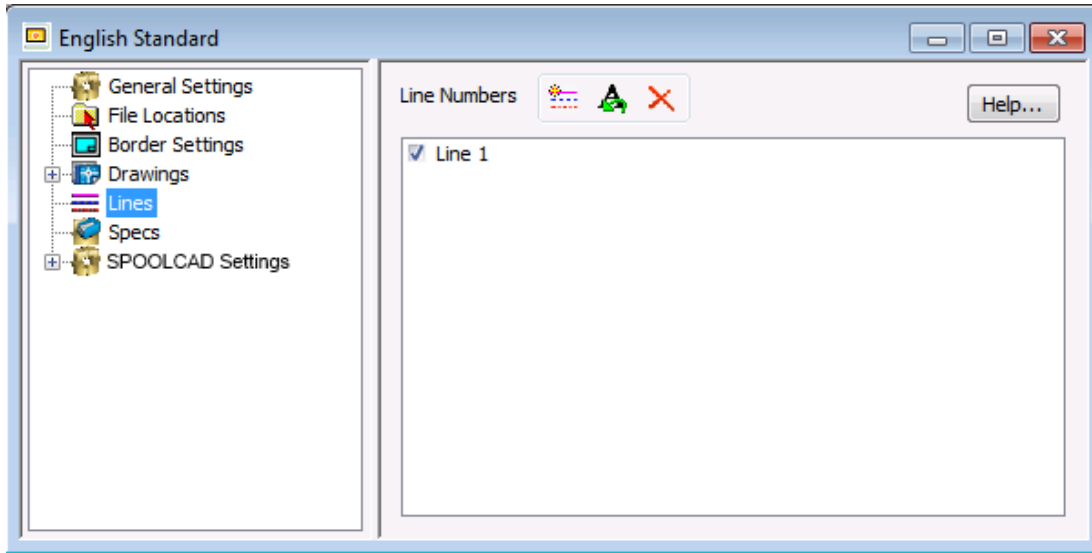
- If you click on the Drawings icon in the Standards Manager, a list will display of all the drawings for all the applications.
- When you click on the Drawings icon, a list of the applications will appear. Click on the application to view its list of drawings.
- You have to be viewing the drawings for a specific application before the **New** button will become enabled (for creating a new drawing).
- When you click the **Delete** button, the selected drawing will be removed permanently from the computer and will not be able to be recovered.

---

## Line Numbers

In the **Line Numbers** dialog box, you can create new line numbers and select which lines you do or do not want to use in the standard.

This dialog box will not be available if the **Use Local File For Line Number** checkbox is activated on the **General Settings** dialog box.



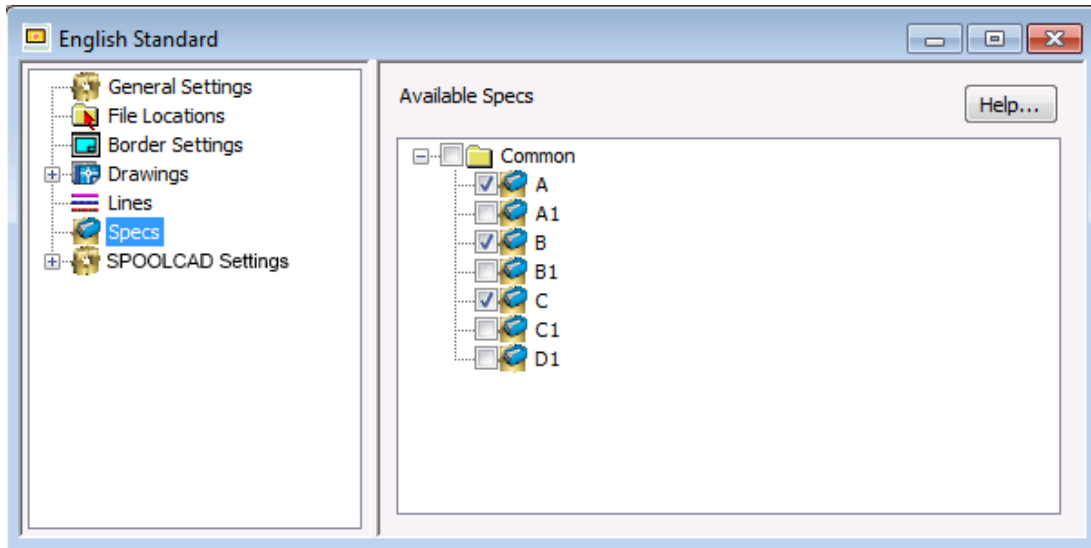
A few things to note about the **Lines** dialog box:

- Users will be unable to use or see the line in the standard unless it is checked off.
- To create a new line, purge, or rename a line, use the buttons at the top of the dialog box.
- When you purge a line, it will be permanently removed from the database and you will be unable to recover it.
- In order for the **Purge** button to be enabled, the line must be unchecked.

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## Available Specs

In the **Specs** dialog box, you can choose which specs you do or do not want to use in the standard.

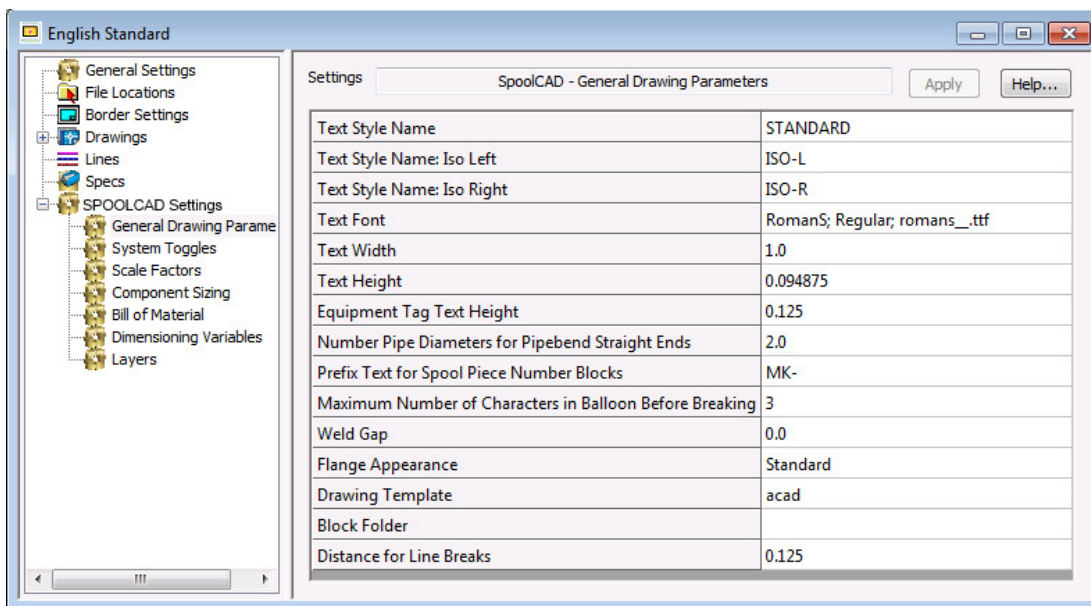


A few things to note about the **Specs** dialog box...

- To do any modifications to a spec, you have to use the **Spec Generator**.
- The user cannot use specs until they are checked off in the list.

## PROCAD SPOOLCAD Application Settings

This dialog box includes PROCAD SPOOLCAD application. Each dialog box displays the settings for each property or attribute.



A few things to note about the **Designer Applications**...

- Each application has a **Drawing Template** drop-down list. The template must be located in the template path that is defined on the *File Locations Dialog Box*.
- Whichever template is selected in the list when a new drawing is created will be the template for that drawing.



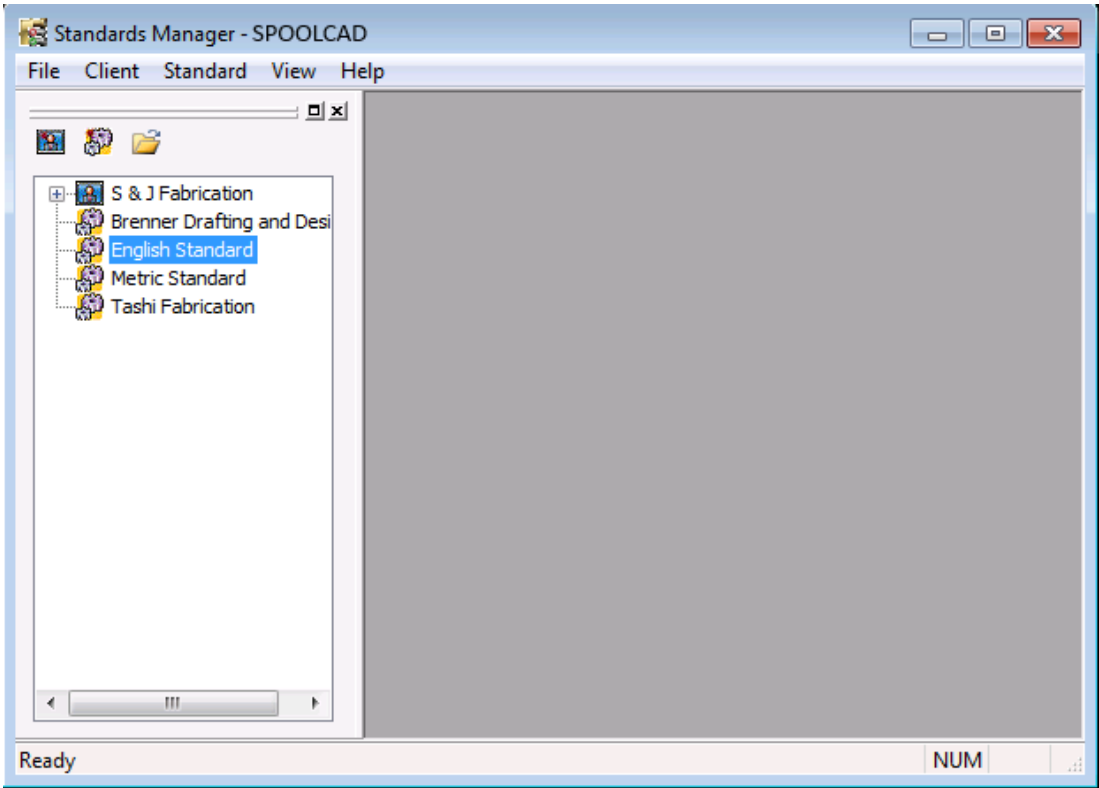
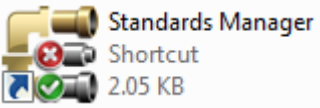
# Dialog Boxes

## Standard Manager Dialog Box


Use this dialog box to open an existing standard, create a new standard, or edit standard parameters.

*To access this dialog box...*

In the PROCAD SPOOLCAD or PROCAD SPOOLCAD+ folder, double-click the **Standards Manager** icon.



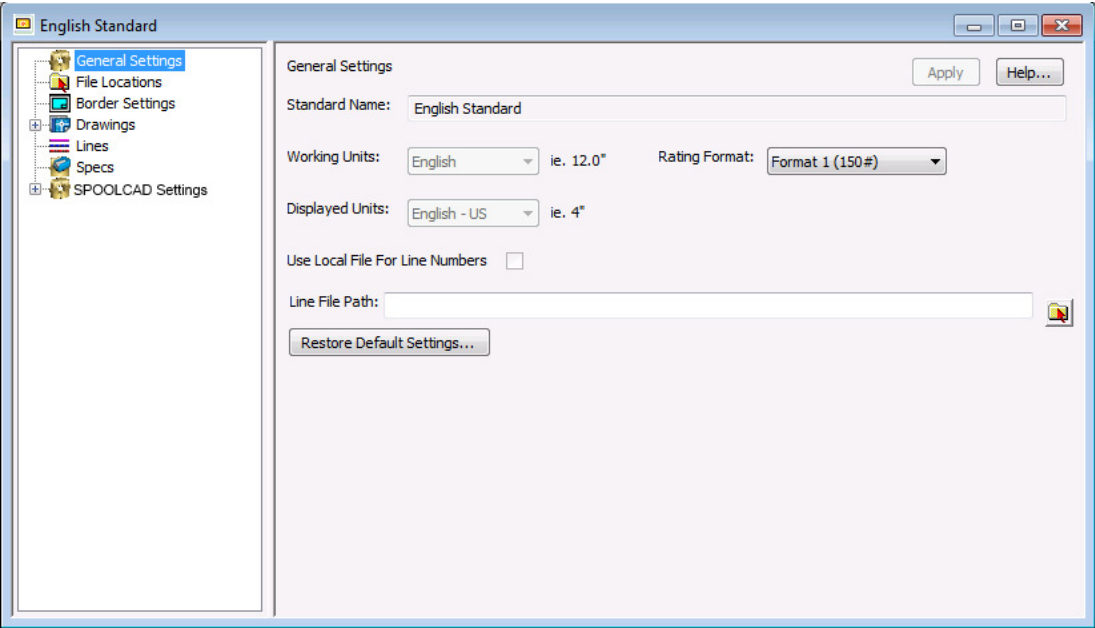
Click this button...	To...
	Click to create a new client.
	Click to create a new standard.

	Click to open the <b>Open</b> button an existing standard.
---	--

## General Settings Dialog Box

*To access this dialog box...*

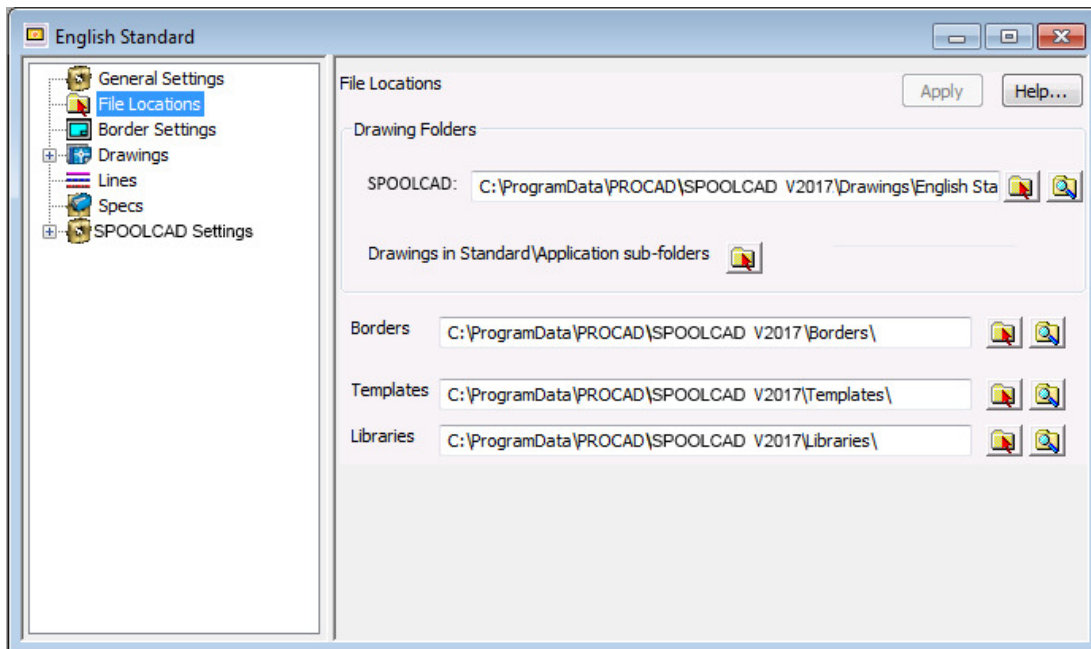
In the **Standard Manager** Dialog box, click the **General** icon.















In this field...	Enter/Do this...
Standard Name	Displays the name of the currently selected standard.
Displayed Units	Displays the units selected for component sizes.
Working Units	Displays the units selected for length and coordinates information.
Rating Format	Displays the format for the ratings within the drawing standard. This format will affect the BOM output. For more information, please refer to the <i>Spec Generator User Guide</i> .
Use Local File For Line Numbers	Some organizations may choose to restrict access to the datafiles, where the specs and standards settings are stored, to read-only for their users. In this case, the <b>Use Local File For Line Number</b> check box must be selected. This option also allows the line numbers to be accumulated on a per drawing basis, as opposed to an overall list for the entire Standard.
Line File Path	If the Local File For Line Numbers checkbox is

	active, the path of the line number *.ini files may be defined. By default, this path is on the local users' machine in the c:\programdata\PROCAD\SPOOLCAD V2017\Lines folder.
Restore Default Settings button	Click to change the settings of the standard back to what they initially were.
Help...	Click to access the <b>Help</b> dialog box.

## File Locations Dialog Box



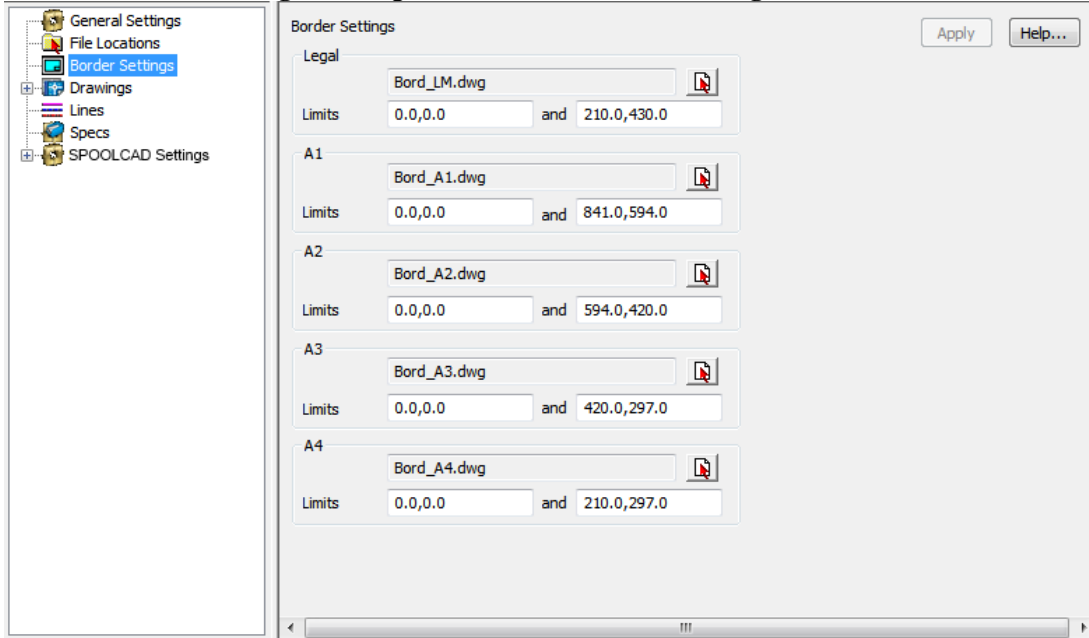
In this field...	Enter/Do this...
PROCAD SPOOLCAD Drawing Folders	All Isometric drawings that will use this drawing standard can be found in this defined location or in any subfolders. Use the  button to change the folder location. Use the  button to view the contents of the currently defined folder.
All drawings in one folder 	Click to set the drawing folder location for all four applications.
Drawings in Standard\Application sub-folders 	Click to set the drawing folder location for all four applications at the same time. The user will select the location, and the Standards Manager will create a folder structure "Standard\Application" for all paths. For

	example, if the user selects the c:\ drive for the Houston Office standard, the P&ID drawing path will be c:\Houston Office\Flow.
Borders	All border drawings that will be used on the Borders dialog box must reside within this folder. Use the  button to change the folder location. Use the  button to view the contents of the currently defined folder
Templates	All template files that you wish to use for any of the applications must be located in this folder. Templates may be set on the General Drawing Parameters dialog for each of the applications. Use the  button to change the folder location. Use the  button to view the contents of the currently defined folder.
Libraries	All dimensional files that you wish to use for any of the applications must be located in this folder. Use the  button to change the folder location. Use the  button to view the contents of the currently defined folder.
	Click to apply the changes.
	Click to access the <b>Help</b> dialog box.



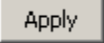

## Border Settings Dialog Box

*To access this dialog box...*

In the **Standard Manager** Dialog box, click the **Border Settings** icon.



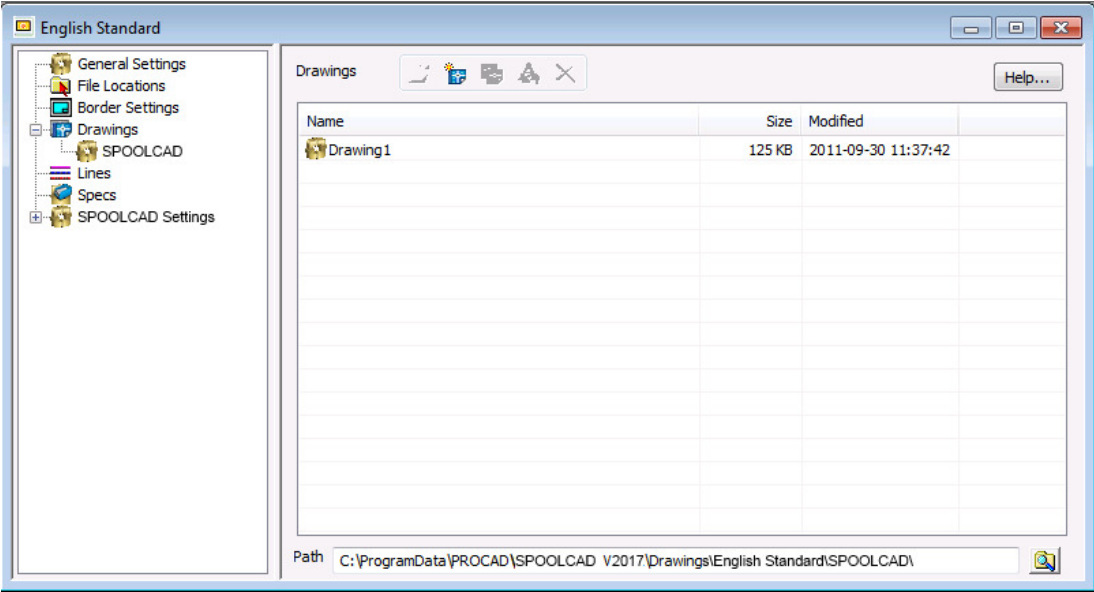
Use this dialog box to assign drawing borders and limits for all paper sizes.

In this field...	Enter/Do this...
Drawing Borders and Limits (grey cells)	These display the currently selected border drawing file for each paper size. To change these files, click the corresponding Change File button. 
Drawing Borders and Limits (white cells)	Enter user-defined limits for drawings. <b>Note:</b> The system does not verify this data. Ensure you enter the correct <b>numerical</b> data.
Apply button	Click this button to save changes and keep the dialog box open.
Close button 	Click to exit the dialog box without saving changes.
	Click to apply the changes.
	Click to access the <b>Help</b> dialog box.






# Drawings Dialog Box

*To access this dialog box...*

In the **Standard Manager** Dialog box, click the **Drawings** icon



Use this dialog box to manager your drawings.

In this field...	Enter/Do this...
	Click to create a new drawing.
	Click to copy the drawing.
	Click to rename the selected drawing.
	Click to delete the selected drawing.
	Click to access the <b>Help</b> dialog box.

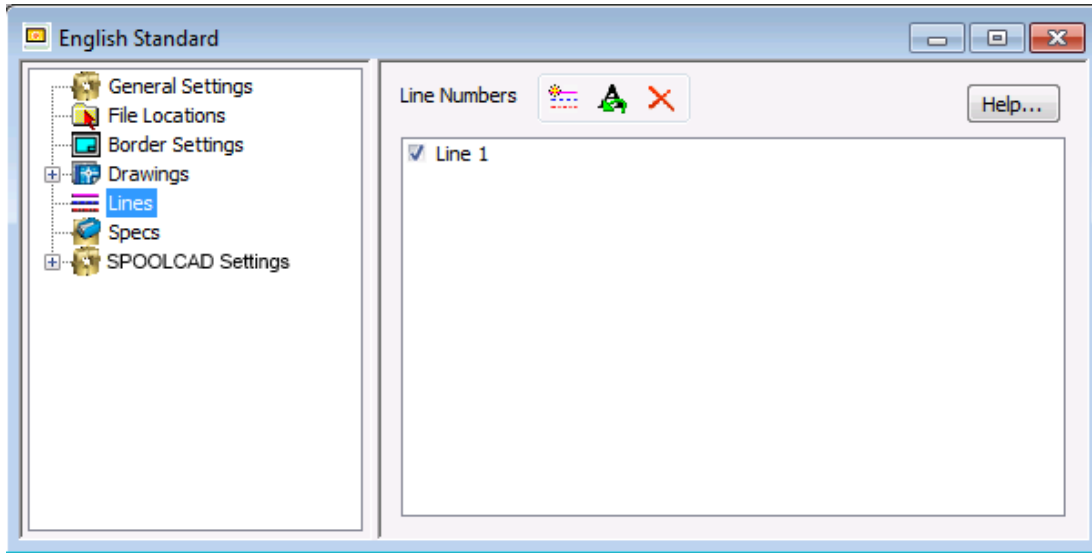
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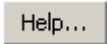
# Lines Dialog Box

Please note that this dialog will not appear if the **Use Local File For Line Number** checkbox is activated.

*To access this dialog box...*

In the **Standard Manager** Dialog box, click the **Lines** icon.

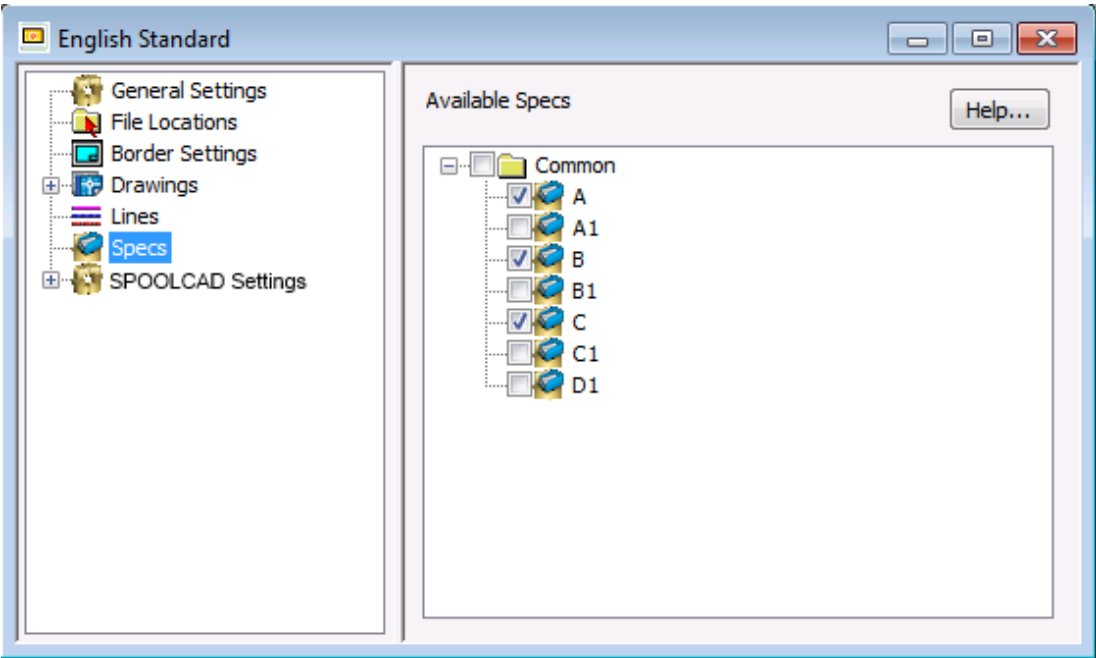


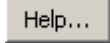
In this field...	Click/Do this...
	Click to create a new line.
	Click to rename a line.
	Click to delete a line.
Line check box	Checked lines will be available for use when the selected drawing standard is loaded.
	Click to access the <b>Help</b> dialog box.

## Specs Dialog Box

*To access this dialog box...*

In the **Standard Manager** Dialog box, click the **Specs** icon.



In this field...	Click/Do this...
Spec check box	Checked specs will be available for use when the selected drawing standard is loaded. Checking the box beside a folder name will include all specs from that folder.
	Click to access the <b>Help</b> dialog box.

## PROCAD SPOOLCAD Settings Dialog Boxes

The settings for PROCAD SPOOLCAD have been divided into seven areas:

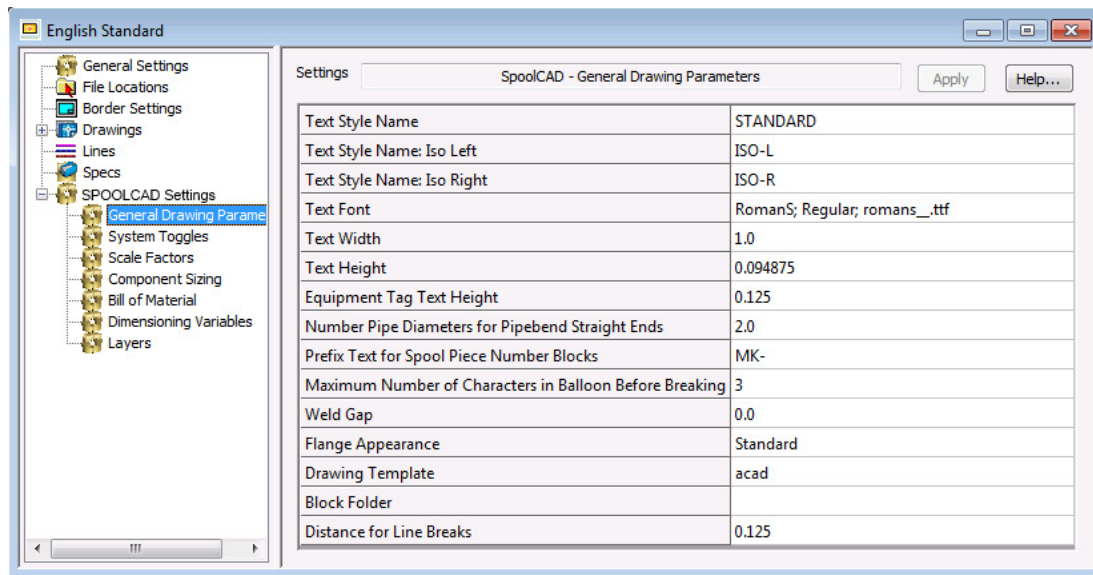
- General Drawing Parameters
- System Toggles
- Scale Factors
- Component Sizing
- Bill of Material
- Dimensioning Variables
- Layers


*To access this dialog box...*

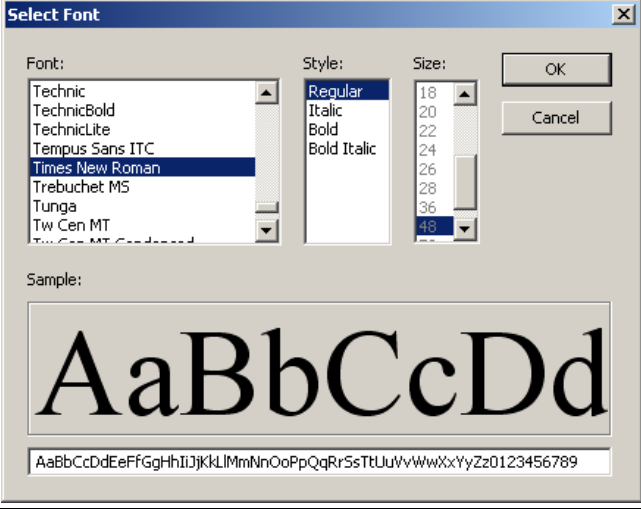

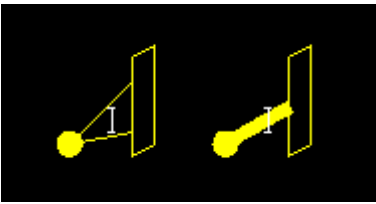
In the **Standards Manager**, click the **Iso Settings** icon.



## General Drawing Parameters Dialog Box

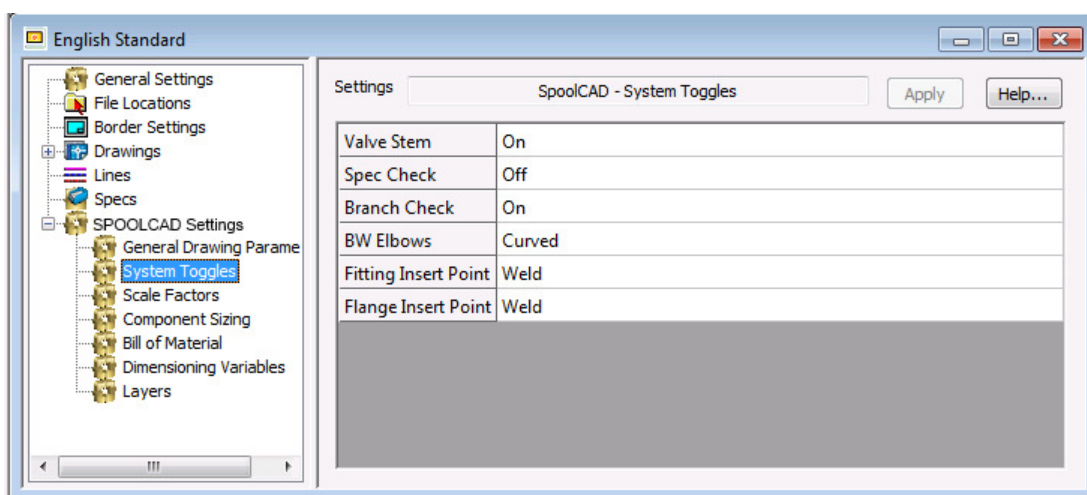


In this field...	Enter/Do this...
Text Style Name	The text style that will be used by default for all PROCAD SPOOLCAD text, such as Equipment Tags. If the text style does not exist in the drawing template, the layer will be created based on the Text Font information provided.
Text Style Name: Iso Left	The text style that will be used by default for all PROCAD SPOOLCAD text, such as Equipment Tags. If the text style does not exist in the drawing template, the layer will be created based on the Text Font information provided.
Text Style Name: Iso Right	The text style that will be used by default for all PROCAD SPOOLCAD text, such as Equipment Tags. If the text style does not exist in the drawing template, the layer will be created based on the Text Font information provided.
Text Font	The font type, style, and size that will be associated with the text style name. To change these properties, highlight the field, and click the  to the right of the field. This will open up the Select Font dialog box.

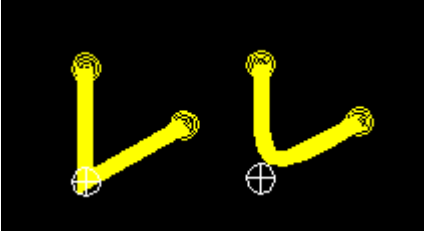
	
Text Height	The height of all text placed by PROCAD SPOOLCAD routines with the exception of Equipment Tags and dimensioning text
Text Width	The width of all text placed by PROCAD SPOOLCAD routines with the exception of Equipment Tags and dimensioning text
Equipment Tag Text Height	The height of the text that will appear when the Equipment Tag routine is used
Number Pipe Diameters for Pipebend Straight Ends	When a pipe bend is placed, the dialog box will automatically use this value as the # Straight End Pipe Diameters value. For a 4" NPS, a value of 2 for this parameter will result in straight ends that are 8" in length.
Prefix Text for Spool Piece Number Blocks	When the Spool Mark Number routine is used, this parameter will define the default prefix text for the spool number.
Maximum Number of Characters in Balloon Before Breaking	<p>If the number of characters in the balloon exceeds this value, the balloon sides will automatically break to accommodate the width of the text.</p>  <p>This applies only to Instrument balloons.</p>
Flange Appearance	<p>When this option is set to Standard, flanges will appear as shown on the left hand side. When the option is set to Line Width, flanges will appear as shown on the right hand side.</p> 
Drawing Template	This is the AutoCAD template that will be used for each new PROCAD SPOOLCAD drawing. Customized templates may be used to include specific layers, dimension styles or text styles into each drawing. The template must be located in the template path that is defined on the <i>File Locations Dialog Box</i> .
Block Folder	This is the location of the AutoCAD block files (*.dwg) that is used by the program. By default, this value is blank and the blocks are located in

	the default program install location. This path should use absolute server names, not mapped drive letters.
Distance for Line Breaks	This is the area where you can specify the break distance when pipes intersect.
Apply	Click to apply the changes.
Help...	Click to access the <b>Help</b> dialog box.

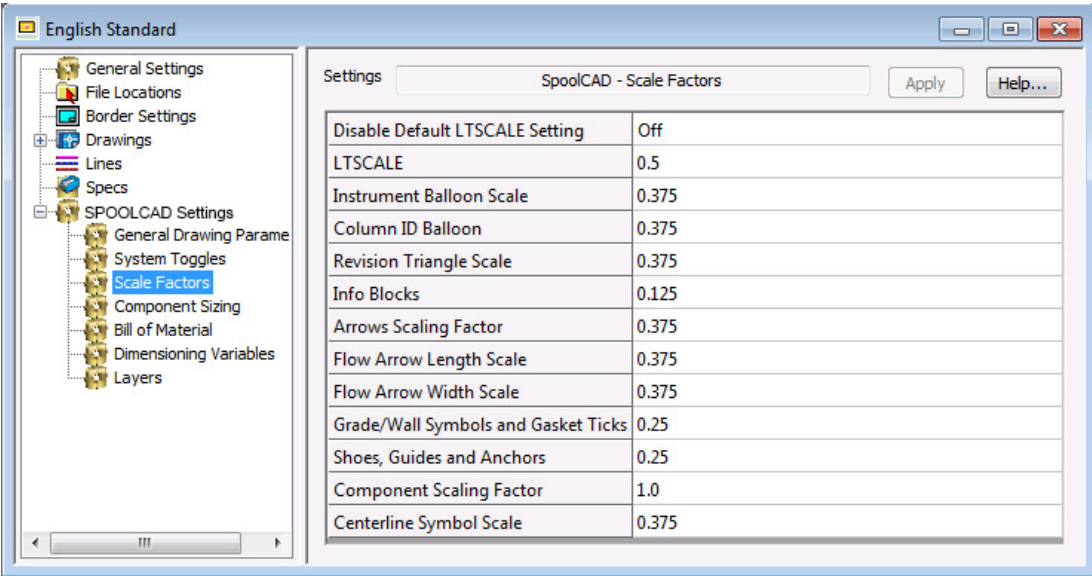
## System Toggles Dialog Box



In this field...	Enter/Do this...
Valve Stem	<p>When the value is set to ON, the user will be prompted to select the orientation of the stem on every valve placed, or to decide that no stem should be placed for individual valve</p> <p>When the value is set to OFF, stems will never be placed when the valves are placed.</p>
Spec Check	<p>When this value is set to ON, the user will be prompted to select When you insert affected components, dialog boxes prompt you to select from a variety of spec options associated with that component.</p> <p>For example, when you insert flanged or welded ball valves, check valves and plug valves, a dialog box prompts you to select a valve type. When you insert screwed/socket weld valves, a dialog box prompts you to select a valve rating.</p>
Branch Check	<p>When the value is set to ON, the branch connection table within the active spec will be checked for each branch component placed, such as a reducing tee or an olet. If the component selected for the active main and branch size does not match the branch connection table, a warning will be issued.</p> <p>When the value is set to OFF, the branch connection table will not be checked.</p>
BW Elbows	When the Square option is selected, welded elbows will appear with

	<p>squared corners, as shown on the left.</p> <p>When the Curved option is selected, welded elbows will appear with rounded corners, as shown on the right.</p> 
Fitting Insert Point	<p>When the Weld option is selected, fittings will be inserted at their weld point, or end point.</p> <p>When the Centre option is selected, fittings will be inserted at their center point.</p>
Flange Insert Point	<p>When the Weld option is selected, the first flange inserted will be inserted at the welded end point.</p> <p>When the Face of Flange option is selected, the first flange inserted will be inserted at the face of flange.</p> <p>This insertion point will toggle within the program as flanges are inserted.</p>
Apply	Click to apply the changes.
Help...	Click to access the <b>Help</b> dialog box.

Scale Factors Dialog Box

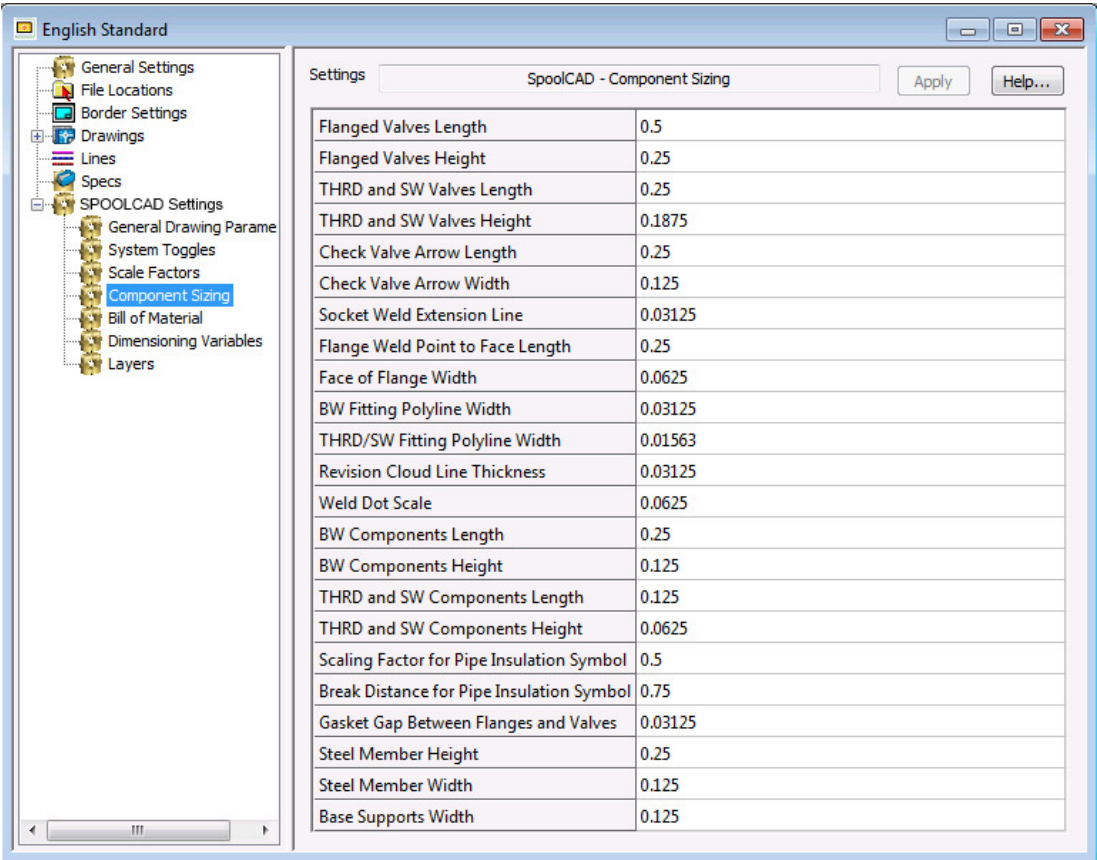


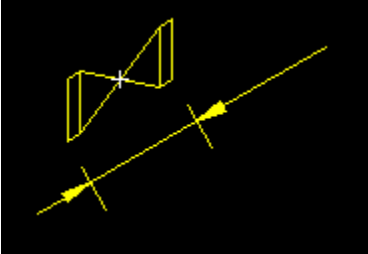
In this field...	Enter/Do this...
Disable Default LTSCALE Setting	When this is OFF, the value in the next line is used as the LTSCALE value. When this is ON, the LTSCALE system variable of the drawing

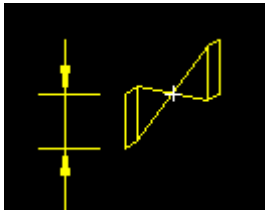
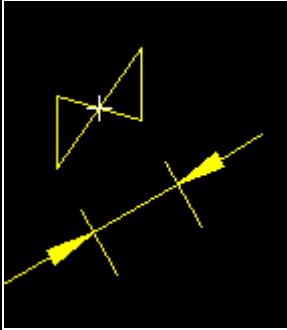
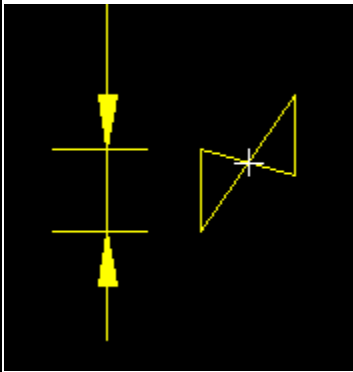
	template will be used.
LTSCALE	This is AutoCAD's LTSCALE system variable that sets the global linetype scale factor. The smaller the scale, the more repetitions of the pattern is generated per drawing unit. For example, with a setting of 0.5, two repetitions of the pattern in the linetype definition are displayed for each drawing unit.
Instrument Balloon Scale	This is the scale of the balloon or hexagon in the drawing when the Balloon with Leader, Hexagon with line or Hexagonal Balloon commands is used. The size in the drawing of the balloon is the size of the original block multiplied by scale factor defined in this parameter. The original size of these blocks is 1".
Column ID Balloon	The size of the balloon in the drawing when the Column ID command on the Pipe Supports tab is used will be the size of the original balloon multiplied by the scale factor defined in this parameter. The original size of the balloon is 1".
Continuation Arrows and Rev Blocks	The continuation arrows and rev blocks will appear in the drawing with dimensions that are the dimensions of the original blocks multiplied by the scale factor defined in this parameter. The original continuation arrow blocks are in the files l-arr*.dwg and r-arr*.dwg
Info Blocks	The size of the info blocks on the components will be the size of the original info blocks multiplied by the scale factor defined in this parameter. The original size of the info blocks is 1/2".
Arrows Scaling Factor	This scaling factor is used for multiple components: North arrow, orifice flanges, pig launcher, slope symbol and the spec break. The size of these components will be the original dimension in the block multiplied by the scale factor defined here.
Flow Arrow Length Scale	The length of the large and small flow arrows in the drawing will be the length of the original flow arrow block multiplied by the scale factor defined in this parameter. These symbols are found in the sm-arrow*.dwg and lg-arrow*.dwg and have lengths of 0.5" and 0.25" respectively.
Flow Arrow Width Scale	The width of the large and small flow arrows in the drawing will be the length of the original flow arrow block multiplied by the scale factor defined in this parameter. These symbols are found in the sm-arrow*.dwg and lg-arrow*.dwg and have lengths of 0.25" and 0.125" respectively.
Grade/Wall Symbols and Gasket Ticks	The dimensions of the grade/wall symbol and the gasket ticks will be the size of the original block multiplied by the scale factor defined in this parameter. These symbols are found in the grde.dwg and gasktick.dwg respectively.
Shoes, Guides and Anchors	The dimensions of the pipe shoes, guides and anchors will be the size of the original block multiplied by the scale factor defined in this parameter. These symbols are found in the show*.dwg, guide*.dwg, and hngr_sym.dwg respectively.
Component Scaling Factor	This scaling factor is used for multiple parameters on the Component Sizing dialog box. Refer to the individual settings for more information.
Scale Factor for Weld Tag	This is the scale of the Weld Tag in the drawing when the weld tag commands is used.
Centerline Symbol Scale	This is the scale of the Centerline symbol in the drawing when the

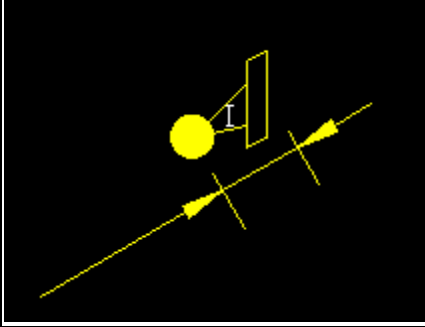
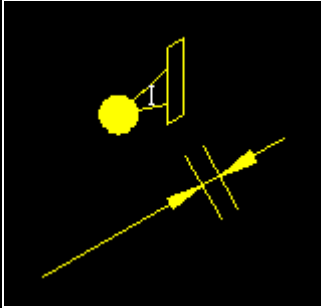
	Centerline symbol command is used.
Apply	Click to apply the changes.
Help...	Click to access the <b>Help</b> dialog box.

Component Sizing Dialog Box

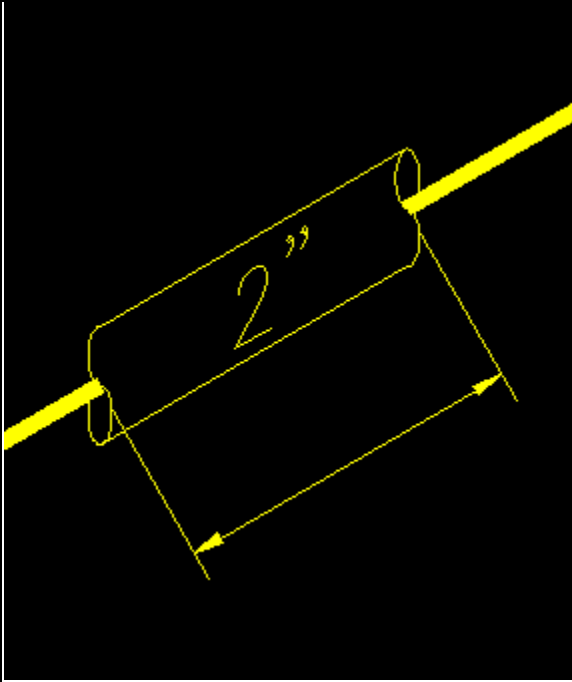
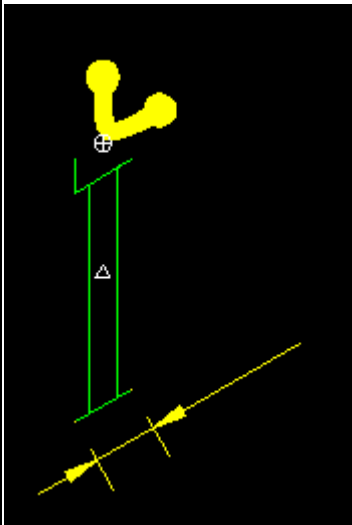


In this field...	Enter/Do this...
Flanged Valves Length	<p>The length of the flanged valves in the drawing will be the size defined in this value multiplied by the Component Scale Factor on the Scale Factors dialog box.</p> 
Flanged Valves Height	<p>The height of the flanged valves in the drawing will be the size defined in this parameter multiplied by the Component Scale Factor on the Scale</p>

	<p>Factors dialog box.</p> 
THRD and SW Valves Length	<p>The length of the small bore valves in the drawing will be the size defined in this parameter multiplied by the Component Scale Factor on the Scale Factors dialog box.</p> 
THRD and SW Valves Height	<p>The height of the small bore valves in the drawing will be the size defined in this parameter multiplied by the Component Scale Factor on the Scale Factors dialog box.</p> 
Check Valves Arrow Length	<p>The length of the check valves in the drawing will be the size of the original check valve multiplied by the scale factor defined in this parameter. The original length of the check valve is 1".</p>
Check Valves Arrow Width	<p>The height of the check valve in the drawing will be the size of the original check valve multiplied by the scale factor defined in this parameter. The original width of the check valve is 0.5".</p>
Socket Weld Extension Line	<p>The distance of the socket weld extension line in the drawing will be the size defined in this parameter multiplied by the Component Scale Factor on the Scale Factors dialog box.</p>
Flange Weld Point to Face Length	<p>The length of the flange symbol from the centre of the weld dot to the centre of the flange face.</p>

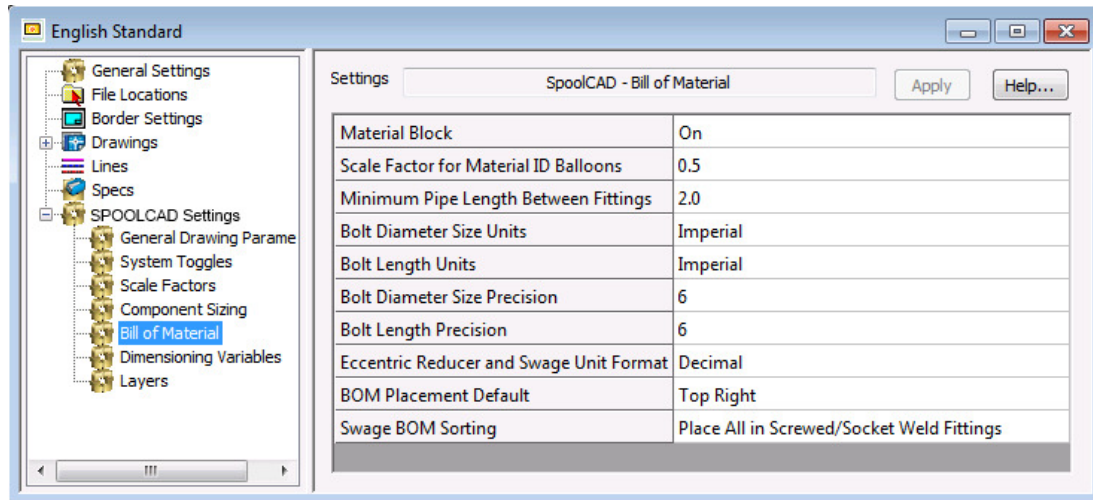
		
Face of Flange Width	The width of the face of flange for the flange symbol. 	
BW Fitting Polyline Width	The thickness of all polylines created when butt weld fittings are placed.	
THRD/SW Fitting Polyline Width	The thickness of all polylines created when threaded or socket weld fittings are placed.	
Revision Cloud Line Thickness	The thickness of the polyline created when the Revision Cloud routine is used.	
Weld Dot Scale	The size of the weld dot will be the size of the original weld dot symbol multiplied by the scale factor defined in this parameter. This symbol is found in weld-dot.dwg and has a diameter of 1”.	
BW Components Length	The larger dimension of butt welded fittings such as the end to end length of the main run of a tee or the end to end dimension of a 180 degree return..	
BW Components Height	The dimension of butt welded fittings such as the centre to end dimension for an elbow, the length of the branch of a tee, or the height of a reducer.	
THRD and SW Components Length	The larger dimension of threaded and socket weld fittings such as the end to end length of the main run of a tee or the length of a pipe nipple.	
THRD and SW Components Height	The smaller dimension of threaded and socket weld fittings such as the centre to end dimension for an elbow or the length of the branch of a tee.	
Scaling Factor for Pipe Insulation Symbol	The height of the pipe insulation symbol and the size of the text will be the dimensions from the original block multiplied by this scale factor. The original pipe insulation block may be found in ins*.dwg.	
Break Distance for Pipe Insulation Symbol	The length of the pipe insulation, from one side of the pipe break to the other side of the pipe break.	



		
Gasket Gap Between Flanges and Valves	The gasket gap in the drawing will be the size defined in this parameter multiplied by the Component Scale Factor on the Scale Factors dialog box.	
Steel Member Height	For wide flanges and channels, the height of the steel member in the drawing will be the height defined in this parameter multiplied by the Component Scale Factor on the Scale Factors dialog box.	
Steel Member Width	For wide flanges and channels, the width of the steel member in the drawing will be the height defined in this parameter multiplied by the Component Scale Factor on the Scale Factors dialog box.	
Base Support Width		The width of the base support in the drawing will be the width defined in this parameter.

Apply	Click to apply the changes.
Help...	Click to access the <b>Help</b> dialog box.

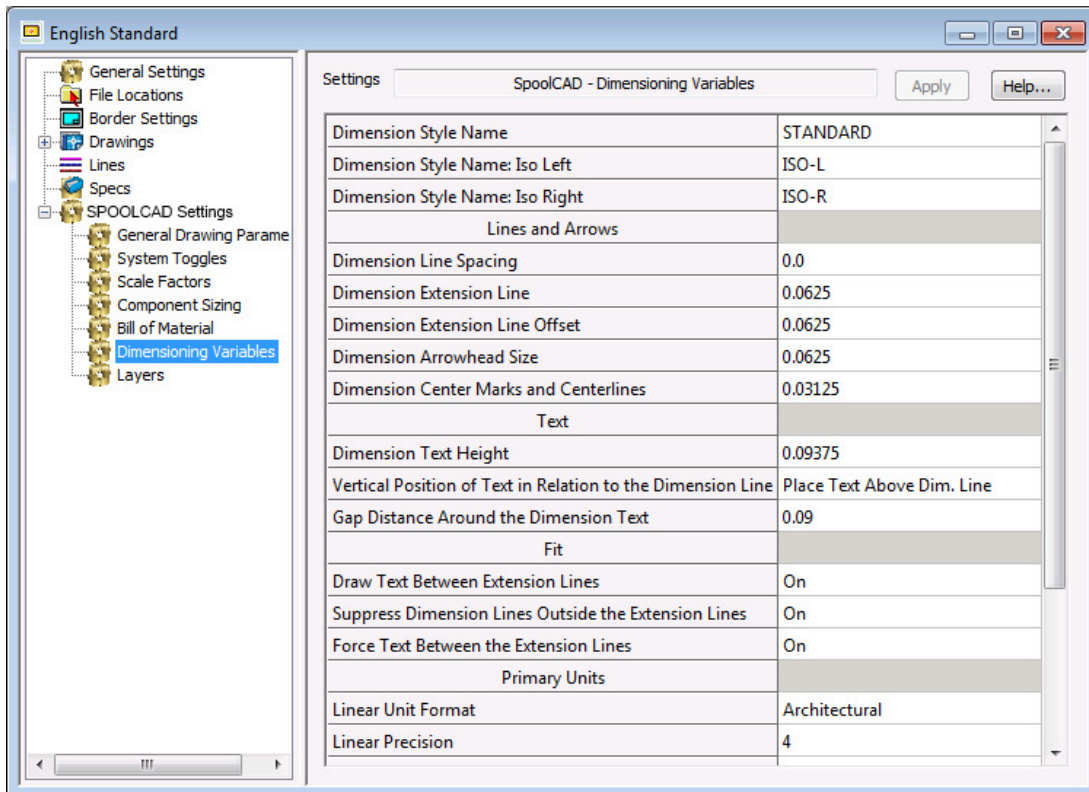
## Bill of Material Dialog Box



In this field...	Enter/Do this...
Material Block	Use this toggle to insert material data when you insert piping components. When you insert components, PROCAD SPOOLCAD writes all available material data into the component's INFO block. <b>Caution:</b> The system uses material data to generate the Bill of Materials. If you deactivate this toggle, you cannot generate a B.O.M. The default setting is 'on'.
Scale Factor for Material ID Balloons	The size of the material ID balloon in the drawing when the Component Numbers or the Single Material Balloon routines on the BOM tab is used will be the size of the original balloon multiplied by the scale factor of the drawing and the scale factor defined in this parameter. The original size of the balloon is 0.5".
Minimum Pipe Length Between Fittings	The minimum pipe allowed between fittings will be this value.
Bolt Diameter Size Units	The units to be used when listing the bolt diameter in the bill of material.
Bolt Length Units	The units to be used when listing the length of the bolts in the bill of material
Bolt Diameter Size Precision	The number of decimal places to use when displaying the diameter of the bolts in the bill of materials.
Bolt Length Precision	The number of decimal places to use when displaying the length of the bolts in the bill of materials.
Eccentric Reducer and Swage Unit Format	Eccentric reducers and swages display the distance between the inlet and outlet centrelines. Select the units that will be used to display this value.
BOM Placement Default	The default insertion point for any lists or bills of material. Default value is Top Right. Other options are top left, bottom right, bottom left.

Swage BOM Sorting	Determines whether swages appear in the Screwed/Socket Weld Fittings or Welded Fittings group in the Bill of Material. Default option is Place All in Screwed/Socket Weld Fittings.
Apply	Click to apply the changes.
Help...	Click to access the <b>Help</b> dialog box.


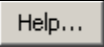
## Dimensioning Variables Dialog Box



All of the fields in the Dimensioning Variables dialog box utilize AutoCAD's dimension style system variables. For more information on these system variables, please refer to AutoCAD's documentation

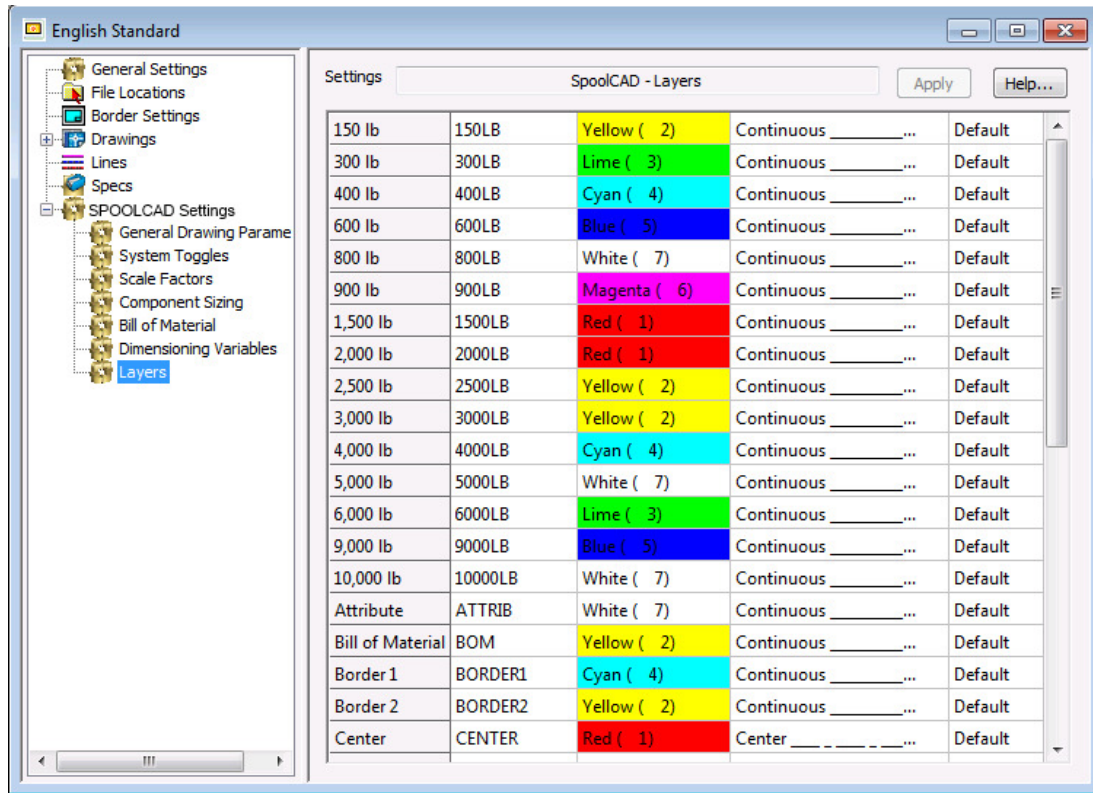
In this field...	Enter/Do this...
Dimension Style Name	The name of the AutoCAD dimension style to be used when using PROCAD SPOOLCAD's Linear Dimension and Continue Dimension routines.
Dimension Style Name: Iso Left	The name of the AutoCAD dimension style to be used when using PROCAD SPOOLCAD's Linear Dimension and Continue Dimension routines.
Dimension Style Name: Iso Right	The name of the AutoCAD dimension style to be used when using PROCAD SPOOLCAD's Linear Dimension and Continue Dimension routines.

Dimension Line Spacing	Sets the spacing between the dimension lines of a baseline dimension. This is AutoCAD's DIMDLI system variable.
Dimension Extension Line	Specifies a distance to extend the extension lines above the dimension line. This is AutoCAD's DIMEXE system variable.
Dimension Extension Line Offset	Sets the distance to offset the extension lines from the points on the drawing that define the dimension. This is AutoCAD's DIMEXO system variable.
Dimension Arrowhead Size	Displays and sets the size of arrowheads. This is AutoCAD's DIMASZ system variable.
Dimension Center Marks and Centerlines	Displays and sets the size of the center mark or centerline. This is AutoCAD's DIMCEN system variable.
Dimension Text Height	Sets the height of the current dimension text style. This is AutoCAD's DIMTXT system variable.
Vertical Position of Text in Relation to the Dimension Line	Controls the vertical placement of dimension text in relation to the dimension line. This is AutoCAD's DIMITAD system variable.
Gap Distance Around the Dimension Text	Sets the current text gap, which is the distance around the dimension text when the dimension line is broken to accommodate the dimension text. Text is positioned inside the extension lines only if the resulting segments are at least as long as the text gap. Text above or below the dimension line is placed inside only if the arrowheads, dimension text, and a margin leave enough room for the text gap. This is AutoCAD's DIMGAP system variable.
Draw Text Between Extension Lines	When this value is ON, text is always placed between extension lines. When this value is OFF, either the text or the arrowheads get moved outside the extension lines based on the best fit. This is AutoCAD's DIMITIX system variable.
Suppress Dimension Lines Outside the Extension Lines	Suppresses arrowheads if not enough space is available inside the extension lines. This is AutoCAD's DIMSOXD system variable.
Force Text Between the Extension Lines	When this value is OFF, dimension lines are not drawn between the measured points when the arrowheads are placed outside the measured points. When this value is ON, dimension lines are drawing between the measured points when the arrowheads are placed outside the measured points. This is AutoCAD's DIMTOFL system variable.
Linear Unit Format	Sets the current units format for all dimension types except Angular. This is AutoCAD's DIMLUNIT system variable.
Linear Precision	Displays and sets the number of decimal places in the dimension text. This is AutoCAD's DIMDEC system variable.
Fraction Format	Sets the format for fractions. This is AutoCAD's DIMFRAC system variable.
Dimension Distance Rounding	Sets rounding rules for dimension measurements for all dimension types except Angular. If you enter a value of 0.25, all distances are rounded to the nearest 0.25 unit. If you enter a value of 1.0, all dimension distances are rounded to the nearest integer. The number of digits displayed after

	the decimal point depends on the Precision setting. This is AutoCAD's DIMRND system variable.
Dimension Linear Scale Factor	Sets a scale factor for linear dimension measurements. It is recommended that you do not change this value from the default value of 1.00. For example, if you enter 2, the dimension for a 1-inch line is displayed as two inches. The value does not apply to angular dimensions and is not applied to rounding values or to plus or minus tolerance values. This is AutoCAD's DIMLFAC system variable.
Dimension Associativity	This is AutoCAD's DIMASSOC system variable. <b>Associative</b> - Automatically adjust their locations, orientations, and measurement values when the geometric objects associated with them are modified. Dimensions in a layout may be associated to objects in model space. <b>Non-Associative</b> - Selected and modified with the geometry they measure. Non-associative dimensions do not change when the geometric objects they measure are modified. <b>Exploded</b> - Contain a collection of separate objects rather than a single dimension object
Show Pipe Cut Length When Dimensioning	This is a toggle which allows you to set the default option to enable or disable pipe cut lengths.
	Click to apply the changes.
	Click to access the <b>Help</b> dialog box.

## Layers Dialog Box

Each designer application includes the **Layers** dialog box. Each layer can be assigned a different name, color, and line type.

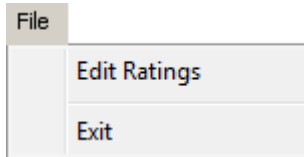


A few things to note about the **Layers** dialog box...

- **First column:** the name of the layer
- **Second column:** the name of the layer as used in the drawing.
- **Third column:** the drop-down list of colors that can be assigned to the layers.
- **Fourth column:** the drop-down list of line types that can be assigned to the layers.
- If you modify the line type and color for a layer, when the user opens the drawing, the properties will automatically change to reflect the modifications.
- If you rename a layer and open an existing drawing, the characteristics will be saved to the new name. Components drawn on the original layer will remain on the original layer and will not move to the new layer.

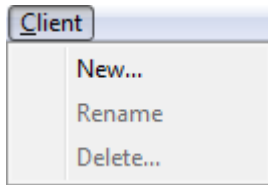
# Menus

## File Menu



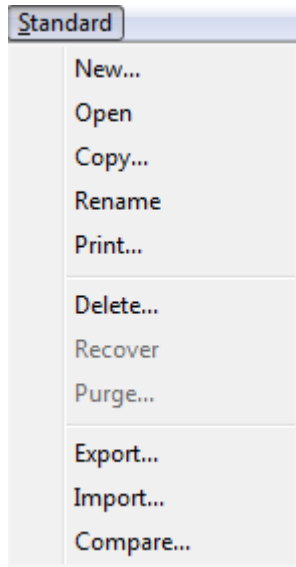
Select this...	To...
Edit Rating	This will allow you to view as well as edit the rating format.
Exit	Click to close the Standard Manager application

## Client



Select this...	To...
New...	This option will allow you to create a new client.
Rename	This option will allow you to rename the selected client.
Delete	This option will allow you to delete the selected client.

## Standard



Click this...	To...
New	Click to create a new standard
Open	Click to open the selected standard
Copy	Click to make a copy of the standard.
Rename	Click to edit the standard name
Print	Click to print a copy of the standard.
Delete	Click to mark a standard as “unavailable” and remove it from the directory tree.
Recover	Click to mark a standard as “available” and make it visible in the directory tree
Purge	Click to remove the standard from the datafile entirely. After you have purged a standard, you are unable to recover it
Export	Click to export the selected PROCAD standard
Import	Click to import a previously exported PROCAD standard
Compare	Click to compare 2 standards side by side.

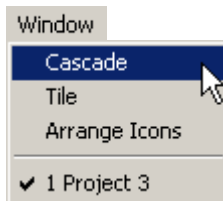
## View Menu





Select this...	To...
Status Bar	Click to show/hide the status bar in the Standard Manager
Navigator Bar	Click to show/hide the navigator bar in the Standard Manager

## Window Menu



Select this...	To...
Cascade	Click to arrange all open data windows so title bars are displayed
Tile	Click to arrange all open data windows so title bars are displayed
Arrange Icons	Click to arrange icons that represent open standard data windows
Open window list	This displays a list of all open data windows. The currently selected window has a check mark.

## Help Menu

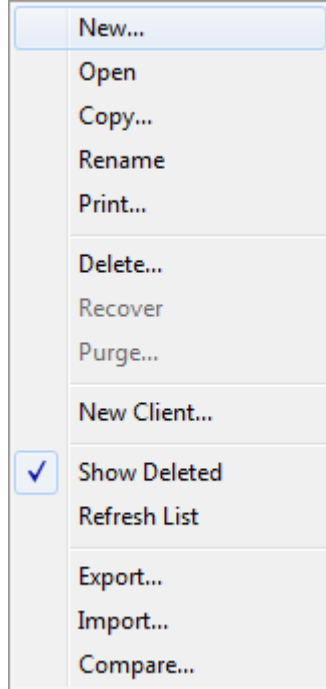


Click this...	To...
About Standard Manager	Click to view information about the program

## Right-click Menus

### Standard Manager Directory Toolbar Right-click Menu

To view this menu, right click on a standard in the Standard Manager directory tree.

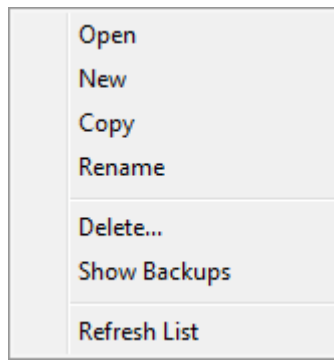


Click this...	To...
New	Click to create a new standard
Open	Click to open the selected standard
Copy	Click to make a copy of the standard.
Rename	Click to edit the standard name
Print	Click to print the standard
Delete	Click to mark a standard as “unavailable” and remove it from the directory tree.
Recover	Click to mark a standard as “available” and make it visible in the directory tree
Purge	Click to remove the standard from the datafile entirely. After you have purged a standard, you are unable to recover it
New Client...	Click to create a new client

Show Deleted	Click to display a list of all deleted standards
Refresh List	Click to update the information on the screen
Export	Click to export the selected PROCAD standard
Import	Click to import a previously exported PROCAD standard
Compare	Click to compare 2 standards side by side.

### Drawings Dialog Box Right-click Menu

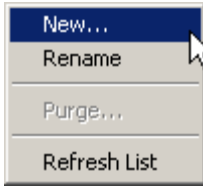
To view this menu, right-click in the Drawings list.



Click this...	To...
Open	Click to open the selected drawing.
New	Click to create a new drawing.
Copy	Click to make a copy of the drawing.
Rename	Click to change the name of the drawing.
Delete	Click to delete the drawing from the database.
Show Backups	Click to show the backup files of the drawings.
Refresh List	Click to update the list of drawings.

### Lines Dialog Box Right-click Menu

To view this menu, right click in the Lines list.



Click this...	To...
New	Click to create a new line
Rename	Click to change the name of the line.
Purge	Click to remove the line from the database.
Refresh List	Click to update the list of lines.

### **Specs Dialog Box Right-click Menu**

To view this menu, right click in the Lines list.



Click this...	To...
Refresh List	Click to update the list of lines.

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