User Guide

CADWorx Plant

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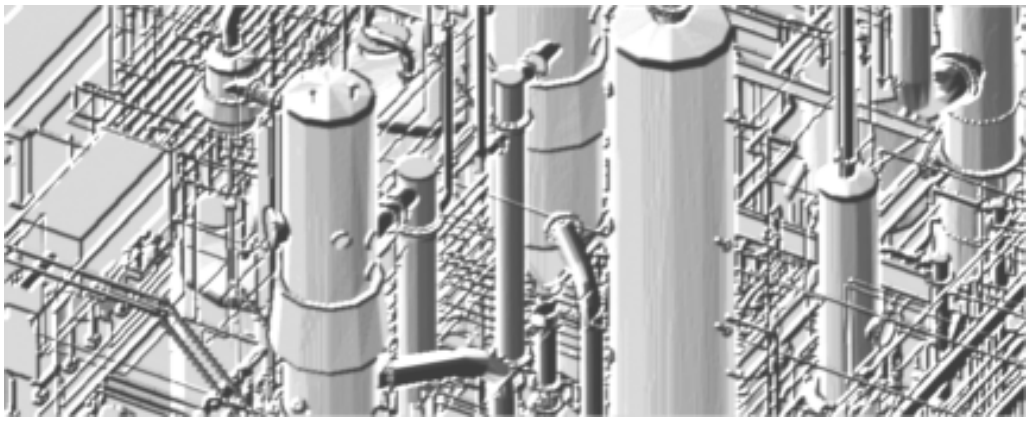
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CHAPTER 1

Setup



In This Chapter

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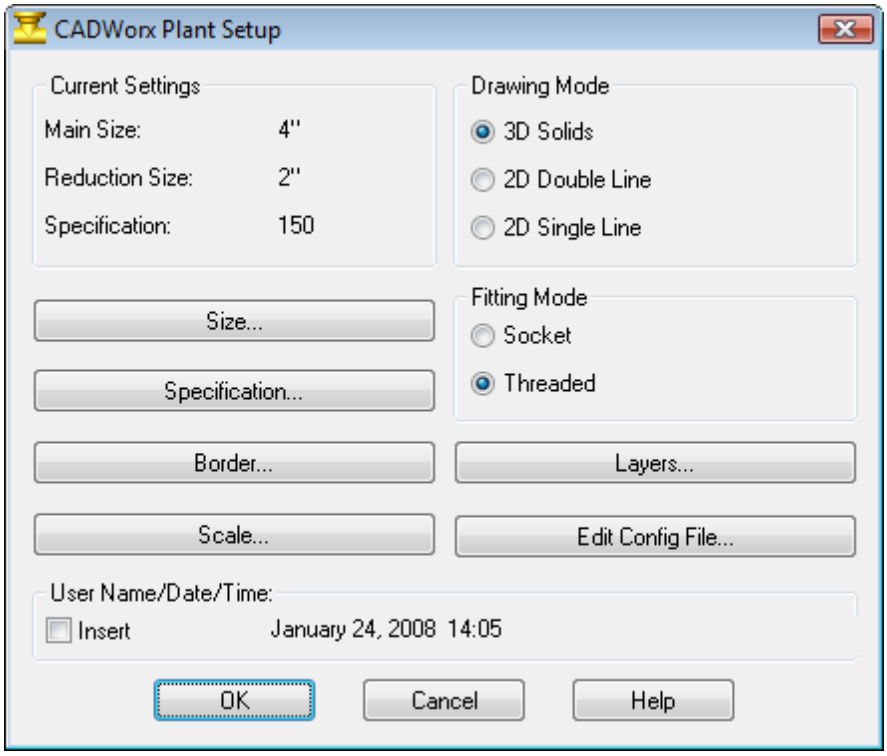
Setup

Plant menu: ▶ **Setup**

Command line: SETUP

Settings toolbar:

By selecting SETUP from the pulldown menu, a dialog providing all available options will be displayed. This dialog also allows various drawing modes and specifications to be set. A complete description of each function or area on this dialog follows below.



Current Settings

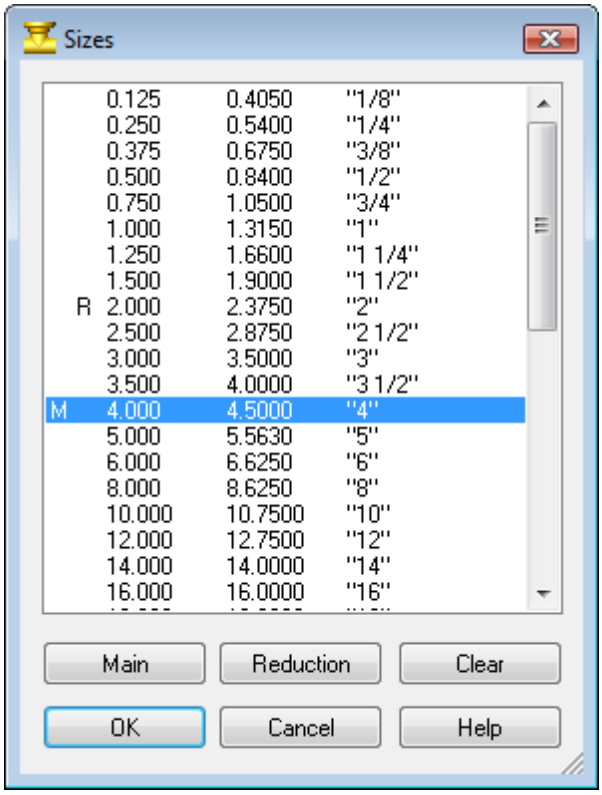
This box displays the current main and reducing sizes. It also displays the currently set specification. If the items are not set it will indicate as such.

Size

Command line: SETSIZE

Settings toolbar:

This button displays the size dialog shown below. The user can double click on the desired size or pick the size and then the desired button at the bottom. One double-click sets the main size and a second double-click sets the reduction size. Cancel will leave the size unchanged.



There are three different commands that can be used to set sizes from the command prompt. The first displays the dialog as shown above. The second allows the main size to be set and the third allows the reduction size to be set at the command line. Using the command alias can greatly speed up the process of setting the main or reduction size.

Command: **MAIN**
Nominal main size <default>: **6** *(or ~ for dialog)*

The dialog shown above can be accessed by typing a tilde (~) at the prompt. At the command line, nominal sizes are entered in various formats, such as; 1.25, 1.25", 1-1/4, 1-1/4", etc. (note that the inch mark is not necessary). Sizes are controlled by the <INSTALL-DIR>\SYSTEM\SETSIZE.TBL (see page 633) file. If a requested size is not in this file, the following prompt will appear:

Size is not available in current libraries...
Edit last library file, C:\CADWorx\SYSTEM\SETSIZE.TBL <Yes>:

By answering yes or enter to this prompt, the user will be able to view or edit the contents of the SETSIZE.TBL file. Modification to this file is allowed for additional sizes.

The reduction size can be set from the command line similar to the main size. It is used for setting reductions for various reducing components and follow the same rules as outlined above.

Command: **RED**
Command: Nominal reduction size <default>: **4** *(or ~ for dialog)*

If a main size is set to 2", a reduction size is set to 4", and a reducing component is requested, the component routine will reverse the order to insert the component in the correct order. Options to reverse the order for certain components (concentric and eccentric reducers) will be provided during the component insertion which will be explained later.

Specification

Command line: SETSPEC

Settings toolbar:

This button provides a standard file selection dialog for choosing specifications to be used in drawings.

CADWorx Plant is supplied with several standard template specifications: 150, 300, 600, 900, 1500, etc. The user can modify, rename, or create any other specifications using the Specification Editor (see page 254) as required. The specifications used with CADWorx Plant are the same as the ones used with CADWorx/P&ID. It is advised to locate the specifications in a central directory if both programs are used together. Please refer to the following Edit Configuration (see page 48) section for more information on locating these directories other than their default <INSTALL-DIR>\SPEC directory.

As with the size dialog, there are two different versions of this command. One will provide a dialog and the other, SPEC will provide a command prompt. Typing in the command generates the following prompt. By typing a tilde(~) in response to the prompts, a standard selection file dialog will be shown as discussed above.

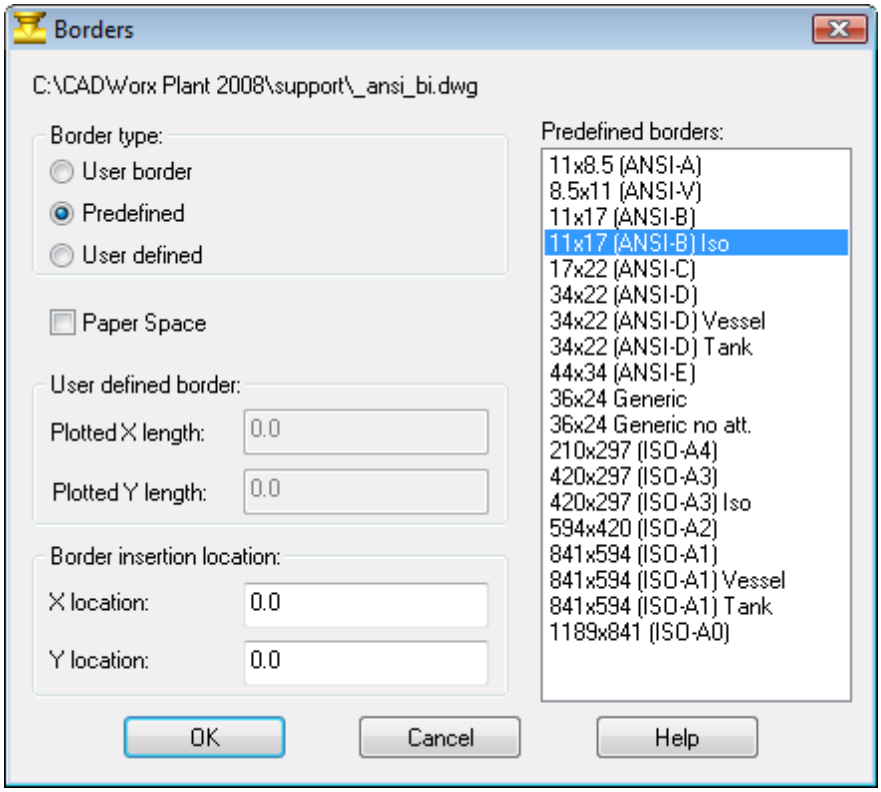
```
Command: SPEC
Specification <default>: 150  (or ~ for dialog)
Prepared E:\CADWORX PIPE\SPEC\150.spc specification...
```

This sets the specification in memory and it also creates a layer named "_150". It will remain set until the user sets another specification or ends the drawing. The "_" (underscore character) in front of the specification layer name allows various CADWorx Plant routines to recognize any user-supplied name. If the specification was not found the following prompt will appear.

```
150 specification not found...
```

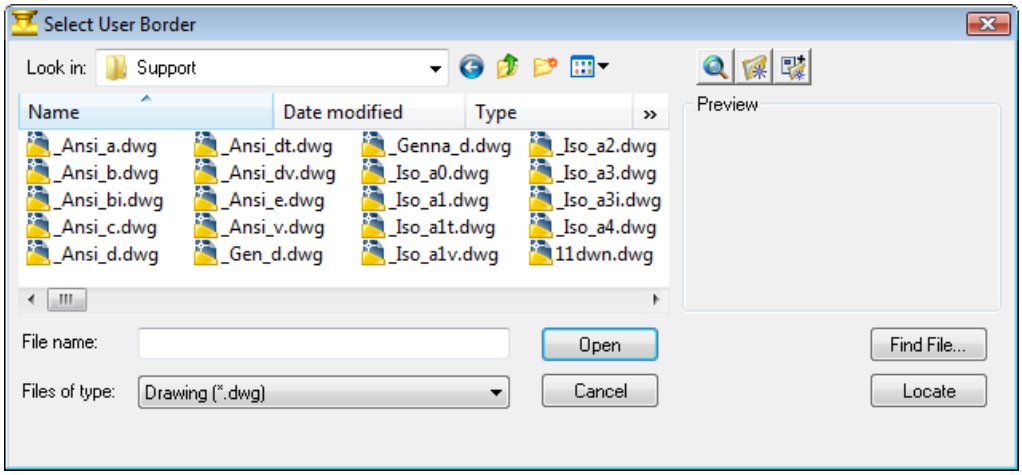
Borders

The following dialog is used for setting up borders within CADWorx Plant.



User Border

This button provides a file selection dialog for choosing the required border. The user can navigate through this dialog to locate any user border or to choose one of the default CADWorx Plant borders.



Predefined

This button supplies the list of drawings in the *Predefined borders* list. The drawings list is retrieved from the MVSETUP.DFS file located in the <INSTALL-DIR>\SUPPORT directory. The MVSETUP.DFS must be located in the first search path of the ACAD support directories.

User-Defined

This button provides the *User defined border* box and allows the user to define a simple rectangle that will be drawn upon exiting these dialogs.

Paper Space

This toggle causes the border to be inserted in the paper space.

User defined border

These edit boxes define the size of the user border. The border will be a simple rectangle. The *Plotted X length* and the *Plotted Y length* will dictate the overall height and width of the border. The rectangle will be drawn with the PLINE command. This edit box will not be available if the *User defined* button above is not selected.

Border insertion location

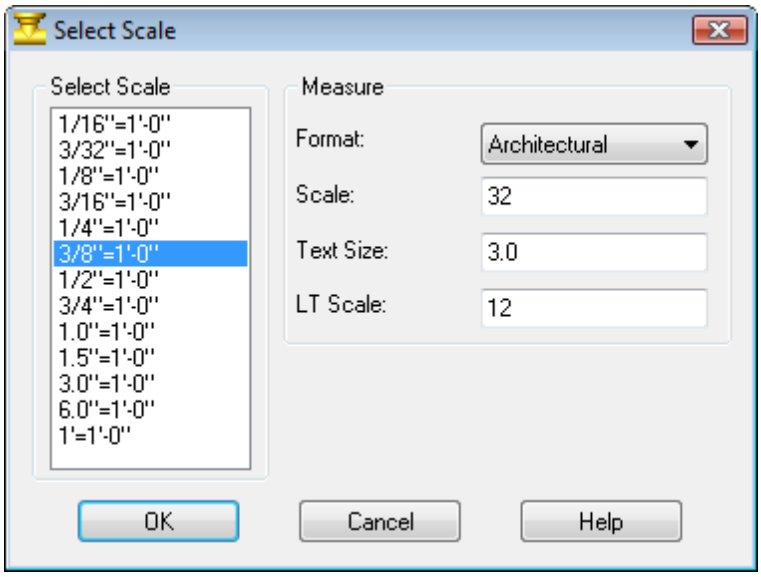
These edit boxes supply the X and Y coordinates for the insertion point of the border drawing. These boxes will be grayed if the *No border* button is selected.

Predefined borders

This list is provided when the user selects the *Predefined* button. As mentioned above, the list is created from the MVSETUP.DFS file. For more details, see the description above.

Scale

This list box displays all the available scales listed in the *Format* list box (description follows). By selecting the desired scale, the *Scale, Textsize* and *LT Scale* edit boxes will be filled in with the appropriate sizes. The scale (DIMSCALE), text size (TEXTSIZE), and line type scale (LTSCALE) are based on the values in the SCALE.DAT file located in the <INSTALL-DIR>\SYSTEM sub-directory. The scales dialog is shown below.



The *Format* list box supplies four different formats to choose from, Architectural, Engineering, Decimal and Metric. Each format updates the Scales list box for the user's selection.

The *Scale* edit box provides a means of automatically or manually setting the scale (DIMSCALE setvar) of the drawing. By selecting an item in the scales list box, each edit box value will be updated. The user may also type a value in these boxes.

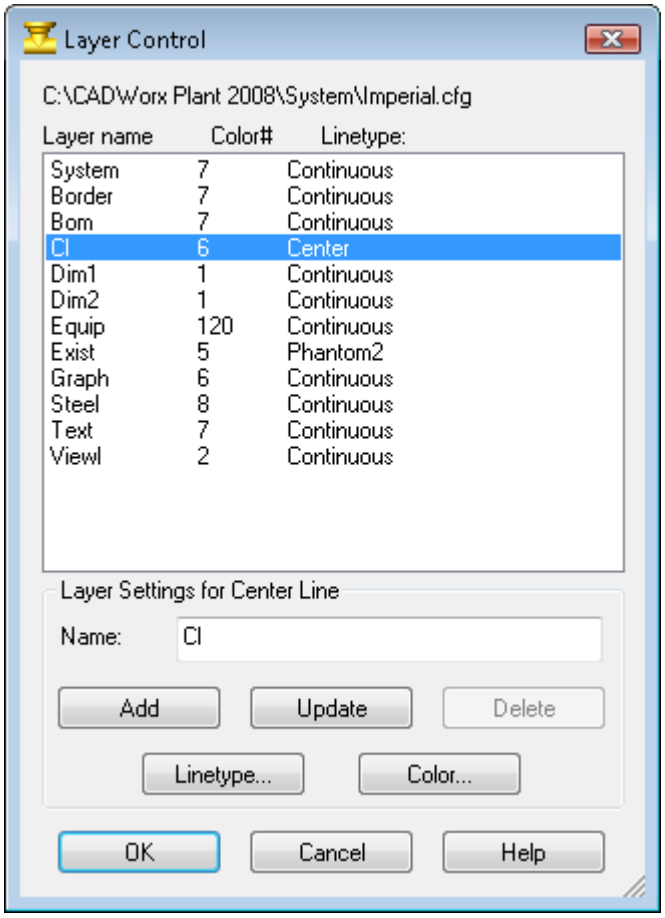
The *Textsize* edit box acts the same as the *Scale* edit box.

The *LT Scale* edit box provides a means of applying a line type scale as above. When a scale is selected from the list box, this edit box automatically updates.

Layers

The following dialog is used for changing the default names, line types, and colors of the startup layers. The user can also add and delete layers here. Some layers are mandatory for use with CADWorx Plant and cannot be deleted.

Changes made here will affect the current drawing session similar to the AutoCAD LAYER command.



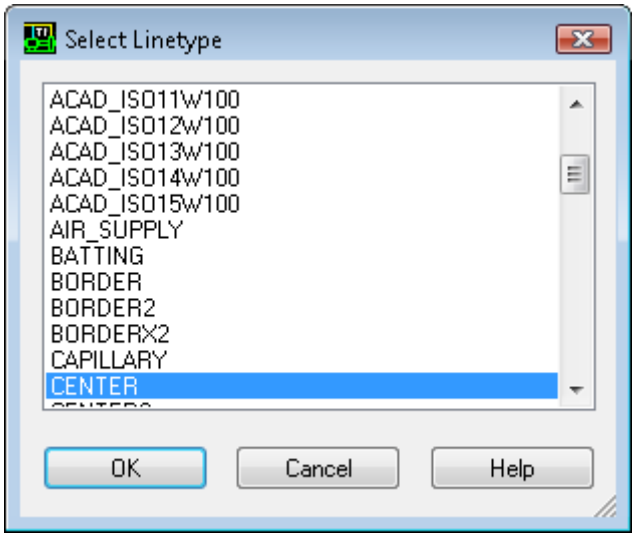
Note: If the configuration file's permission is set to read-only, only the cancel button will be enabled.

Name

After picking a layer name in the list box above, this edit box allows the layer to be renamed. The layer name may be anything that is allowed by AutoCAD. Wild card characters are not allowed.

Linetype

This button searches through the CFM.LIN and the ACAD.LIN (only if found in AutoCAD search path) and presents all the line types in these files for use in the startup procedure as shown below.



Color

This button provides the standard color dialog for selecting the color of the layer.

Add

This button allows the user to add any layers required. These layers will be created when the drawing is initialized with CADWorx Plant.

Update

This button updates the list box after the name of the layer has been changed. Changing focus to another location will accomplish the same effect.

Delete

This button simply deletes any layer that is not needed. Some layers cannot be deleted which will cause this button to be grayed out.

Note: If "CI" and "CI_Steel" layers are frozen or off, some CADWorx commands many not function since the CADWorx data is stored on the center line that is on these layers.

CADWorx Plant, Steel, Equipment, PV Fabricator Layers

Layer#	Default Name	Layer Used For
Layer1	System	Non-graphical components (Gaskets, Bolts, TAP, OTAP)
Layer2	Border	Border drawing
Layer3	BOM	BOM graphics
Layer4	CI	Center line of components - piping
Layer5	Dim1	Dimensions
Layer6	Dim2	Dimensions
Layer7	Equip	Equipment components
Layer8	Exist	Existing Components
Layer9	Graph	Graphics annotations (Arrows, Section, Plan, Detail Labels)
Layer10	Steel	Steel Components
Layer11	Text	Text annotations
Layey12	Viewl	Viewports
Layey13	CI_Steel	Center line of components - steel
Layey14	HVAC	HVAC components
Layey15	CI_HVAC	Center line of components - HVAC

Layey16	TopWorks	Valve TopWorks solid
Layey17	Clash	Clash entity
Layey18	CADWorx1	Future use
Layer19	CADWorx2	Future use
Layer20	CADWorx3	Future use
Layer21 to Layer100	(User Defined)	(User Defined)

CADWorx P&ID Layers

Layer#	Default Name	Layer Used For
Layer1	System	Non-graphical components
Layer2	Border	Border drawing
Layer3	Major-1	Major Process Lines 1
Layer4	Major-2	Major Process Lines 2
Layer5	Minor-1	Minor Process Lines 1
Layer6	Minor-2	Minor Process Lines 2
Layer7	Inst-1	Instrument Lines 1
Layer8	Inst-2	Instrument Lines 2
Layer9	Elec1	Electrical Lines 1
Layer10	Elec-2	Electrical Lines 2
Layer11	Equip	Equipment
Layer12	Text	Text
Layer13	Graph	Graphics
Layer14	Exist	Existing
Layer15 to Layer100	(User Defined)	(User Defined)

Edit Config File

This button brings up the dialog box discussed in the Configuration (see page 48) section. The Edit Configuration File button is provided in the Setup dialog box so the user can modify the configuration file <INSTALL-DIR>\SYSTEM\IMPERIAL.CFG (metric METRIC.CFG).

Note: If the configuration file's permission is set to read-only, only the cancel button will be enabled.

Drawing Modes

These toggles allow setting the default drawing mode. Available modes include 3D Solids, 2D Double Line and 2D Single Line. See Mode Convert (see page 295) for more information on drawing modes.

Fitting Mode

Select either socket weld or threaded for hub based components.

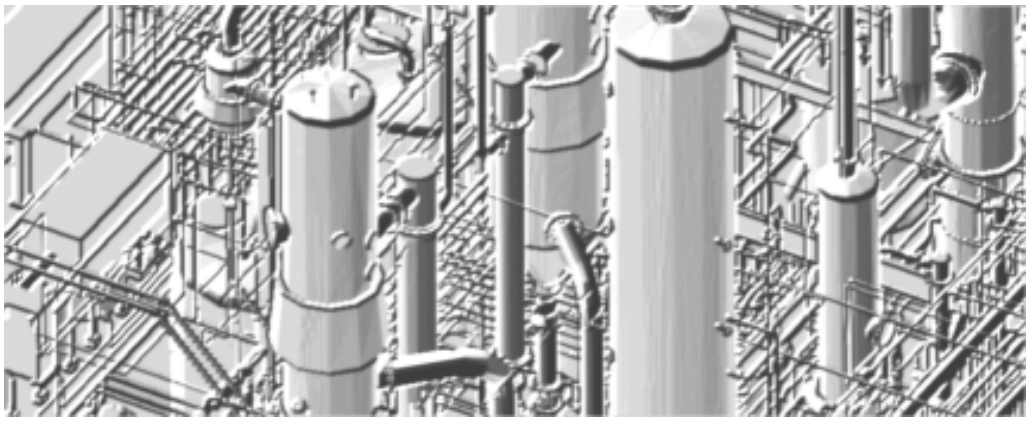
If threaded mode is set, all components that are of the hub-type category will have properties based on the threaded section of the specification. If socket weld mode is set, all components that are of the hub-type category will have properties based on the socket weld section of the specification. Both socket weld and threaded modes will reference the file location described in the specifications for exact parametric dimensional properties and engagements.

User Insert

The user name is the current user in the Windows environment working in AutoCAD. The insert toggle places the name and date in the lower left corner of the drawing for future reference.

CHAPTER 2

Installation



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Conventions

Before reviewing this guide, it is important to understand the terms and conventions used in the documentation.

Use the information below to identify the convention and the type of information.

Convention	Type of Information
<INSTALL-DIR>	This refers to the CADWorx installation directory on your computer. For example, C:\CADWorx Plant. Substitute your installation directory anywhere this notation is used.
<ACAD-DIR>	This refers to your AutoCAD installation directory. For example, C:\AutoCAD 2000i. Substitute your installation directory anywhere this notation is used.

<CD-DRIVE>	This refers to your CD-ROM drive. Substitute your CD-ROM drive letter anywhere this notation is used.
Command Line	Items that need to be typed in or appear at the AutoCAD command line.
User Responses	This refers to user responses to command prompts.
Commands	CADWorx and AutoCAD commands.
Options	Command line options.
CAPITALS	AutoCAD system variables or keywords.
KEY+KEY	Key combinations for which the user must press and hold down one key and then press another, for example, CTRL+P, or ALT+F4.

Program Support / User Assistance

CADWorx Plant has been developed by designers and engineers, for designers and engineers. The design of CADWorx Plant targeted ease of use, wide applicability, transfer links to pipe stress, and overcoming the limitations of other piping CAD packages. CADWorx Plant can provide orthographic, isometric, and 3D models.

In many cases, the difficulties and incompleteness of current software products makes drawing in 3D a haphazard affair. CADWorx Plant addresses these concerns - models can be generated in single line and turned into orthographic plans and elevations, or converted into 3D. Alternatively, the model can be constructed initially using 3D objects. The AutoISO option makes creating isometric drawings an automatic, hands-off operation.

A major capability of CADWorx Plant is its bi-directional data link (see page 193) with the pipe stress program CAESAR II. Using this link, a model can be generated on the CAD side, transferred to CAESAR II for analysis and modification, and then the altered model transferred back for drawing revision. Alternatively, a model initially created in CAESAR II can be sent to CADWorx Plant for drawing generation. In addition, the results of the stress analysis (stresses, restraint loads, hanger design) can be accessed for the development of stress-isometrics.

During the development of CADWorx Plant, the comments and suggestions of many senior pipe designers were incorporated to provide a wide range of capabilities. CADWorx Plant provides the most efficient tool available today for generating piping drawings. This user manual provides an explanation of all of the functions and routines of CADWorx Plant.

Comments and suggestions from the user community are always welcome. If the user has suggestions about CADWorx Plant, or has a “better idea”, we welcome them to contact COADE technical support.

COADE understands the designers need to produce efficient, economical and expeditious designs and drawings. To that end, COADE has a staff of helpful professionals ready to address any CADWorx Plant issues raised by the users.

Installation Overview

CADWorx Plant is installed on the system hard drive using the installation program called SETUP.EXE located on the CD ROM. This installation program has been designed to allow full installation and ease of updating. This section will explain the operation of the installation program, the format of CADWorx Plant program on the CD ROM, and the options available to the user.

The installation process is relatively simple to accomplish. After starting the installation program the user will be prompted for the required information. If the computer that the installation is being performed has auto run enabled, the setup procedure will automatically start.

The CD ROM contains various files, all of which are compressed into files that the installation can read. The only way to extract these files out is to run the installation program (SETUP.EXE).

Installation Start

Quick Start

Run the *SETUP.EXE*.

To start CADWorx Plant, pick the icon created during installation.

Dialogs will carry the user through the rest of the installation.

First time execution of the program will require certain startup parameters to be set. Most of the time, the defaults will be sufficient.

Procedure

The installation is similar to most other Windows installations. The following information will be required:

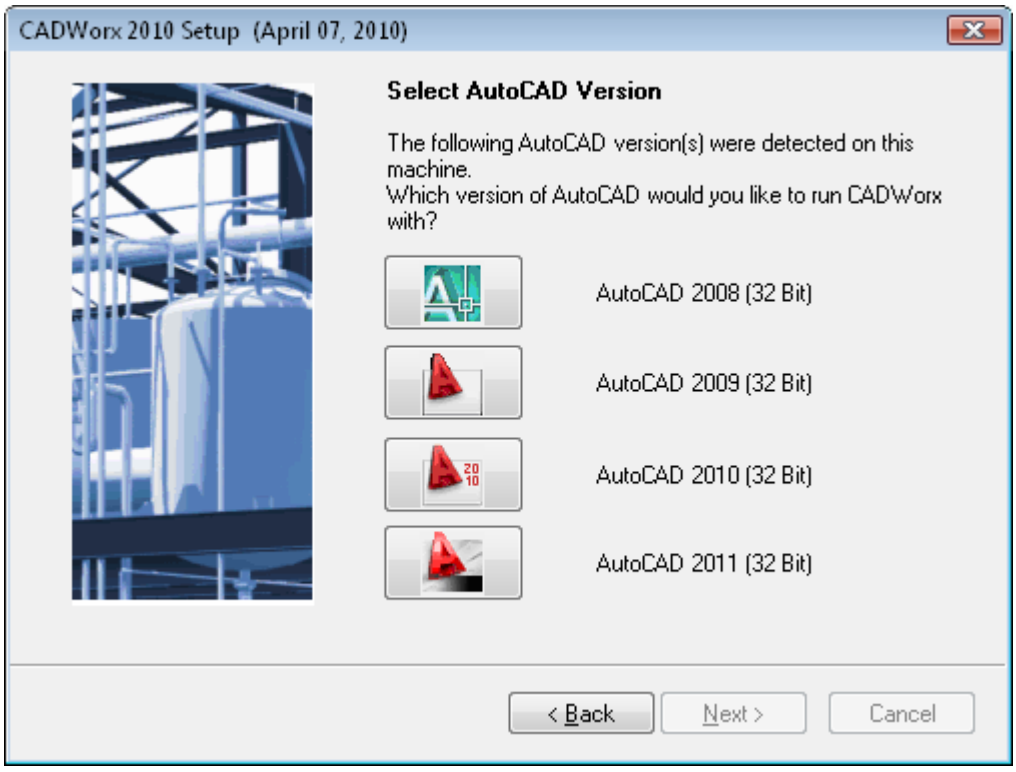
- What drive will be used?
- Which version of AutoCAD will CADWorx Plant use?
- What directory will CADWorx Plant be installed in?
- The name of the Windows program group.
- The type of the hardware lock

Once these items are known, the installation process can be started. The installation process is detailed in the steps below:

1. Insert the CD ROM in the drive.

2. Select the Start button.
3. Select Run.
4. In the command line type in <CD-DRIVE>:\SETUP.EXE, (or use the Browse... button to locate SETUP.EXE on the CD ROM drive)
5. Select the OK button and this will start the installation procedure.

The next two questions pertain to AutoCAD version and install directory. The installation program will automatically try to locate a compatible AutoCAD install directory.



CADWorx 2010 is compatible with the following AutoCAD versions:

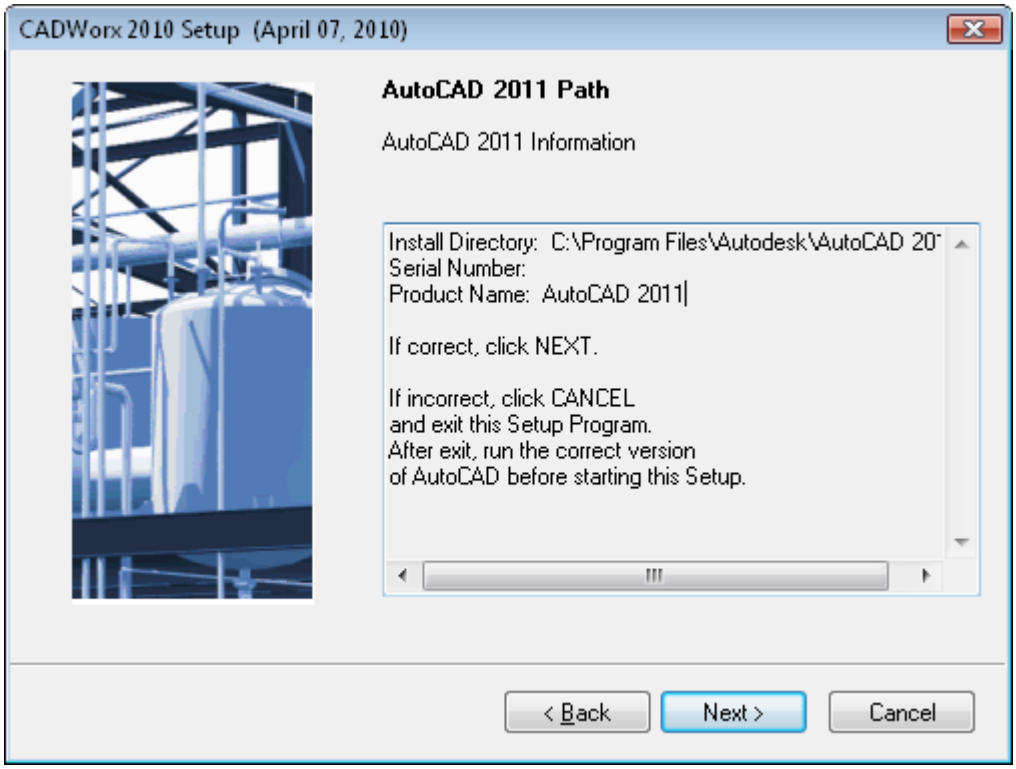
- AutoCAD 2011 (32 bit) / AutoCAD 2011 (64 bit)
- AutoCAD 2010 (32 bit) / AutoCAD 2010 (64 bit)
- AutoCAD 2009 (32 bit) / AutoCAD 2009 (64 bit)
- AutoCAD 2008 (32 bit) / AutoCAD 2008 (64 bit)
- AutoCAD 2007

CADWorx 2010 is compatible with the following Autodesk vertical products:

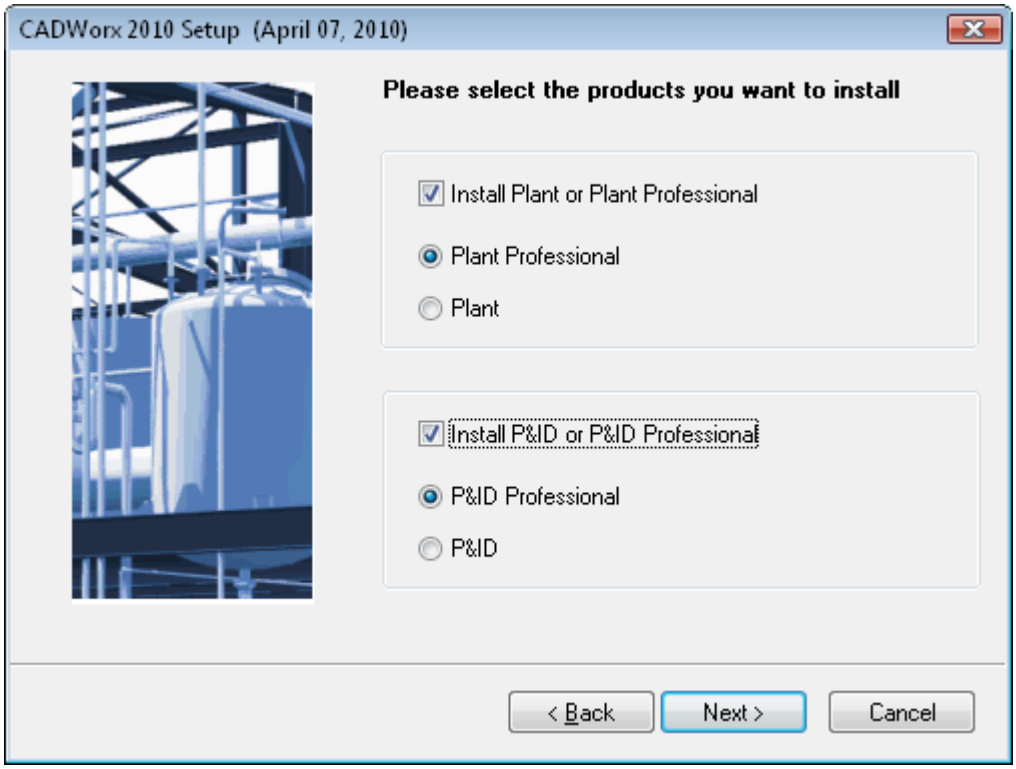
- AutoCAD Mechanical 2011/2010/2009/2008/2007
- AutoCAD Electrical 2011/2010/2009/2008/2007
- Autodesk Mechanical Desktop 2009/2008/2007
- AutoCAD Architecture 2011/2010/2009/2008, Autodesk Architectural Desktop 2007
- AutoCAD MEP 2011/2010/2009/2008, Autodesk Building Systems 2007.1/2007
- AutoCAD Land Desktop 2011/2010/2009/2008, Autodesk Land Desktop 2007
- AutoCAD Civil 3D 2011/2010/2009/2008, Autodesk Civil 3D 2007
- AutoCAD Map 3D 2011/2010/2009/2008, Autodesk Map 3D 2007

CADWorx 2010 is **NOT** compatible:

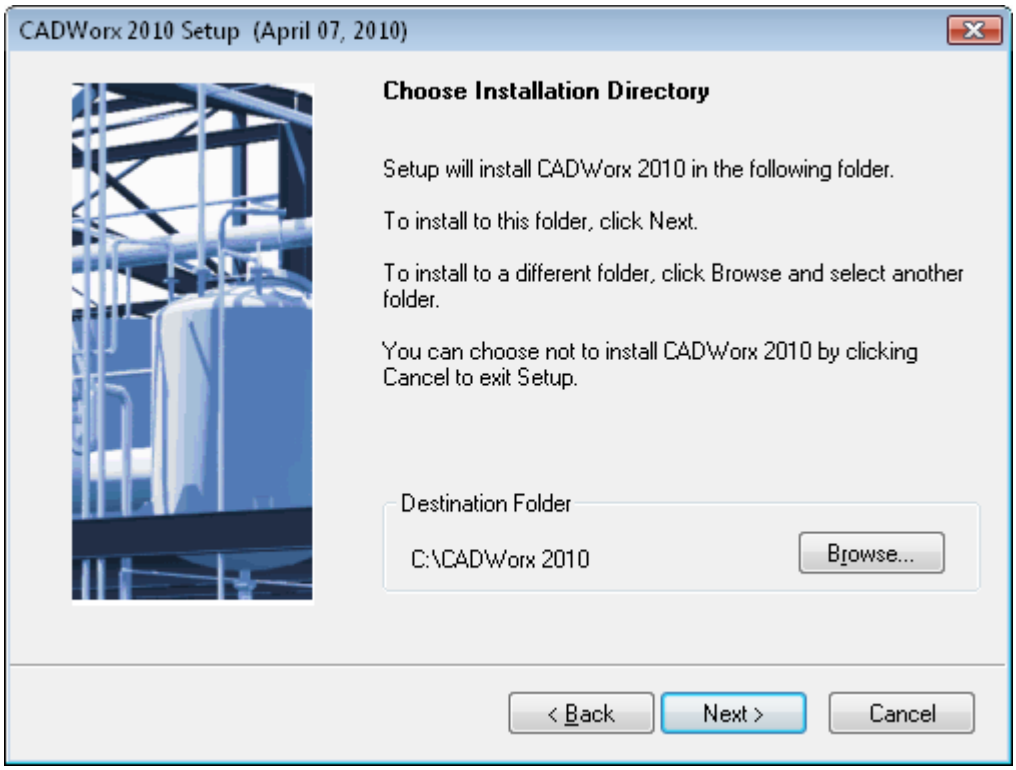
- AutoCAD LT
- Autodesk Inventor
- Autodesk Revit



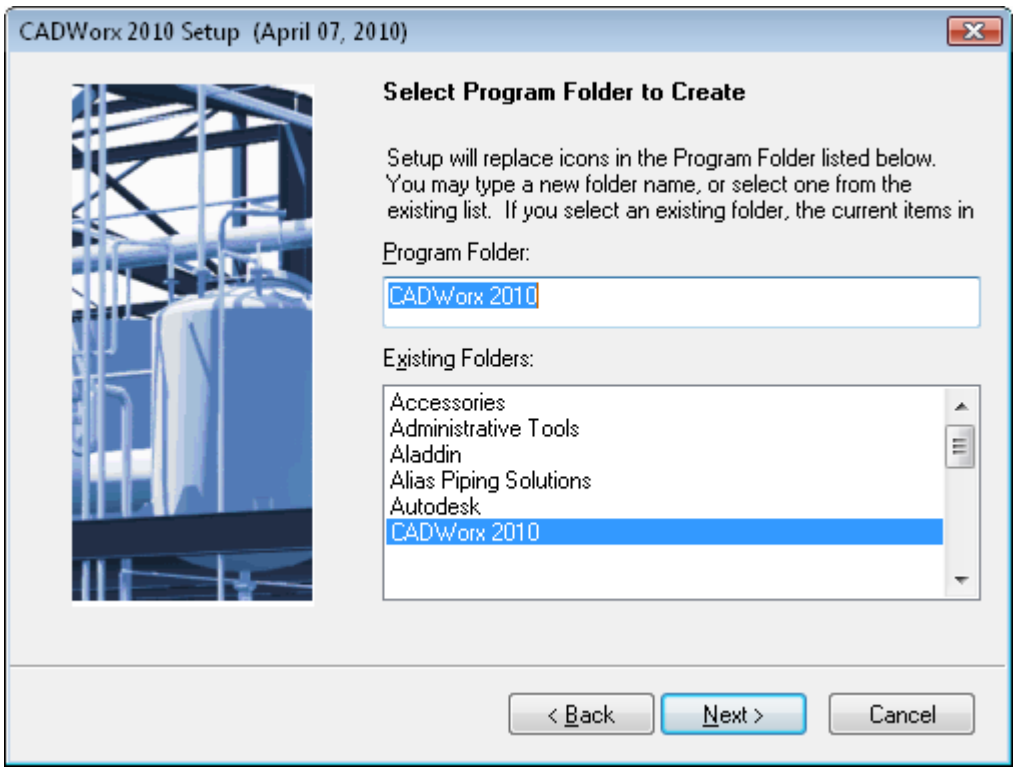
For CADWorx Plant or CADWorx P&ID, select the appropriate application to install.



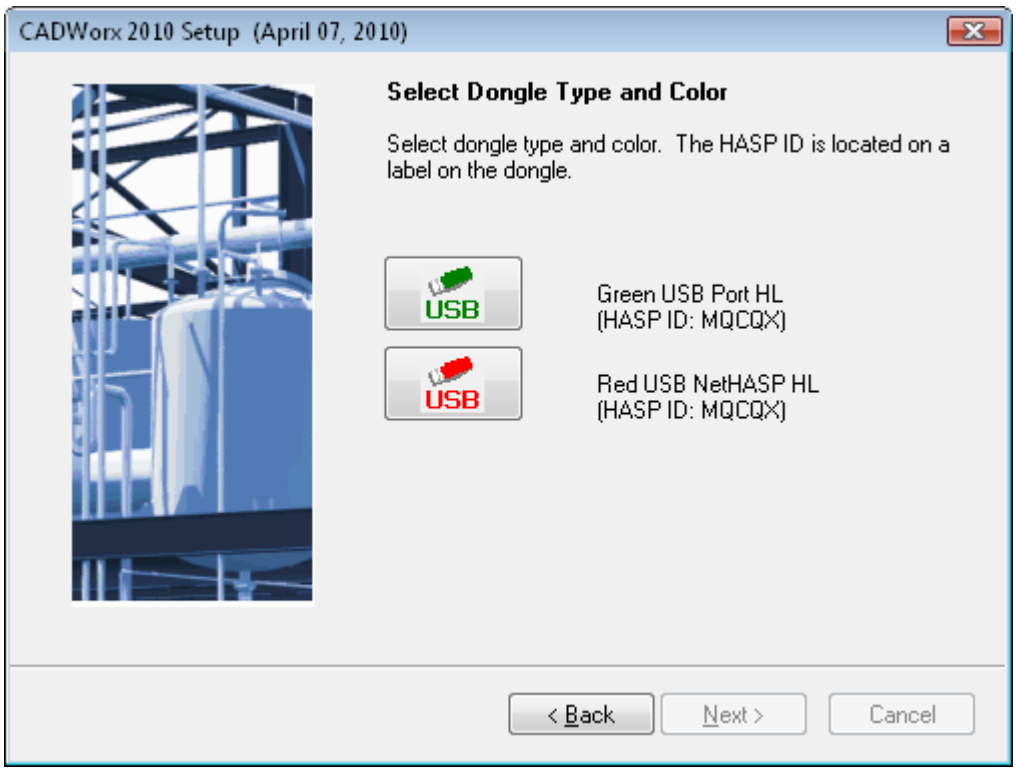
The next step questions where to install CADWorx Plant. The drive and directory location will be required. Once the target disk drive and directory are known, the installation program checks their existence and available space. If the target directory does not exist, it is created.



The next step asks for the Windows program group. The dialog will show all current program groups on the machine. The user can either choose a new one, or overwrite the entries in an existing group.



The next step asks for the hardware lock type and color. The installation will install the necessary files and drivers for the option chosen.



During the installation process, a progress bar will be displayed along with the status of the installation process.

Icon Properties

CADWorx Plant can be run from any location on any directory (network or local). The user has to add all support search paths that might be required to find certain project or job folders. These search paths can be added or modified from the CADWorx Plant profile. CADWorx Plant uses the AutoCAD command line switch /P (User Profile) for starting and running the within AutoCAD.

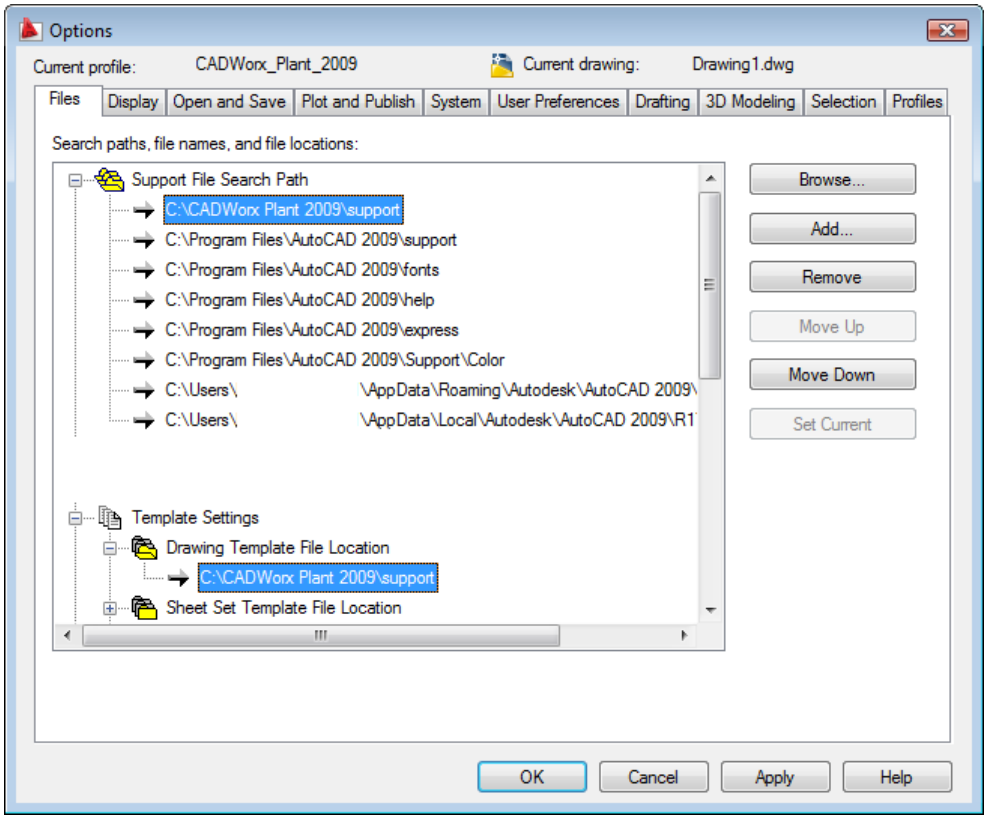
If AutoCAD 2011 is installed in C:\Program Files\Autodesk\AutoCAD 2011, the properties of the shortcut created will resemble the following:

```
"C:\Program Files\Autodesk\AutoCAD 2011\Acad.exe"  /P CADWorx_Plant
```

The installation program will automatically create a CADWorx_Plant user profile. The profile can be created manually by adding the support file search paths (shown below) and the template file location.

The CADWorx Plant support directory has to be first in the search path because the ACAD.RX file used to load CADWorx Plant is found in the <INSTALL-DIR>\SUPPORT directory. The ACAD.RX file is a text file that AutoCAD uses to load ARX routines. If the user wants another support directory to be first, copy the ACAD.RX file into that directory. The CADWorx Plant menu is loaded by the ARX file.

Note: To setup the profile for another user login on the same machine, run the utility SETUP_PROFILE_Plant.EXE. This utility is located in the root install directory of CADWorx Plant


Options
Current profile: CADWorx_Plant_2009
Current drawing: Drawing1.dwg
Files
Display
Open and Save
Plot and Publish
System
User Preferences
Drafting
3D Modeling
Selection
Profiles
Search paths, file names, and file locations:
Support File Search Path
C:\CADWorx Plant 2009\support
C:\Program Files\AutoCAD 2009\support
C:\Program Files\AutoCAD 2009\fonts
C:\Program Files\AutoCAD 2009\help
C:\Program Files\AutoCAD 2009\express
C:\Program Files\AutoCAD 2009\Support\Color
C:\Users\ \AppData\Roaming\Autodesk\AutoCAD 2009\
C:\Users\ \AppData\Local\Autodesk\AutoCAD 2009\R1
Template Settings
Drawing Template File Location
C:\CADWorx Plant 2009\support
Sheet Set Template File Location
Browse...
Add...
Remove
Move Up
Move Down
Set Current
OK
Cancel
Apply
Help

Software Installation on Network Drive

The installation program treats a network drive no differently than a local hard drive. Simply specify the target installation drive and directory and the software will be installed accordingly. Some networks protect installation directories from subsequent modification by users. CADWorx Plant requires setting the access rights to the installation directory to write, read and modify in specific directories.

Since CADWorx Plant utilizes data files specific to the installation that a user may need to modify, these files cannot be located in the protected installation directory. These data files are located in a sub-directory named <INSTALL-DIR>\SYSTEM and <INSTALL-DIR>\SUPPORT. Users should be given all access rights to this <INSTALL-DIR>\SYSTEM and <INSTALL-DIR>\SUPPORT directory (while the actual name of the program's installation directory can be specified by the person installing the software, the \SYSTEM and \SUPPORT sub-directory name is fixed, and is automatically created). Renaming the sub-directories will cause the software to fail and generate errors.

If the users are allowed to modify the specifications and/or component data files, these default directories also need to allow full access. The specification files are found in the <INSTALL-DIR>\SPEC directory. The data files are found in the <INSTALL-DIR>\LIB_I, \LIB_M, \LIB_MM, \STEEL_I, and \STEEL_M directories. These directories can be moved to a user specified location after the program is installed by using the configuration facilities within the Setup (see page 18) function within CADWorx Plant

When the software is installed on a network drive, the individual user workstation must contain the command line directives in the icon to properly locate and run the software. The easiest way to create the icon on the user workstation is to run the SETUP_PROFILE_Plant.EXE routine on the workstation.

Note: Administrator rights will be required on the machine to install the program. After the installation has been completed, administrator rights can be revoked. Administrator rights are not required to run CADWorx Plant.

CADWorx Plant can be run on a network or stand alone workstations. There are three different network installation configurations possible.

1. The first configuration occurs when the software is installed on the network drive, and the users all have local hardware lock (a white hardware lock attached on a user workstation).

2. The second configuration occurs when a single network (a red lock attached on a network system) lock is to be used, and subsequently accessed by multiple users. The software itself is installed on a network drive or on local workstations.

3. The third configuration occurs when the software is installed on the network drive, and both local hardware locks and a network hardware locks are used.

All three configurations will work with CADWorx Plant.

External Software Lock (ESL)

The installation program installs the drivers for the External Software Lock (ESL). ESLs are red, white, green, or purple in color and attach to parallel or USB ports.

All ESL related files and drivers can be found in the <INSTALL-DIR>\ASSIDRV directory. The _ReadMe.txt file describes most of the files in the <INSTALL-DIR>\ASSIDRV directory. The _NetHASP_Setup.txt file is a quick guide to installing the network (red) ESL on a Windows network. Please review these files for more detailed information

The remainder of this section deals with detailed information on ESL and is not required reading for a local installation.

The ESL is the security protection method employed by COADE. The ESL responds to queries by the software at various intervals. As long as the response from the ESL is as expected, the software continues to execute and operate normally. An invalid response from the ESL causes the software to return an error and terminate the program.

The parallel type ESL can be easily attached to the parallel port of the computer. The printer cable should then be attached to the other side of the ESL.

The USB type ESL can be easily attached to the USB port of the computer. Most computers will have several USB ports available and any can be used.

Note: Do not connect a Plant and a P&ID lock (two separate locks) to the same machine. The program will find the first lock and return its code. If the Plant program finds the P&ID lock, it will return an error. If the same machine is going to run both Plant and P&ID, then please contact COADE Sales Department (sales@coade.com) to request that both programs be licensed from the same lock.

The ESL contains the response to the program queries, and other client specific information. This information includes the company name and ESL number. Additional data may be stored on the ESL depending on the specific program and on the specific client.

Network ESLs must be attached to a machine on the network (this can be a workstation or th server). In order for the network to respond to software requests for the ESL, the NetHASP License Manager utility program must be running on the machine where the ESL is attached.

Notes on Network ESLs

There are advantages and disadvantages in utilizing a network ESL. The advantage is that many users have access (from a variety of computers) to the software from a single server.

Additional points for consideration are:

- Depending on the number of licenses allowed by the network ESL, some users may receive error messages when attempting to access the software. For example, assume the ESL has been configured to allow four simultaneous users. When the fifth user attempts to access the software an error message will be generated stating no licenses are available. The fifth user will not be able to access the software until one of the first four exits and releases a license.
- Due to the communication procedures between the workstations and the file server, memory access to the network ESL is much slower than to a local ESL. This access time delay only occurs once, when the software is first started after installation.
- As previously stated, there is no network specific version of the software. The software looks first for a local ESL, then if necessary for a network ESL. This allows both a network ESL and several local ESLs to be utilized on the same system. This transparent ESL access procedure allows a single version of the software to be used on the network, and on remote machines.
- It is suggested that only 70 to 80 percent of the desired licenses be assigned to a network ESL. The remaining 20 to 30 percent of the licenses would be assigned to local ESLs. This enables the local ESL to be moved between computers. Note that if all of the licenses are on the network ESL, a user must be logged into the network to access the software. A few local ESLs provide much greater operating flexibility.

ESL Installation

The following topics explain how to install the different types of hardware locks supported by CADWorx Plant.

Local Lock Installation

Hardware Lock Instructions for Windows XP/2003/Vista:

1. Double click on the file "..\ASSIDRV\HaspHLDriverSetup.exe" in Windows Explorer.
2. Follow the on screen instructions to install the HASP HL driver.

Note: Some steps are automatically performed by the installation program. They are listed here for your reference only. All files referenced here can be found in the <INSTALL-DIR>\ASSIDRV directory

Network Lock Installation

General information about Red Network ESLs.

1. The Red Network ESL should be setup by a person who is FAMILIAR with Windows based networks such as your Network Administrator. If you are not familiar with concepts such as Network Protocols, Broadcast, IP Addresses, TCP, UDP, and Windows Services, please do not attempt to setup the Red Network ESL.
2. The Red Network ESL does NOT have to be attached to the Network Server. COADE recommends attaching the Red Network ESL to a machine that is always up and running, can be re-booted without impact to users, and is not the primary machine for any user on the network.
3. If you install a COADE application on a network drive for many users to access, please make the ..\SYSTEM directory under the COADE application directory has read/write privileges for all users. COADE applications write temporary files to the ..\SYSTEM directory that pertain to the install. For CADWorx, the ..\SUPPORT directory also needs to have read/write privileges for all users.
4. If using other network protocols (NetBIOS or IPX), check Aladdin web site for instructions: http://www.aladdin.com/support/hasp/enduser.asp. The only protocol tested with COADE applications is TCP/IP.
5. The information provided in this file is a supplement to the NetHasp Documentation file called _NETHASP.PDF in the "..\ASSIDRV" directory. Please review _NETHASP.PDF file for any issues not addressed in this file.

NetHASP License Manager / NETHASP.INI Instructions

1. You must have ADMINISTRATOR level privileges to perform these steps.
2. Attach the Red network hardware lock to the parallel port or USB port of the machine.
3. Run the file "LMSETUP.EXE" on the machine where the hardware lock is attached. The LMSETUP.EXE file will be at ..\CADWorx\ASSIDRV\LMSETUP.EXE. Follow the on screen instructions and install the NetHASP License Manager as a "Service".
4. The NetHasp License Manager will automatically start every time the user re-boots the machine. This occurs because the NetHasp License Manager is setup as a start up service inside Windows 2000/XP/2003/Vista. To verify that this has occurred, go to Windows Start Button->Settings->Control Panel->Services Icon. In the Services dialog box, there should be an entry called "HASP Loader". The HASP Loader Service will start even if no one logs into the machine.
5. Determine what protocols the NetHasp License Manager is listening to. The NetHasp License Manager will display a message such as:

   "Server is listening to IPX : Loaded"

   "Server is listening to TCP/IP : Loaded"

   "Server is listening to NetBIOS : loaded"
6. The "loaded" means that this protocol is active on the machine. Hit the Activity Log pull down menu. This will bring up the activity log which will confirm what protocols are loaded. Do not be alarmed if the activity log says that "NHSRV.INI" file not found.

7. The Activity Log has a box in the top left that will allow you to bring up just the information specific to each protocol. Bring up the Activity Log for the protocol you want to use. Make sure there are no error messages here.
8. For TCP/IP Protocol, the Activity Log should say something like:

   "Server IP address: 111.111.111.111"

   "Server Host Name: PDC_NT01"
9. To speed up response time to the Network key, you may "unload" the any protocols that you are not planning on using with the NetHasp License Manager. This can be done from the Remove pull down menu.

NETHASP.INI modification for TCP/IP Protocol:

1. To configure this protocol, you must know the IP address of the machine where the Red Network hardware lock is attached. Example: 111.111.111.111
2. Modify the NETHASP.INI file located at ..\CADWorx\ASSIDRV\NETHASP.INI with the following changes.

   ```
   [NH_COMMON]
   NH_TCPIP = Enabled
   [NH_TCPIP]
   NH_SERVER_ADDR = 111.111.111.111
   NH_USE_BROADCAST = Disabled
   ```

   You may also have to modify other entries in [NH_TCPIP] Section depending on your network setup. The most common entries to modify are: NH_TCPIP_METHOD
3. Copy the NETHASP.INI file into AutoCAD EXE (ACAD.EXE) directory:

   Example: AutoCAD 2007 - (C:\Program Files\AutoCAD 2007)

   Example: AutoCAD 2008 - (C:\Program Files\AutoCAD 2008)

   Example: AutoCAD 2009 - (C:\Program Files\AutoCAD 2009)

   Example: AutoCAD 2010 - (C:\Program Files\AutoCAD 2010)

   Example: AutoCAD 2011 - (C:\Program Files\Autodesk\AutoCAD 2011)

   For CADWorx Design Review, copy the NETHASP.INI into the C:\Program Files\COADE\COADE CADWorx Design Review\ directory
4. Start CADWorx Plant.

Troubleshooting Network Locks

1. Verify that the NetHasp License Manager is running on the machine where the Red Network ESL is attached.
2. Verify the Protocols that NetHasp License Manager is broadcasting. Make sure that these protocols are installed on the Client machines.

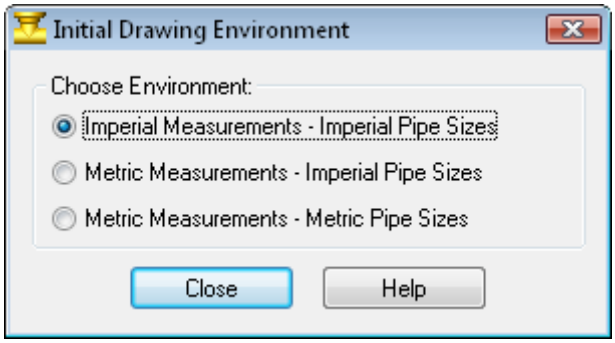
3. Install and run the Monitor Utility that is located in the <INSTALL-DIR>\ASSIDRV\NetHASPMonitorSetup.exe. If NetHASP Monitor cannot find the Red Network ESL, than the CADWorx Plant will not find the Red Network ESL

Startup Defaults

When starting CADWorx Plant for the first time, the user will be prompted for the system of measurement to be used in the drawing, i.e., imperial measurements with imperial pipe size, metric measurements with imperial pipe size, or metric measurements with metric pipe size. If imperial is chosen, then the default configuration file would be the IMPERIAL.CFG located in the <INSTALL-DIR>\SYSTEM directory. If metric measurements were chosen, then the default configuration file would be METRIC.CFG located in the <INSTALL-DIR>\SYSTEM directory.

These configuration files can be located anywhere desired. The user can also use a CADWorx P&ID configuration file. These configuration files can be chosen with the Restore button located in the Edit Configuration (see page 48) dialog.

This dialog will only appear once and the configuration file location and name is then stored in the registry.

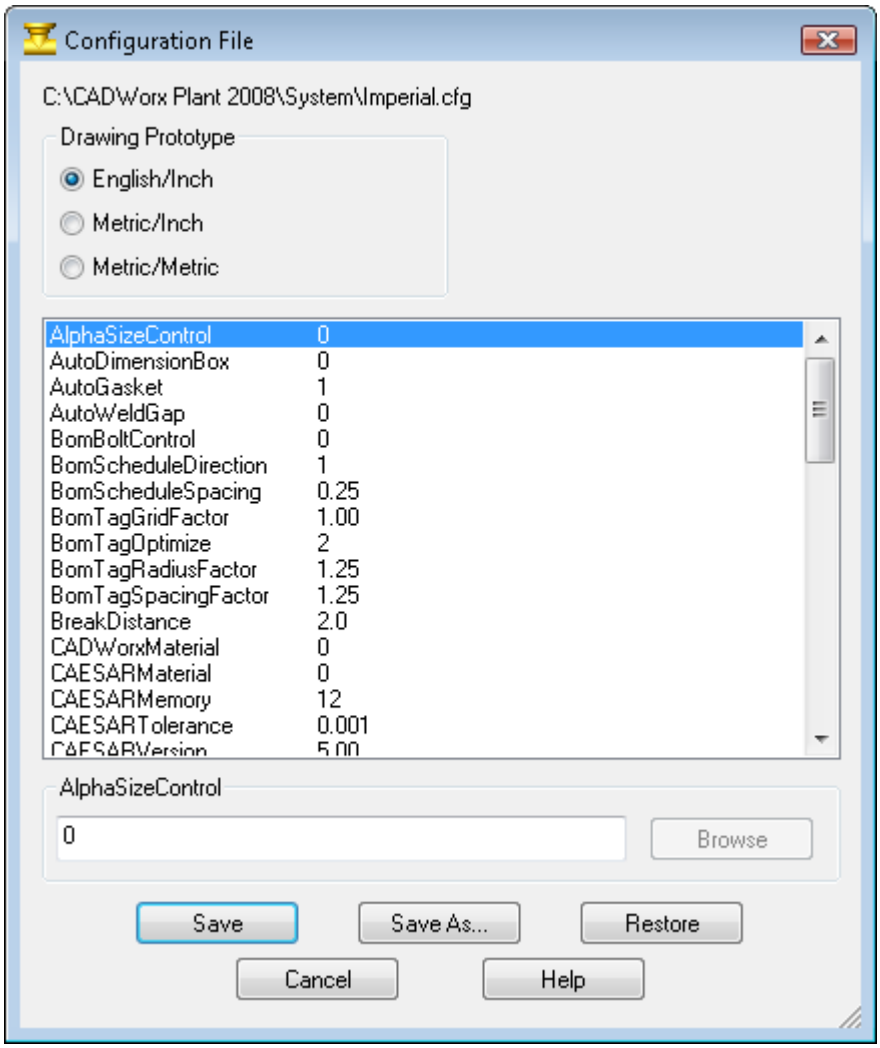


Choose Environment

Use these radio buttons to chose the initial drawing environment as explained above.

Configuration

The Configuration File dialog enables the user to modify the registry recorded configuration file. The default configuration file is <INSTALL-DIR>\SYSTEM\IMPERIAL.CFG (or METRIC.CFG for metric use). Any configuration file can be saved or restored as desired.



Drawing Prototype

This section allows the user to select the drawing environment type with the three radio buttons supplied here. When the Imperial - Inch radio button is selected, the program restores the <INSTALL-DIR>\SYSTEM\IMPERIAL.CFG. When the Metric-Inch or Metric-Metric radio button is selected, the program restores the <INSTALL-DIR>\SYSTEM\METRIC.CFG.

Note: The radio buttons only change the setting for the default configuration files (IMPERIAL.CFG or METRIC.CFG in the <INSTALL-DIR>\SYSTEM\ directory). If you are using a different configuration file, use the SystemMeasure (see page 66) variable below to change the drawing prototype setting.

List

This list box allows the user to select or highlight a variable that needs changing. Use the edit box below to change the variable after it has been selected. See the next topic, Startup Variables (see page 50) for more information on the items in this list.

Save

Use this button to save the current configuration file to its present location.

Save As

Use this button to save the current configuration file to another location.

Restore

Use this button to restore a saved configuration file to the current drawing environment. This will place the configuration file's location in the registry and it will be used the next time CADWorx Plant is initialized.

Cancel

Use this button to undo any action performed and to cancel out of this dialog.

Help

Use this button to display Help.

Startup Variables

These startup parameters provide default values each time the program is started. This file can be modified as desired for the required drawing defaults. The last configuration file is stored in the registry and will be used again and again until it has been changed to another with the *Restore* button.

The program accesses the configuration file whenever it is started. If this file does not exist, the program will prompt the user to select the initial drawing environment as explained earlier. If the default templates cannot be found, another dialog will appear indicating that the program will need to be re-installed.

The following lines of information will be displayed in the dialog and placed in this file. Each line below will have a description immediately following it outlining its purpose.

AlphaSizeControl

Default value: 0
Allowed values: 0, 1, 2, 3, 4, 5, 6, 7, 8, 9, 10

This variable controls how many decimal places are used in Alpha Size. Setting this variable to 0 (zero) will place no decimals in Alpha Size. Setting the variable to a value from 1 to 9 will place 1 to 9 decimal places in Alpha Size. Setting this variable to 10 (ten) will suppress trailing zeros in Alpha Size.

Example with value a of 0:

- For Imperial pipe size: 4 1/2" would be 4 1/2" or 6" would be 6"
- For Metric pipe size: 50 would be 50 or 60.3 would be 60

Example with value a of 3:

- For Imperial pipe size: 4 1/2" would be 4.500" or 6" would be 6.000"
- For Metric pipe size: 50 would be 50.000 or 60.3 would be 60.300

Example with value a of 10:

- For Imperial pipe size: 4 1/2" would be 4.5" or 6" would be 6"
- For Metric pipe size: 50 would be 50 or 60.3 would be 60.3

Note: If you want decimals to display in Alpha Size in Imperial or Metric, the ideal setting is 10.

AutoDimensionBox

Default value: 0
Allowed values: 0 (off) or 1 (on)

This variable controls whether boxes are used in dimensioned rolling offset during automatic dimensioning. When on, the program draws a box for the rolling offset. When off, the program draws pre version 3.0 type graphics.

AutoGasket

Default value: 1
Allowed values: 0 (off) or 1(on)

This variable sets automatic gaskets on or off. When on, the gasket will be placed immediately after placing a flanged component. When off, gaskets will have to be inserted manually.

AutoWeldGap

Default value: 0
Allowed values: 0, 1, 2, 3, or 4

This variable sets automatic weld gaps on or off. When turned on, the weld gaps are automatically placed on all ends of any buttweld or socket weld component placed into the drawing. When turned off, weld gaps will have to be inserted manually. Weld gaps can be manually placed in the drawing with the WELD (see page 123) command.

The are three settings which determine how weld gaps are controlled:

- 0 turns the automatic weld gaps off.
- 1 turns the automatic weld gaps on but only for buttweld components.
- 2 turns the automatic weld gaps on for both buttweld and socket weld components.
- 3 Same as 1 and weld gaps will be shown in the CADWorx (AutoISO) BOM schedule.
- 4 Same as 2 and weld gaps will be shown in the CADWorx (AutoISO) BOM schedule.

Note: The following information only applies to ISOGEN and weld gap components: If "Misc" BOM Item Type (see page 303) is set in CEDIT (see page 303) for weld gaps, ISOGEN will represent this weld gap as a "Field Fit Weld". Use ISOGEN Option Switch 22 to increase cut piece add on allowances for field fit welds.

Note: To create AutoISO input files (*.pmi) from a database, a setting of 3 or 4 must be used. A setting of 3 or 4 allows weld gaps to be written to a database. Weld gaps are required in the database to maintain continuity for the AutoISO Out function when using a database.

Note: A setting of 3 or 4 allows weld gaps to be written to a database. Use this setting to export weld gaps to a database (with DBFGEN or Live DB).

Note: If the InsertMode variable is set to 0, automatic weld gaps will NOT be placed regardless of the settings above.

BomBoltControl

Default value: 0
Allowed values: 0, 1, 2, or 3

This variable controls how bolts are placed in the CADWorx Plant AutoISO bill of material schedules and the CADWorx Plant ISOGEN material list.

- 0 places the quantity for a set as pre CADWorx Plant 2002 versions - i.e.

  QUANTITY = 4, ALPHA_SIZE = 3/4", LONG_DESC = (8) Studs with nuts.

- 1 places the quantity for total bolts and strips the set number from the long description - i.e.

  QUANTITY = 32, ALPHA_SIZE = 3/4", LONG_DESC = Studs with nuts.

- 2 places the quantity for a set, places the alpha size based on the flange size, and appends the bolt diameter to the long description - i.e.

  QUANTITY = 4, ALPHA_SIZE = 12", LONG_DESC = (8) 3/4" Studs with nuts.

- 3 places the quantity for total bolts, strips the set number from the long description, places the alpha size based on the flange size, and appends the bolt diameter to the long description - i.e.

  QUANTITY = 32, ALPHA_SIZE = 12", LONG_DESC = 3/4" Studs with nuts

Note: When BomBoltControl is set to 1 or 3 the long description of the bolt must contain the number of bolts in the set enclosed by parenthesis. The program will read the value in parenthesis to get the new quantity. The number in set is then automatically removed from the long description for display in the material list.

BomScheduleDirection

Default value: 1
Allowed values: 0 or 1

This variable controls whether the Bill of Material schedule will be placed in the upward direction (0) or in the downward direction (1).

BomScheduleSpacing

Default value: 0.25
Allowed values: Positive real

This variable provides the user the means of controlling the spacing on the Bill of Material schedule.

BomTagGridFactor

Default value: 1.0
Allowed values: Positive real

This variable controls the BOM tag grid density on the drawing. When the BomTagOptimize (see page 53) = 2, this variable should be 0.5, 1.0, 2.0 or 3.0.

BomTagOptimize

Default value: 2
Allowed values: 0, 1 or 2

This variable is used to control the tagging algorithm for the Bill of Material.

- 0 turns the tagging algorithm off.
- 1 turns the algorithm on and does not stack any tags.
- 2 turns the algorithm on and stacks flange, bolt and gasket tags.

BomTagRadiusFactor

Default value: 1.25
Allowed values: Positive real

This variable is used to control the radius of the tag bubbles. It will size the bubbles using this factor multiplied by the AutoCAD setvar TEXTSIZE. This factor can also be used in the stress isometric annotation routines.

BomTagSpacingFactor

Default value: 1.25
Allowed values: Positive real

This variable controls how far BOM bubbles are from the model or components within the drawing.

BreakDistance

Default value: 2.0
Allowed values: Positive real

This variable is the distance the Manual/Auto Break function uses.

PipeSupportSetting

Default value: 1
Allowed values: 0, 1, 2

This variable controls what layer the pipe support component is placed on.

- 0: The pipe support solid will be placed on a layer set from the CADWorx Layer Settings.
- 1: The pipe support solid will be placed on the same layer as the connecting pipe solid or elbow solid component.
- 2: The pipe support solid will be placed on a layer set from the CADWorx Layer Settings. The layer name will be derived by appending the component layer name and the PipeSupport layer name: Example: Pipe solid is on layer "_150", PipeSupport layer name is "SUPT", the pipe support solid will be on a layer "_150SUPT"

CADWorxMaterial

Default value: 0
Allowed values: 0 (off) or 1 (on)

This variable is used for the CADWorx Plant material generation control (System In).

When turned off, the program provides component descriptions in CADWorx Plant based on the material types in CAESAR II. It combines the component name (derived from the <INSTALL-DIR>\SYSTEM\PRGM.TBL) and the material type from CAESAR II.

When turned on, the program provides component descriptions in CADWorx Plant based on the specifications. It will ignore the material types from CAESAR II and place the description according to the currently set specification.

CAESARMaterial

Default value: 0
Allowed values: 0, 1 or 2

This variable is used for the CAESAR II material generation control (System out).

Note: CAESAR II version 3.23 will only use option 1 below.

- 0 provides extended (material database) material types in CAESAR II based on the pipe material in the spec under INFO (unless the component was created in CAESAR II originally).
- 1 provides basic material types in CAESAR II (1-17) corresponding to the component material derived from the long description in the spec.
- 2 provides extended (material database) material types in CAESAR II based on the actual component material derived from the long description in the specification.

CAESARMemory

Default value: 12.0
Allowed values: Positive real

This variable is used when it is necessary to import a large CAESAR II input model. Setting this option is not mandatory, but it might help if the import fails. The setting is multiplied by megabytes.

CAESARTolerance

Default value: 0.001
Allowed values: Positive real

This variable controls the tolerance for items that might not transfer correctly to CAESAR II. For example, hangers placed on a skewed line will sometimes fail and require a larger tolerance factor.

CAESARVersion

Default value: 5.20
Allowed values: 5.00, 5.10, 5.20

This variable controls the format of the CAESAR input file (*.c2) created by CADWorx.

This setting should match the appropriate CAESAR version that the (*.c2) file will be opened with.

DataBaseCodes

Default value: DBCODES.DAT
Allowed values: NONE, SPEC or codes file name

This variable controls the data code component entry system in CADWorx Plant.

- When set to NONE, the system is turned off.
- When set to SPEC, the system refers to code file as set by the specification currently being used. The file can then be set in each individual specification file. To set the file name, go to the Info (see page 259) dialog within Specification Editor (see page 254).
- When set to a valid and properly formatted code file, CADWorx Plant looks up the corresponding size-code combination from one data file and places the code in the component as it is drawn and is used by all specifications. The file will need to be located in the same directory as the specification file.

DataBaseCodesISOGEN

Default value: 0
Allowed values: 0 (OFF) or 1 (ON)

This variable controls how CADWorx Plant ISOGEN uses the DataBaseCodes (see page 55) system.

- 0 - does not use DataBaseCodes system.
- 1 - requires ISOGEN to use the DataBaseCodes system within CADWorx Plant.

When DataBaseCodesISOGEN is OFF, CADWorx Plant creates generic ITEM-CODEs for each run of ISOGEN. Components with identical long annotation and specification file name will be given the same ITEM-CODE. ITEM-CODEs will start at one (1) for each run of ISOGEN.

When DataBaseCodesISOGEN is ON, CADWorx Plant will create ITEM-CODEs based on the DataBaseCodes system.

Warning: If this control is set on and there is not a valid DataBaseCodes system established, the Bill of Material generation within ISOGEN will not be correct.

DimensionsExcludedSlope

Default value: 10.0
Allowed values: Positive real

This variable instructs the automatic dimensioning feature within CADWorx Plant not to dimension the vertical leg of slope pipe under the percent here. For example, if this variable was set to 5.0 and a pipe was sloped at 2% (rise 2 units – run 100 units), it would not have the vertical dimension and the associated hatched graphics indicating a rise. It would however have the horizontal dimension.

DimensionsExistingISOGEN

Default value: 0
Allowed values: 0, 1, or 2

This variable controls ISOGEN isometric representation of existing components.

- 0 : Existing components are exported with dotted lines and are dimensioned
- 1 : Existing components are exported with dotted lines and are not dimensioned
- 2 : Existing components are exported with solid lines and are dimensioned

Note: The individual Dimension Status setting from the ISOGEN Data dialog will overwrite this setting.

DrawingMode

Default value: 2D
Allowed values: 2D or 3D

This variable controls the startup drawing mode.

ElbowCenterLine

Default value: 0
Allowed values: 0 (off) or 1 (on)

This variable controls whether CADWorx Plant draws square elbow centerlines while drawing elbow type components. When on, this function draws an additional square centerline only. It will not draw square elbow graphics. The automatic isometric configuration dialog also has a toggle for this variable.

FittingMode

Default value: T
Allowed values: T or S

This variable controls which fitting mode is initially used at startup. The two options are threaded (T) and socket weld (S).

FittingWidth

Default value: 1.0
Allowed values: Positive real

This variable controls the appearance of all flanged, socket weld and threaded fittings. When set to 1.0, the program draws the fitting hubs or flange ODs at 100% of the width as specified in their data file. This variable is useful when drawing socket weld and threaded type hub fittings where the hubs overlap (0.7 is a good medium). The automatic isometric generator uses this variable extensively. The ISORATIO.TBL file located in the <INSTALL-DIR>\SYSTEM directory and is used to automatically apply different widths based on different sizes when the automatic isometric generator is used.

InsertMode

Default value: 2
Allowed values: 0, 1, or 2

This variable controls how components are inserted into pipe type components.

- 0 turns the auto insertion mechanism off.
- 1 turns the auto insertion mechanism on, but does not alert the user of specification or size mismatches.
- 2 is the same as 1 with alerts about size and specification mismatches.

Note: InsertMode (Auto Connect) does work in conjunction with Auto Route (see page 293) or Auto Weld Gap (see page 51).

ISOGENDefaultStyle

Default value: (none)
Allowed values: full path to ISOGEN.FLS in ISOGEN Style folder

This variable sets the ISOGEN style used by ISOGENOUT and ISOGENBATCH. Select the ISOGEN.FLS file in the ISOGEN Style folder.

Note: ISOGENOUT will not show the Style Select dialog and use the style set with this setting.

IsometricAppearance

Default value: 1
Allowed values: 0 (off) or 1 (on)

This variable controls different graphics for isometric drawings. When on, the program draws new graphics for isometrics and single line components (post version 2.1). When off, the program draws old graphics (pre version 2.1).

IsometricColor

Default value: 0
Allowed values: 0 through 255

This variable controls the color for automatically generated isometric drawings. The default value of 0 uses the color recorded in the specification.

LanguageFile

Default value: ENGLISH.DIC
Allowed values: Valid and properly formatted language file

This variable controls the language of all CADWorx dialogs and prompts. The specified file must be present in the <INSTALL-DIR>\SUPPORT directory for proper operation of CADWorx Plant.

LayerByLineNumber

Default value: 0
Allowed values: 0, 1, and 2

This variable controls what layers a component is placed on.

- 0 places the component on a layer which is the same name as the component's specification.
- 1 places the components on a layer which is the same name as the component line number. The line number system must be on and set to a valid line number or the program will use the 0 setting above. The centerline of the component will also be placed on the same layer.
- 2 is the same as 1 above but changes the centerline of the component to the linetype for the default center line layer.

LibraryDirectory

Default value: <INSTALL-DIR>
Allowed values: Valid directory name

This variable provides the location of the data files directory that are to be used with a particular specification. CADWorx Plant specifications contain a particular data file sub directory for each component. Those sub directories must be under this directory. Please refer to Modifying Specification and Data Files (see page 620) for more information.

Note: This directory can be set as READ-ONLY.

LineMode

Default value: D
Allowed values: S or D

This variable provides either single line (S) or double line (D) components at the startup of the drawing environment.

LineNumberSystem

Default value: 1
Allowed values: 0 (off), 1(on), 2 (static)

This variable controls the Line Number System. When on, the program uses the current size, spec and count number to tags the line number accordingly. When off, the line number has to be manually entered.

NodePlacement

Default value: 0
Allowed values: 0 (off) or 1(on)

This variable is used to automatically place nodes (an AutoCAD entity – not CAESAR II nodes) at key locations when the components are placed. For example, a tee would have 4 nodes placed – one at the center and three at all the different ends. These nodes can be used with the object snap mechanism within AutoCAD for proper component connections.

PipeDescriptionCat

Default value: 0
Allowed values: 0, 1, or 2

If set to 1 or 2, end conditions are appended to the long description and/or the DataBaseCodes (see page 55) when the component is drawn. This is done for the small bore pipe located on the fittings toolbar (PIPF command) and the nipple component. This setting is also used when a small-bore pipe is broken by an inserted component, with the router command, and spec change command.

- 0 : End types will not be added to any component.
- 1 : End types will be added to long descriptions and DataBaseCodes for pipe/nipples.
- 2 : End types will be added to long descriptions for pipe/nipples.
- 3 : End types will be added to long descriptions and DataBaseCodes for pipe.
- 4 : End types will be added to long descriptions for pipe.

The possible end descriptions are:

BBE	BEVEL x BEVEL
BOE-TOE	BEVEL x THREADED
BOE-POE	BEVEL x PLAIN
PBE	PLAIN x PLAIN
POE-TOE	PLAIN x THREADED
TBE	THREADED x THREADED

PipeSupportDirectory

Default value: <INSTALL-DIR>\PipeSupport
Allowed values: Valid directory name

This variable locates the pipe support parametric data files used with the current configuration file. Pipe support data files can be located anywhere on the local machine, local network or wide area network.

PipeSupportSetting

Default value: 1
Allowed values: 0, 1, 2

This variable controls what layer the pipe support component is placed on.

- 0: The pipe support solid will be placed on a layer set from the CADWorx Layer Settings.
- 1: The pipe support solid will be placed on the same layer as the connecting pipe solid or elbow solid component.
- 2: The pipe support solid will be placed on a layer set from the CADWorx Layer Settings. The layer name will be derived by appending the component layer name and the PipeSupport layer name: Example: Pipe solid is on layer "_150", PipeSupport layer name is "SUPT", the pipe support solid will be on a layer "_150SUPT"

SaveLastProfileUsed

Default value: 1
Allowed values: 0 (false) or 1 (true)

When a drawing file is double-clicked in Windows Explorer or if AutoCAD is started using an icon without specifying any profile, AutoCAD always loads the last profile used. Setting this value to 0 will prevent CADWorx from saving the last profile used i.e., CADWorx will only start if the proper icon is used.

SingleLinePipe

Default value: 0
Allowed values: 0 (off) or 1 (on)

This variable if turned on forces all pipe type components drawn (not components such as flanges and elbows) to be drawn single line.

SingleLineWidth

Default value: 0.02
Allowed values: Positive real

This variable sets the default width for a single line pipe. (default = 0.02)

SpecificationDirectory

Default value: <INSTALL-DIR>\SPEC
Allowed values: Valid directory name

This variable locates the specifications used with the current configuration file. Specifications can be located anywhere on the local machine, local network or wide area network.

Note: This directory can be set as READ-ONLY.

SpecSizeOverride

Default value: 1
Allowed values: 0 (off) or 1 (on)

This variable allows the user to override the spec and size when drawing components. If override is turned off, the user will not be able to draw out of spec or out of size.

SteelCopeBoltDistance

Default value: 0.5 (Imperial) 15 (Metric)
Allowed values: Positive real

This variable controls the distance used when two steel beams are coped using the bolted method.

SteelCopeWeldDistance

Default value: 0.0
Allowed values: Positive real

This variable controls the distance used when two steel beams are coped using the welded method.

SteelLibraryDirectory

Default value: <INSTALL-DIR>\Steel_I (<INSTALL_DIR>\Steel_M for metric)
Allowed values: Valid directory name

This variable locates the data file used for drawing Steel. The data files can be located anywhere on the local machine, local network or wide area network.

Note: This directory can be set as READ-ONLY.

SteelNeutralAxis

Default value: 0
Allowed values: 0, 1, 2 or 3.

This variable controls where the center lines for steel channels and angles are drawn.

- When this value is set to 0, the center line is drawn at the value specified in the data file. The channel and angle data files have a "X" column that is used to specify the neutral axis.
- When this value is set to 1, the center line is drawn at the mid point of the flange.
- When this value is set to 2, the center line is drawn on the inside of the flange. A setting of 2 should be used if channels and angles are coped with other members.
- When this value is set to 3, the center line is drawn on the outside of the flange. A setting of 3 should be used if channels and angles are mitered with other members.

SteelPartNumberFile

Default value: SamplePartNumbers.dat
Allowed values: Valid and properly formatted part number file

This variable controls the file that contains part numbers that can be automatically entered for every steel beam that is drawn. As a steel member is drawn, it looks in this data file for an entry which is the same name as the data file. This entry must be enclosed in square brackets. If it finds the entry, it then looks for a description which matches the name of the member being drawn. If found, it then retrieves the part number (numeric, alpha, or any combination) and places it to the right side of the entry. It then places this part number into the dialog and stores it in the member drawn into the drawing.

SteelRoundControl

Default value: 0
Allowed values: 0, 1, 2 or 3.

This variable controls how the values are read from the steel data files are interpreted.

- 0 turns the system off.
- 1 rounds **up** to the nearest multiple of the value specified in SteelRoundUnit.
- 2 rounds **down** to the nearest multiple of the value specified in SteelRoundUnit.
- 3 rounds **up or down** to the nearest multiple of the value specified in SteelRoundUnit.

SteelRoundUnit

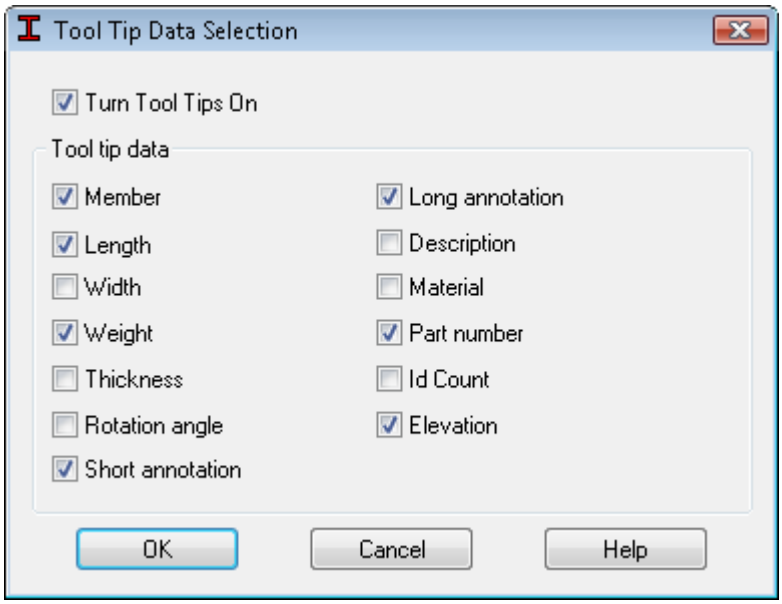
Default value: 0.0
Allowed values: Positive real.

This variable is used when the SteelRoundControl variable is turned on (1, 2 or 3). It is used to round the values read in the from the steel data files.

SteelToolTipSetting

Default value: 2435
Allowed values: Positive integer

This variable controls the items the user sees in the tool tip display, when the mouse hovers over a steel component. When enabled, the user is allowed to click the Settings button and select items to view on the tool tip as shown below.



SyncOnStartUp

Default value: 1
Allowed values: 0, 1, or 2

This variable controls the synchronize option on opening a drawing with an external database.

- 0 turns the synchronize option off when a drawing is loaded into AutoCAD.
- 1 asks the user whether the program should synchronize the drawing with the external database.
- 2 provides CADWorx Plant with the ability to synchronize the drawing with the external database without prompting the user.

Note: When calling the SYNC from the toolbar or command line, value 0 and 1 will prompt the user and value 2 will synchronize without any prompts.

SystemMeasure

Default value: 1
Allowed values: 0, 1, or 2

This variable is set according to the type of system that is currently being used by CADWorx Plant.

- 0 sets the metric system with metric input.
- 1 sets the imperial system with nominal inch input.
- 2 sets the metric system with nominal inch input.

TemplateDatabaseFile

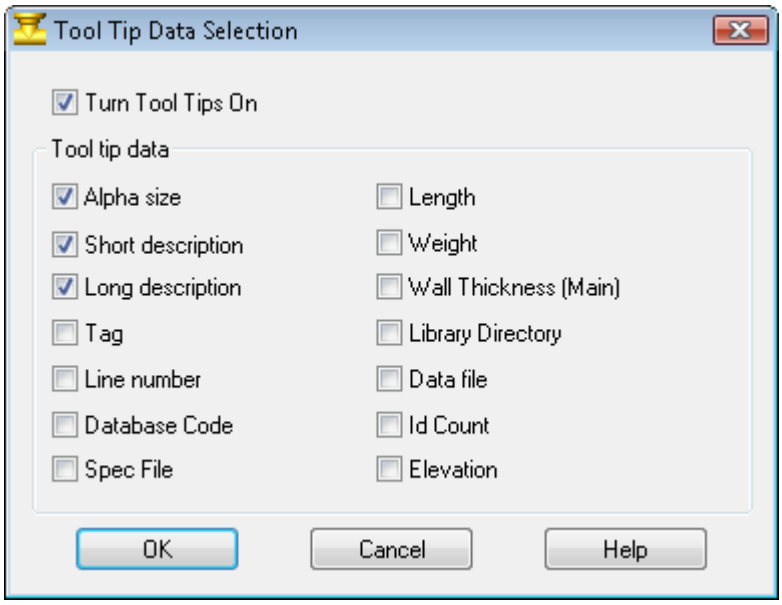
Default value: PIPE.MDB
Allowed values: Valid access database file name

This setting locates the database file that is used with the Bill of Material generator.

ToolTipSetting

Default value: 7
Allowed values: Positive integer

This variable controls the items the user sees in the tool tip displayed, when the mouse hovers over a component. When selected, the user is allowed to click on the Settings button and select items to view on the tool tip.



TopWorksDirectory

Default value: <INSTALL-DIR>\TopWorks
Allowed values: Valid directory name

This variable locates the top works data files used with the current configuration file. Top works data files can be located anywhere on the local machine, local network or wide area network.

TopWorksSetting

Default value: 1
Allowed values: 0, 1, 2, 3, 4

This variable controls what layer the top works component is placed on.

- 0: Top works will NOT be added to the valve even if a top works file is set in the specification file.
- 1: Top works will be added to the valve. The top works solid will be unioned with the valve solid to create one solid.
- 2: Top works will be added to the valve. The top works solid will be grouped with the valve solid and center line. The top works solid will be placed on the same layer as the valve solid.
- 3: Top works will be added to the valve. The top works solid will be grouped with the valve solid and center line. The top works solid will be placed on a layer set from the CADWorx Layer Settings.
- 4: Top works will be added to the valve. The top works solid will be grouped with the valve solid and center line. The top works solid will be placed on a layer set from the CADWorx Layer Settings. The layer name will be derived by appending the valve layer name and the TopWorks layer name: Example: Valve is on layer "_150", TopWorks layer name is "TOP", the top works solid will be on a layer "_150TOP"

UpgradedLayersSection

This setting is internally used by the program and should not be changed by the user. The program will automatically manage this setting.

It is set to 1 when the layer section of the configuration file has been updated with the new layers used with CADWorx 2009/2010 new features.

UseObjectsFromXrefs

Default value: 1
Allowed values: 0 (no) or 1 (yes)

This variable allows objects from Xrefs (eXternal References) to be used during the current selection process. When on, objects from Xrefs will be used for the function being run. When off, objects from Xrefs will not be used.

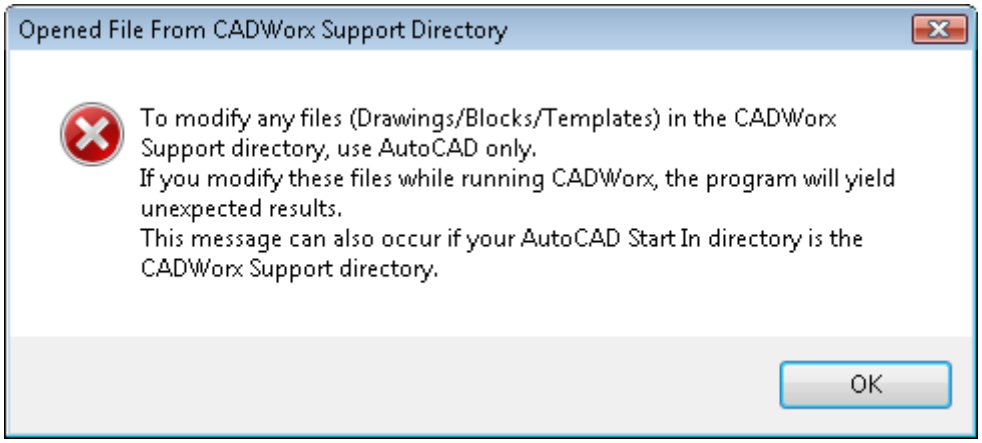
WeldDotSize

Default value: 1.0
Allowed values: Positive real

This variable will set the default weld dot size on single line buttweld piping components.

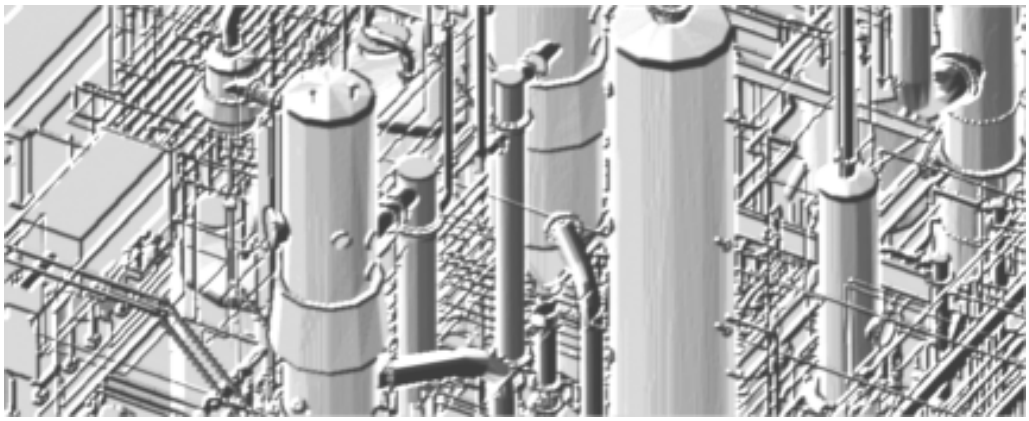
Support Directory

Drawings should not be opened in the <INSTALL-DIR>\SUPPORT sub-directory with CADWorx Plant. If a drawing is started in this directory, an error will be reported indicating that the program should not be run at this location. The purpose of this message is to stop CADWorx Plant from initializing or preparing the support drawings. Open a base AutoCAD session for modifying these drawings. For creating a base profile see the Creating a Base Profile (see page 619) topic.



CHAPTER 3

Toolbars



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Notes

This section provides details for all the different toolbars. Shown at the beginning of each section will be the toolbar itself. These toolbars show the primary buttons and will not show the secondary buttons (the flyouts). Unlike the AutoCAD toolbars, buttons will not rotate to the top (most recent used will not be on the top level). CADWorx Plant maintains the location of the buttons so that the user can later find the desired button.

The flyouts will mostly contain horizontal and vertical complements of the primary program icon. The component routines located on the flyouts are primary for 2D graphics. A tee can be drawn with the primary icon and it will be represented as laying flat on the ground. The program function name is TESW (TEe Straight Welded).

If the tee were placed on the ground setting on it branch connection, then the user would need to use the HTESW (Horizontal TEe Straight Welded) program function. The program calls this a horizontal placed tee. This function would ask the user if the branch should point up or down. This might not be considered a horizontal tee, but it is a method that the program uses to name all these different commands.

If the tee were placed on the ground setting on one of it main run connections, then the user would need to use the VTESW (Vertical TEe Straight Welded) program function. This is called a vertical placed tee. This function would just ask for a direction of the branch run and then draw the main run pointing up and down.

Accessing and Docking the Toolbars

To access the CADWorx Plant toolbars list from the AutoCAD pulldown menu, select Plant>Toolbars. From this list, the user can select the names of the toolbars that they want to use. They will be placed in the drawing environment for placement or docking.

Another way of loading the toolbars is to right click on a blank area of the existing menu. This displays a context menu for choosing toolbars for a designated menu. CADWorx Plant will have two menus associated with it -- an alternate menu (PIPE_ALT.MNU) and a flyout menu (PIPE_FLY.MNU). The user should use the alternate menu for loading toolbars unless they have some specific reason for choosing one from the flyout menu.

The user could also type TOOLBAR at the command line, select the PIPE_ALT menu, and then choose the toolbars they wish to use.

The user can dock the toolbars where they prefer. They will remain there until they are moved to another location or removed. Please refer to the AutoCAD documentation for instructions on docking the toolbars.

Note: The user should hold the control key down if they do not want a toolbar to dock.

Buttweld

Plant menu: ▶ **Toolbars** ▶ **Buttweld**



Note: The arrow in lower right corner of button indicates a flyout for more options.

Pipe

Command line: PIPW

See Appendix D type 1 (see page 635) prompts for this component. Special options available in this command include *TOP* and *BOP* which start the pipe at the top of pipe and bottom of pipe respectively.

90 Degree Long Radius Elbow

Command line: 90LR **(also** H90LR**)**

See Appendix D type 2 (see page 636) prompts for this component.

90 Degree Short Radius Elbow

Command line: 90SR **(also** H90SR**)**

See Appendix D type 2 (see page 636) prompts for this component.

45 Degree Elbow

Command line: 45LR **(also** H45LR **or** V45LR**)**

See Appendix D type 3 (see page 637) prompts for this component. Special options available in this command include *Opposite* direction that allows the user to continue from an existing 45 degree run. If coming off a 45 degree rotated pipe, use this option and the Osnap, NEArest of, and select back along the centerline of pipe. If coming off another 45 elbow use this option and the Osnap, INT of, and select the corner of the existing 45 elbow.

90 Degree Reducing Elbow

Command line: 90RD **(also** H90RD**)**

See Appendix D type 2 (see page 636) prompts for this component. Other options included with this function are the *Small* and *Large* end options.

Straight Tee

Command line: TESW **(also** HTESW **or** VTESW**)**

See Appendix D type 5 (see page 638) prompts for this component.

Reducing Tee

Command line: TERW **(also** HTERW **or** VTERW**)**

See Appendix D type 5 (see page 638) prompts for this component.

180 Degree Long Radius Return

Command line: 18LR **(also** H18LR **or** V18LR**)**

See Appendix D type 4 (see page 638) prompts for this component.

180 Degree Short Radius Return

Command line: 18SR **(also** H18SR **or** V18SR**)**

See Appendix D type 4 (see page 638) prompts for this component.

Concentric and Eccentric Reducer

Command line: CONC **(also** ECC**)**

See Appendix D type 1 (see page 635) prompts for this component. This routine will reverse the main and reduction sizes automatically after execution.

Cap

Command line: CAPW

See Appendix D type 1 (see page 635) prompts for this component.

4-Way Straight Cross

Command line: CRSW **(also** HCRSW**)**

See Appendix D type 6 (see page 639) prompts for this component.

4-Way Reducing Cross

Command line: CRRW **(also** HCRRW **or** VCRRW**)**

See Appendix D type 6 (see page 639) prompts for this component.

Lateral

Command line: LATW **(also** HLATW **or** VLATW**)**

See Appendix D type 7 (see page 640) prompts for this component.

Miscellaneous

Mitered Pipe

Command line: MPIP

This command is used for drawing mitered pipe. The user must first draw a 2D polyline (not a 3D polyline) that will be converted to mitered pipe. A miter will be placed at each vertex of the polyline. The miter command can also be used for pipe with bends. Draw a polyline with square corners and then fillet the corners using the Poly option of the Fillet command. After converting the polyline to mitered pipe, erase the miter joints.

```
Command: MPIP
4" MITERED PIPE, S/40 SMLS, ASTM A-106 GR B
Select polyline:
```

ERRORS:

There must be a change of direction at each vertex. If two polylines segments form a straight line, the routine will not work and the user will get the following error message.

```
An extra vertex has been detected in one of the straight line segments...
Use pedit to find, then edit and straighten...
```

The user cannot fit or spline the polyline to be converted. The polyline must be composed of straight segments or the user will get the following error message.

```
Pline has been fit or spline...
```

As mentioned before, the user cannot use a 3D polyline. If the user does the following error message will appear.

```
Cannot use 3dpoly...
```

Rolled Pipe

Command line: SPIP

This command is used for rolled pipe as in an flex connection or a hose. Draw any Polyline desired and then use the Fit or Spline option of PEDIT on the polyline. The polyline is now ready to be converted into rolled pipe in either 2D or 3D Mode. This routine needs enough, but not too many vertices in the polyline. The user cannot use a 3D polyline.

Command: **SPIP**
4" ROLLED PIPE, S/40 SMLS, ASTM A-106 GR B
Select polyline:

ERRORS:

Vertices in a straight line might generate the following error message. Use the Edit option of PEDIT command to check the vertices, if the user finds that some are in a straight line, use the Straighten option of PEDIT to fix. Tight bends will also cause problems. If this is the case, the user will have to make the bends with a larger radius. If either of these situations exist, the user may receive the following error message.

Error, fit pline probably has straight tangent ends or a too small radius...
Use pedit to find, then edit and straighten...

The polyline needs to be curve fitted. If it is not, the user will get the following error message. Use the Fit or Spline of the PEDIT command to fix.

Pline has not been fit or spline...

As mentioned above the user cannot use a 3D polyline.

Cannot use 3dpoly...

Adjustable Bend

Command line: BENT

Command: **BENT**
Enter an option [**Trimmed elbow/Select arc/Enter radius**] <Enter radius>:

Trimmed elbow

This would be similar to the insertion of a 90 elbow except that the bend will be drawn at the exact pick directions.

Pick elbow's intersection: *pick point*
Select first direction: *pick direction toward first buttweld*
Select second direction: *pick direction toward other buttweld*

If this component is greater than 45 degree, then the following prompt will appear.

Enter option [**Long radius/Short radius**]:

Short radius

Select this option for short radius 90 degree elbow or 180 degree return according to the current specification.

Long radius

Select this option for long radius 90 degree elbow or 180 degree return according to the current specification.

Select arc

This option allows the user to select an existing arc created with AutoCAD. The selection has to be an arc and not a pline with a radius or bulge factor.

Select arc: *select an existing arc*
Tangent length: *enter a real (ex: 2)*

Enter radius

This is the conventional way of creating an adjustable bend. This command acts just like the AutoCAD ARC command (works counter clockwise only). In other words, use it like the ARC command (Center, Start, End).

Center point of bent: *pick point (the center of the arc)*
First end point of bend: *pick start end*
Enter an option [**Angle/Degrees/Second end point bend**] <Second end point bend> :

Angle *(enter a real, ex: 180)*

Use this option when supplying angles for the adjustable bend. For example, after selecting an endpoint at 90 degrees (from center) for the start, enter an angle of 180 would draw a 90 degree bend.

Degrees *(enter a real, ex: 180)*

Use this option when supplying a bend with a total degree. For example, after selecting an endpoint at 90 degrees (from center) for the start, enter a total of 180 would draw a 180 degree bend.

Second end point bend

This requires the user to pick an end point.

The last prompt for this option requires the user to enter a tangent length (straight pieces of pipe commonly required with a machine bend). The user could also enter 0 here.

Tangent length: *enter a tangent length (ex: 2")*

Rolled

Rolled 90 Degree Elbows

Command line: R90LR **(also** R90SR**)**

This command allows the user to insert the following fittings rolled at a 45 degree angle. This will only work on welded fittings.

Command: **R90LR**
4" ELL, 90 LR S/40, ASTM A-234 GR WPB
Pick start point or [**Corner**] <last point>:

Corner

This option starts an elbow at the center point.

Pick corner point: *pick point*
Pick first direction: *pick direction*
Pick second direction: *pick direction*
Enter direction [**Up/Down**] <Up>:

Down

Select if elbow is desired in the downward or the negative Z direction.

Up

Select if elbow is desired in the upward or positive Z direction.

Pick start point

This option starts an elbow at the centerline of the buttweld. An enter will cause it to start at the last placed component.

Pick corner direction: *pick direction*
Pick direction: *pick direction*
Enter direction [**Up/Down**] <Up>:

Down

Same as above.

Up

Same as above.

Note: The centerline will be placed out of plane. For modification (even after using Convert Centerline) the user will be required to use the AutoCAD command UCS entity option to obtain the proper plane for modification.

Rolled 45 Degree Elbow

Command line: R45LR

This command allows the user to insert the following fittings rolled at a 45 degree angle. This will only work on welded fittings.

Command: **R45LR**
4" ELL, 45 LR S/40, ASTM A-234 GR WPB
Pick start point or [**Corner**] <last point>:

Corner

This option starts a 45 degree elbow at the corner point.

Pick corner point: *pick point*
Pick first direction: *pick direction*
Pick second direction: *pick direction*
Enter direction [**Up/Down**] <Up>:

Down

Select if elbow is desired in the downward or the negative Z direction.

Up

Select if elbow is desired in the upward or positive Z direction.

Pick start point

This option starts a 45 degree elbow at the buttweld end. An enter will cause it to start at the last placed component.

Pick corner direction or [**Opposite**]:

Opposite

This option is handy if the user is coming off an existing 45 degree run. If coming off a pipe use Osnap, NEArest of, and select centerline of existing pipe.

Pick opposite direction: *pick opposite direction desired*
Pick other elbow direction: *pick point*
Enter direction [**Up/Down**] <Up>:

Down

Same as above.

Up

Same as above.

Select corner direction

This option allows the user to select the corner direction.

Pick corner direction: *pick direction*
Enter direction [**Up/Down**] <Up>:

Down

Same as above.

Up

Same as above.

Rolled Tee

Command line: RTESW

This command allows the user to insert the following fittings rolled at a 45 degree angle. This will only work on welded fittings.

Command: **RTESW**
4" TEE, STR. S/40, ASTM A-234 GR WPB
Pick start point or [**Center**] <last point>:

Center

This option starts a tee at the center point.

Pick center point: *pick point*
Pick main end direction: *pick direction*
Pick branch end direction: *pick direction*
Enter direction [**Up/Down**] <Up>:

Down

Select if branch is desired in the downward or the negative Z direction.

Up

Select if branch is desired in the upward or positive Z direction.

Pick start point

This option starts the tee at the main buttweld end.

Pick start point or [Center] <last point>:
Pick main end direction:
Pick branch end direction:
Enter direction [**Up/Down**] <Up>:

Down

Same as above.

Up

Same as above.

11.25 Degree Elbow

Command line: 11LR **(also** H11LR **or** V11LR**)**

See Appendix D type (see page 636) 3 prompts for this component. Special options available in this command include *Opposite* direction that allows the user to continue from an existing 11.25 degree run. If coming off a 11.25 degree rotated pipe, use this option and the Osnap, NEArest of, and select back along the centerline of pipe. If coming off another 11.25 elbow use this option and the Osnap, INT of, and select the corner of the existing 11.25 elbow.

22.5 Degree Elbow

Command line: 22LR **(also** H22LR **or** V22LR**)**

See Appendix D type 3 (see page 636) prompts for this component. Special options available in this command include *Opposite* direction that allows the user to continue from an existing 22.5 degree run. If coming off a 22.5 degree rotated pipe, use this option and the Osnap, NEArest of, and select back along the centerline of pipe. If coming off another 22.5 elbow use this option and the Osnap, INT of, and select the corner of the existing 22.5 elbow.

Wye

Command line: YB **(also** HYB **or** VYB**)**

See Appendix D type 8 (see page 640) prompts for this component.

Flanges

Plant menu: ➤ **Toolbars** ➤ **Flanges**



For convenience and ease of drawing flanges, CADWorx Plant switches the default from end connection (buttweld, socket engagement, etc.) to the face of flange start point after a gasket has been inserted. For example, draw a flange starting from the buttweld end to the face. Then draw a gasket (if AutoGasket (see page 51) is on, one will be automatically inserted). After re-selecting the flange again, the following prompt will appear. The pick start point here is referring to the face of the flange. The user has the option to start a the buttweld again if desired.

Pick start point or [**Buttweld end**] <last point>:

A return will put the face of the flange at the insert of the gasket. Now pick a direction. If the user picks the flange again, the default will be the end connection. This will be true for the weld neck, slip-on, socket, and threaded flanges.

Weld Neck Flange

Command line: FLGW

See Appendix D type 1 (see page 635) prompts for this component.

Slip-On Flange

Command line: FLGO

See Appendix D type 1 (see page 635) prompts for this component.

Blind Flange

Command line: FLGB

See Appendix D type 1 (see page 635) prompts for this component.

Socket Type Flange

Command line: FLGS

See Appendix D type 1 (see page 635) prompts for this component.

Threaded Flange

Command line: FLGT

See Appendix D type 1 (see page 635) prompts for this component.

Lap Joint Flange

Command line: FLGL

See Appendix D type 1 (see page 635) prompts for this component.

Stub End

Command line: STB

See Appendix D type 1 (see page 635) prompts for this component. This component also provides a *Length* option that can be used to make it any length desired.

Long Weld Neck

Command line: LWN

See Appendix D type 1 (see page 635) prompts for this component. This component also provides a *Length* option that can be used to make it any length desired.

Reducing Slip-On Flange

Command line: FLRO

See Appendix D type 1 (see page 635) prompts for this component.

Reducing Threaded Flange

Command line: FLRT

See Appendix D type 1 (see page 635) prompts for this component.

Reducing Weld Neck Flange

Command line: FLRW

See Appendix D type 1 (see page 635) prompts for this component.

Flanged/Buttweld Valves

Plant menu: ➤ **Toolbars** ➤ **Flgd/BW Valves**



Note: The arrow in the lower right corner of button indicates a flyout for more options.

Buttweld valves can be created by placing a negative value within its data files for the flange outside diameter. Example buttweld data files supplied with CADWorx Plant have ½" flange thickness. The flange OD is also 1" larger than the actual pipe diameters and then the actual diameter is changed to a negative. An data file example would be...

```
( "NOM"     "OD"    "FLG_OD"   "FLG_THK"   "LENGTH"    "WGT"    )
=============================================================
( 8.0000    8.6250   -9.6250     0.5000      11.5000    1340.00 )
```

Where flanges don't exist, such as a butterfly valve, the flange thickness can be changed to 0.0 and the flange outside diameter to the actual pipe outside diameter. This provides a wafer type valve where no flange exist. An data file example would be...

```
( "NOM"     "OD"    "FLG_OD"   "FLG_THK"   "LENGTH"    "WGT"    )
=============================================================
( 8.0000    8.6250    8.6250     0.0000      11.5000    1340.00 )
```

Also, the use of the sanitary type components can be used to create buttweld valves. These components can also be used to create flanged by welded, male by female valves. Once a sanitary component is created, it can be set as an optional component and the button can be dragged over to this toolbar or any other toolbar.

Gate Valve

Command line: GATF

See Appendix D type 1 (see page 635) prompts for this component. Special options included with this component include an alternate *Center* starting location.

Ball Valve

Command line: BALF

See Appendix D type 1 (see page 635) prompts for this component. Special options included with this component include an alternate *Center* starting location.

Globe Valve

Command line: GLBF

See Appendix D type 1 (see page 635) prompts for this component. Special options included with this component include an alternate *Center* starting location.

Plug Valve

Command line: PLUF

See Appendix D type 1 (see page 635) prompts for this component. Special options included with this component include an alternate *Center* starting location.

Check Valve

Command line: CHKF

See Appendix D type 1 (see page 635) prompts for this component. Special options available in this command include *Reverse* direction that allows the user to select the direction of the valve that will position the flow indicator pointing in the opposite direction.

Butterfly Valve

Command line: BFYF

See Appendix D type 1 (see page 635) prompts for this component. Special options included with this component include an alternate *Center* starting location.

Diaphragm Valve

Command line: DIAF

See Appendix D type 1 (see page 635) prompts for this component. Special options included with this component include an alternate *Center* starting location.

Rotary Valve

Command line: ROTF

See Appendix D type 1 (see page 635) prompts for this component. Special options included with this component include an alternate *Center* starting location.

Dual Check Valve

Command line: DCHF

See Appendix D type 1 (see page 635) prompts for this component. Special options included with this component include an alternate *Center* starting location.

Control Valve

Command line: CNTF

See Appendix D type 1 (see page 635) prompts for this component. This component also provides a *Length* option that can be used to make it any length desired.

3-Way Valve

Command line: 3VF **(also** H3VF **or** V3VF**)**

See Appendix D type 5 (see page 635) prompts for this component.

4-Way Valve

Command line: 4VF **(also** H4VF **)**

See Appendix D type 6 (see page 635) prompts for this component.

Relief Valve

Command line: RELF **(also** HRELF **or** VRELF**)**

Command: **RELF**
Pick start point or [**Length/Relief/Corner**] <last point>:

Pick start point

This option starts the relief valve at the face at the pressure side or inlet.

Pick corner direction: *pick point*

If RELF is used, the user selects the relief end direction.

Pick relief end direction: *pick direction*

If HRELF or VRELF is used, the user selects either the Up or Down directions.

Enter direction [**Up/Down**] <Up>:

Down

Select if the relief valve is desired in the downward or the negative Z direction.

Up

Select if the relief valve is desired in the upward or positive Z direction.

Length

This option allows the user to specify the length of both the pressure and the relief side of the relief valve. This is useful since relief valve dimensions and orifice sizes vary among manufacturers.

Enter length of pressure side: *enter length*
Enter length of relief side: *enter length*
Pick pressure end point: *pick point*
Pick corner direction: *pick direction*

If RELF is used, the user selects the relief end direction.

Pick relief end direction: *pick direction*

If HRELF or VRELF is used, the user selects either the Up or Down directions.

Enter direction [**Up/Down**] <Up>:

Down

Same as above.

Up

Same as above.

Relief

This option starts the relief valve at the centerline of the face of flange at the relief side or outlet. This option is not available in HRELF or VRELF.

Pick relief end point <last point>: *pick point*
Pick corner direction: *pick direction*
Pick pressure end direction: *pick direction*

Corner

This option starts the relief valve at the corner point.

Pick corner point: *pick point*
Pick pressure end direction: *pick direction*

If RELF is used, the user selects the relief end direction.

Pick relief end direction: *pick direction*

If HRELF or VRELF is used, the user selects either the Up or Down directions.

Enter direction [**Up/Down**] <Up>:

Down

Same as above.

Up

Same as above.

Angle Valve

Command line: ANGF **(also** HANGF**)**

See Appendix D type 2 (see page 636) prompts for this component.

Lined

Plant menu: ▶ **Toolbars** ▶ **Lined**



Note: The arrow in the lower right corner of button indicates a flyout for more options.

The user must set the LINED.SPC default specification that is supplied with CADWorx Plant for the icons to work on the toolbar.

Located in the flanged component data files are columns (FLGS) which control placement of flanges on these components. The FLGS parameter is an integer that is a bit wise control factor. If the number is 1, it draws a flange on the first end of the component only. If it is 2, it draws a flange on the second end only. If the number is 4, it places a flange on the third end (such as a tee). If the flanges are needed on all ends, the number will be the sum of the first, second and third i.e 1 + 2 + 4 = 7. If the number were 3, it would draw flanges on the first and second end (1 + 2). 6 would draw flanges on the second and third end (2+ 4). A tee's branch connection is the second end. This provides many other uses for these flanged components.

Flanged Pipe

Command line: PIPFL

See Appendix D type 1 (see page 635) prompts for this component. This component uses the sanitary ends conventions. Please refer to the Sanitary Toolbar (see page 94) for more information on how this component can be used.

Flanged 90 Degree Elbow

Command line: 90FL **(also** H90FL**)**

See Appendix D type 2 (see page 636) prompts for this component.

Flanged 45 Degree Elbow

Command line: 45FL **(also** H45FL **or** V45FL**)**

See Appendix D type 3 (see page 637) prompts for this component. Special options available in this command include *Opposite* direction which allows the user to continue from an existing 45 degree run. If coming off a 45 degree rotated pipe, use this option and the Osnap, NEArest of, and select back along the centerline of pipe. If coming off another 45 elbow use this option and the Osnap, INT of, and select the corner of the existing 45 elbow.

Flanged Straight Tee

Command line: TEFL **(also** HTEFL **or** VTEFL**)**

See Appendix D type 5 (see page 638) prompts for this component.

Flanged Reducing Tee

Command line: TERE **(also** HTERE **or** VTERE**)**

See Appendix D type 5 (see page 638) prompts for this component.

Flanged Concentric and Eccentric Reducer

Command line: CNFL **(also** ECFL**)**

See Appendix D type 1 (see page 635) prompts for this component.

Note: This routine will reverse the main and reduction sizes automatically after execution.

Flanged Wye

Command line: YFL **(also** HYFL **or** VYFL**)**

See Appendix D type 8 (see page 640) prompts for this component.

Flanged Straight Cross

Command line: CRSE **(also** HCRSE**)**

See Appendix D type 6 (see page 639) prompts for this component.

Flanged Reducing Cross

Command line: CRRE **(also** HCRRE **or** VCRRE**)**

See Appendix D type 6 (see page 639) prompts for this component.

Flanged Lateral

Command line: LATE **(also** HLATE **or** VLATE**)**

See Appendix D type 7 (see page 640) prompts for this component.

Spacer/Bleed Ring

Command line: BLD

See Appendix D type 1 (see page 635) prompts for this component.

Reducing Slip-On Flange

Command line: FLRO

See Appendix D type 1 (see page 635) prompts for this component.

Blind Flange

Command line: FLGB

See Appendix D type 1 (see page 635) prompts for this component.

Fiberglass

Plant menu: ➤ **Toolbars** ➤ **Fiberglass**



Note: The arrow in the lower right corner of the button indicates a flyout for more options.

These components are only a collection of other available components within CADWorx Plant. They have been collected in this toolbar for convenience. The user must set the FRP.SPC default specification that is supplied with CADWorx Plant for the icons to work on the toolbar.

Pipe

Command line: PIPW

See Appendix D type 1 (see page 635) prompts for this component. Special options available in this command include *TOP* and *BOP* which start the pipe at the top of pipe and bottom of pipe respectively.

Threaded/Socket 90 Degree Elbow

Command line: 90S **(also** H90S**)**

See Appendix D type 2 (see page 636) prompts for this component.

Threaded/Socket 45 Degree Elbow

Command line: 45S **(also** H45S **or** V45S**)**

See Appendix D type 3 (see page 637) prompts for this component. Special options available in this command include *Opposite* direction which allows the user to continue from an existing 45 degree run. If coming off a 45 degree rotated pipe use this option and the Osnap, NEArest of, and select back along the centerline of pipe. If coming off another 45 elbow use this option and the Osnap, INT of, and select the corner of the existing 45 elbow.

Threaded/Socket Straight Tee

Command line: TESS **(also** HTESS **or** VTESS**)**

See Appendix D type 5 (see page 638) prompts for this component.

Threaded/Socket Reducers

Command line: SWGCS **(also** SWGES**)**

See Appendix D type 1 (see page 635) prompts for this component.

Threaded/Socket Straight Cross

Command line: CRSS **(also** HCRSS**)**

See Appendix D type 6 (see page 639) prompts for this component.

Threaded/Socket Lateral

Command line: LATS **(also** HLATS **or** VLATS**)**

See Appendix D type 7 (see page 640) prompts for this component.

Full Coupling

Command line: CPS **(also** VCPS**)**

See Appendix D type 1 (see page 635) prompts for this component.

Threaded/Socket Cap

Command line: CAPS

See Appendix D type 1 (see page 635) prompts for this component.

Threaded/Socket Insert/Bushing

Command line: BUSS

See Appendix D type 2 prompts for this component.

Soc-O-Let

Command line: SOL **(also** VSOL**)**

See Appendix D type 2 prompts for this component.

Blind Flange

Command line: FLGB

See Appendix D type 1 (see page 635) prompts for this component.

Socket Type Flange

Command line: FLGS

See Appendix D type 1 (see page 635) prompts for this component.

Flanged 90 Degree Elbow

Command line: 90FL **(also** H90FL**)**

See Appendix D type 2 (see page 636) prompts for this component.

Flanged 45 Degree Elbow

Command line: 45FL **(also** H45FL **or** V45FL**)**

See Appendix D type 3 (see page 637) prompts for this component. Special options available in this command include *Opposite* direction which allows the user to continue from an existing 45 degree run. If coming off a 45 degree rotated pipe, use this option and the Osnap, NEArest of, and select back along the centerline of pipe. If coming off another 45 elbow use this option and the Osnap, INT of, and select the corner of the existing 45 elbow.

Flanged Straight Tee

Command line: TEFL **(also** HTEFL **or** VTEFL**)**

See Appendix D type 5 (see page 638) prompts for this component.

Flanged Concentric and Eccentric Reducer

Command line: CNFL **(also** ECFL**)**

See Appendix D type 1 (see page 635) prompts for this component.

Note: This routine will reverse the main and reduction sizes automatically after execution.

Sanitary

Plant menu: ▶ Toolbars ▶ Sanitary

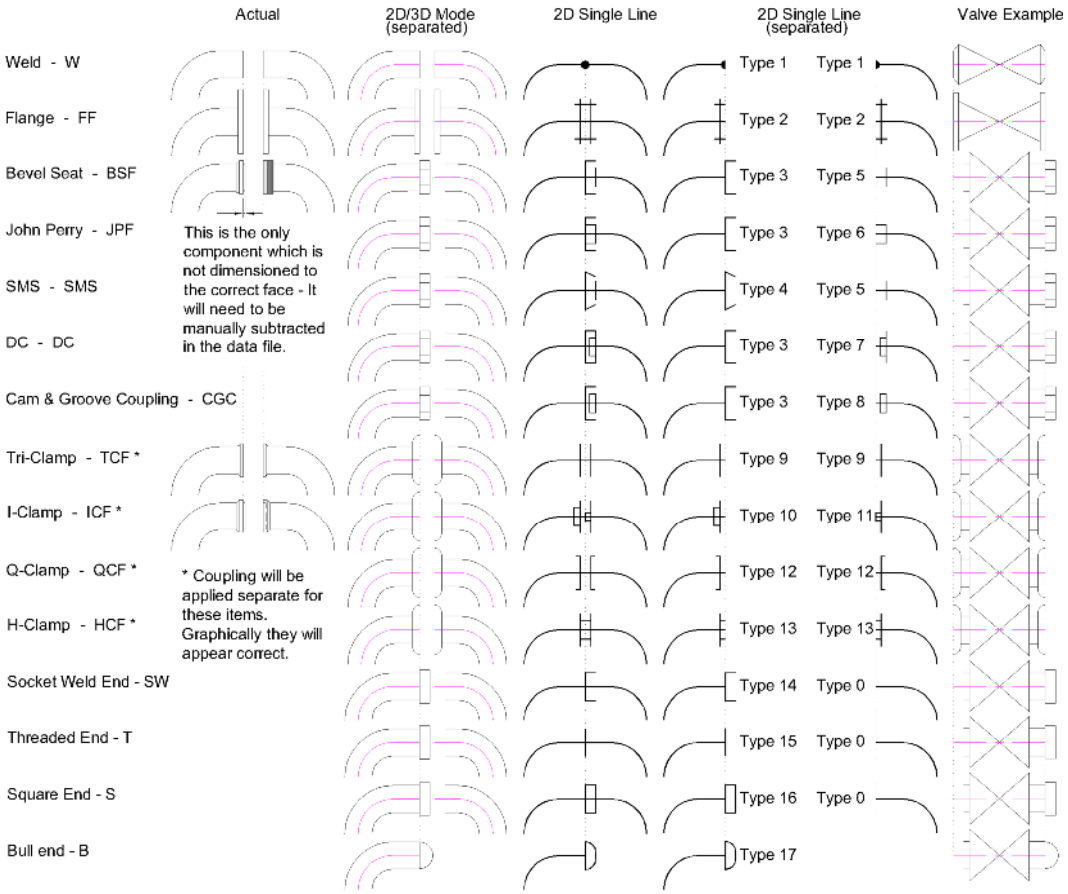


Note: The arrow in the lower right corner of the button indicates a flyout for more options.

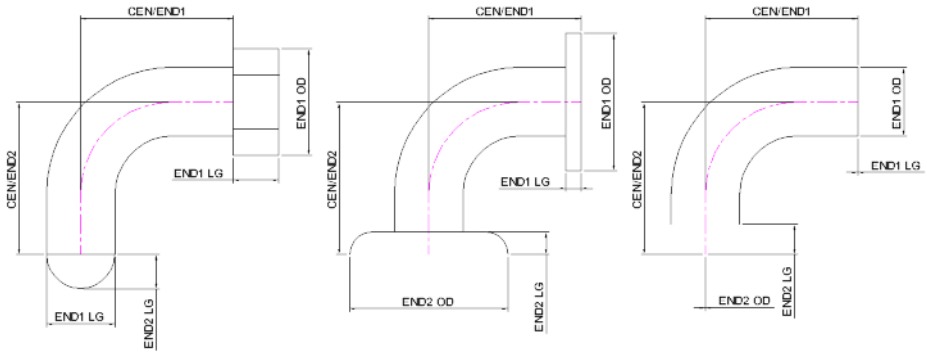
The user must use one of the default sanitary specifications that are supplied with CADWorx Plant for the icons to work on this toolbar.

Sanitary fittings are special because of the different ends that can be applied. In 2D double line and 3D solids, sanitary fittings can have 8 different end types – buttweld, flanged, ferrule, coupling, specific engagement type, hub, square, bull, and a simple engagement which can be used with threaded and socket weld piping. In 2D single-line and isometrics, these fittings can have up to 17 different end types. These end types can be individual to each end of a component. A tee could have a flange on the branch (type 2), a ferrule on one end (type 3), and an engagement on the other end (type 0). They can be mixed to form any variety of components.

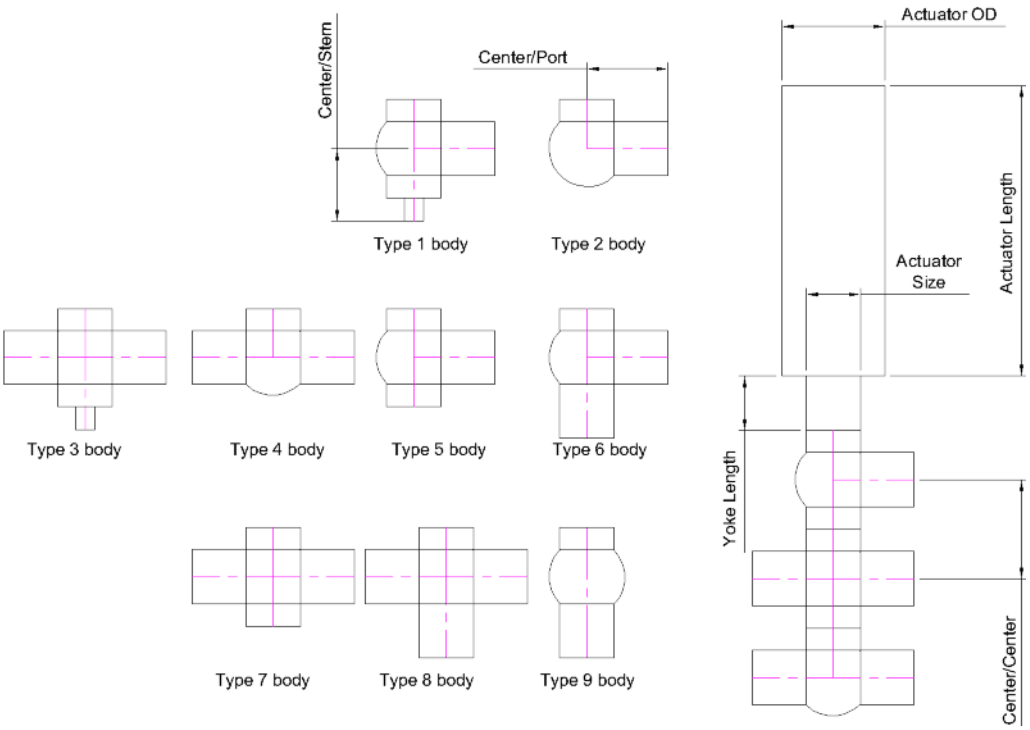
Examine drawings END1.DWG through END17.DWG in the <INSTALL-DIR>\SUPPORT directory (as shown below). These 17 different ends (only for single-line and isometric) represent one of the 8 ends mentioned for the 2D double line and 3D solids. These end types are controlled in the data files associated with the components as listed in the specifications. The prompts for these components will change according to the end types located in the data files.



Shown below are different examples of end types and their data file designations.



The Multi-port valves have special port controls in the data files. Listed below are all the port types available for these valves. When all three ports are specified in the data file, all three ports will be drawn on the drawing. When only one or two ports are required, place 0 in the last port entries and this supplies either a one or two port valve.



The CADWorx end types are exported to ISOGEN based on the table below:

CADWorx End Type	ISOGEN End Type
0	PL (Plain)
1	BW (Butt weld)
2	FL (Flanged)

3	LN (Liner/Nut)
4	LN (Liner/Nut)
5	MP (Male Part)
6	MP (Male Part)
7	MP (Male Part)
8	MP (Male Part)
9	CL (Clamped)
10	CL (Clamped)
11	CL (Clamped)
12	CL (Clamped)
13	CL (Clamped)
14	SW (Socket Weld)
15	SC (Screwed)

Pipe

Command line: PIPW

See Appendix D type 1 (see page 635) prompts for this component. Special options available in this command include *TOP* and *BOP* which start the pipe at the top of pipe and bottom of pipe respectively.

90 Degree Elbow w/ Special Ends

Command line: 90EN **(also** H90EN**)**

See Appendix D type 2 (see page 636) prompts for this component.

90 Degree Reducing Elbow w/ Special Ends

Command line: 90RE **(also** H90RE**)**

See Appendix D type 2 (see page 636) prompts for this component.

45 Degree Elbow w/ Special Ends

Command line: 45EN **(also** H45EN **or** V45EN**)**

See Appendix D type 3 (see page 637) prompts for this component.

Straight Tee w/ Special Ends

Command line: TESE **(also** HTESE **or** VTESE**)**

See Appendix D type 5 (see page 638) prompts for this component.

Reducing Tee w/ Special Ends

Command line: TERE **(also** HTERE **or** VTERE**)**

See Appendix D type 5 (see page 638) prompts for this component.

Straight Cross w/ Special Ends

Command line: CRSE **(also** HCRSE**)**

See Appendix D type 6 (see page 639) prompts for this component.

Reducing Cross w/ Special Ends

Command line: CRRE **(also** HCRRE **or** VCRRE**)**

See Appendix D type 6 (see page 639) prompts for this component.

Straight Lateral w/ Special Ends

Command line: LATE **(also** HLATE **or** VLATE**)**

See Appendix D type 7 (see page 640) prompts for this component.

Reducing Lateral w/ Special Ends

Command line: LARE **(also** HLARE **or** VLARE**)**

See Appendix D type 7 (see page 640) prompts for this component.

22.5 Degree Elbow w/ Special Ends

Command line: 22EN **(also** H22EN **or** V22EN**)**

See Appendix D type 3 (see page 637) prompts for this component.

11.25 Degree Elbow w/ Special Ends

Command line: 11EN **(also** H11EN **or** V11EN**)**

See Appendix D type 3 (see page 637) prompts for this component.

U-Bend w/ Special Ends

Command line: UBEN **(also** HUBEN **or** VUBEN**)**

See Appendix D type 4 (see page 638) prompts for this component. This component has a branch connection that will be placed tangential to the bottom of the return projected either from the side or downward. This branch location is controlled by the PROJ column in the component's data file. 0 projects the branch downward and 1 projects the branch out the side.

Reducer w/ Special Ends

Command line: CONCE **(also** ECCE**)**

See Appendix D type 1 (see page 635) prompts for this component. This routine will reverse the main and reduction sizes automatically after execution.

Stub w/ Special Ends

Command line: STUE

See Appendix D type 1 (see page 635) prompts for this component.

Wye w/ Special Ends

Command line: YE **(also** HYE **or** VYE**)**

See Appendix D type 8 (see page 640) prompts for this component.

Cap w/ Special Ends

Command line: CAPE

See Appendix D type 1 (see page 635) prompts for this component.

Nut/Coupling w/ Special Ends

Command line: NUTE

See Appendix D type 1 (see page 635) prompts for this component.

This component will act several different ways. In the data file, there are two entries for end types. It will only accept flange, ferrule and coupling end types. If the first entry is of flanged type, then the second one will be ignored. That means that it will only draw one flange at a time. If the first entry is of ferrule type, then the second entry will be ignored. That means that it will only draw one ferrule in the direction chosen. If the first entry is of coupling type, then the second entry will be used. It will expect a coupling type, but can draw any of the other types (they might get scaled improperly).

Gate Valve w/ Special Ends

Command line: GATE

See Appendix D type 1 (see page 635) prompts for this component.

Ball Valve w/ Special Ends

Command line: BALE

See Appendix D type 1 (see page 635) prompts for this component.

Globe Valve w/ Special Ends

Command line: GLBE

See Appendix D type 1 (see page 635) prompts for this component.

Plug Valve w/ Special Ends

Command line: PLUE

See Appendix D type 1 (see page 635) prompts for this component.

Check Valve w/ Special Ends

Command line: CHKE

See Appendix D type 1 (see page 635) prompts for this component.

Butterfly Valve w/ Special Ends

Command line: BFYE

See Appendix D type 1 (see page 635) prompts for this component.

3-Way Valve w/ Special Ends

Command line: 3VE **(also** H3VE **or** V3VE**)**

See Appendix D type 5 (see page 638) prompts for this component.

Angle Valve w/ Special Ends

Command line: ANGE **(also** HANGE**)**

See Appendix D type 2 (see page 636) prompts for this component.

Air Valve - One Port w/ Special Ends

Command line: AIR1

Command: **AIR1**
2" MULTIPORT VALVE
Pick start point <last point>: *pick a point at the first port center*
Pick actuator direction or [**Up**]:

Pick actuator direction

This option allows the user to place the multi port air valve horizontal. After picking a direction, the user must answer the following prompt with numeric input.

Up

This places the air valve in a vertical position. After placing a valve in the vertical direction, the user can pick or use numeric input for the rotations below.

Rotation 1 for body - TYPE 5: *pick a point or enter numeric valve*
Rotation 2 for body - TYPE 5: *pick a point or enter numeric valve*
Rotation 3 for body - TYPE 1: *pick a point or enter numeric valve*

Note: The data file that is specified by the specification controls the number of ports on the multi port air valve. If the single port air valve is directed to a data file that has all three ports assigned, then it will be drawn using all three ports. If the data file has only one port assigned, then it will draw only a single port. To remove ports, just place 0 in the data file.

Air Valve - Two Port w/ Special Ends

Command line: AIR2

See Air Valve (see page 101).

Air Valve - Three Port w/ Special Ends

Command line: AIR3

See Air Valve (see page 101).

Victaulic

Plant menu: ▶ **Toolbars** ▶ **Victaulic**



Note: The arrow in the lower right corner of the button indicates a flyout for more options.

The user must use one of the sanitary default specifications that are supplied with CADWorx Plant for the icons to work on the toolbar.

This toolbar provides components manufactured by Victaulic Corporation. The Victaulic specifications and data files have been provided by Victaulic and can be used to draw components supplied by this company. The specifications supplied are for AWWA, IPS, 304, 316, and PFCS type components.

These components are only a collection of other available components within CADWorx Plant. They have been collected in this toolbar for convenience.

Pipe

Command line: PIPW

See Appendix D type 1 (see page 635) prompts for this component. Special options available in this command include *TOP* and *BOP* which start the pipe at the top of pipe and bottom of pipe respectively.

90 Degree Long Radius Elbow

Command line: 90LR **(also** H90LR**)**

See Appendix D type 2 (see page 636) prompts for this component.

90 Degree Short Radius Elbow

Command line: 90SR **(also** H90SR**)**

See Appendix D type 2 (see page 636) prompts for this component.

45 Degree Elbow

Command line: 45LR **(also** H45LR **or** V45LR**)**

See Appendix D type 3 (see page 637) prompts for this component. Special options available in this command include *Opposite* direction that allows the user to continue from an existing 45 degree run. If coming off a 45 degree rotated pipe, use this option and the Osnap, NEArest of, and select back along the centerline of pipe. If coming off another 45 elbow use this option and the Osnap, INT of, and select the corner of the existing 45 elbow.

90 Degree Reducing Elbow

Command line: 90RD **(also** H90RD**)**

See Appendix D type 2 (see page 636) prompts for this component. Other options included with this function are the *Small* and *Large* end options.

11.25 Degree Elbow

Command line: 11LR **(also** H11LR **or** V11LR**)**

See Appendix D type (see page 636) 3 prompts for this component. Special options available in this command include *Opposite* direction that allows the user to continue from an existing 11.25 degree run. If coming off a 11.25 degree rotated pipe, use this option and the Osnap, NEArest of, and select back along the centerline of pipe. If coming off another 11.25 elbow use this option and the Osnap, INT of, and select the corner of the existing 11.25 elbow.

22.5 Degree Elbow

Command line: 22LR **(also** H22LR **or** V22LR**)**

See Appendix D type 3 (see page 636) prompts for this component. Special options available in this command include *Opposite* direction that allows the user to continue from an existing 22.5 degree run. If coming off a 22.5 degree rotated pipe, use this option and the Osnap, NEArest of, and select back along the centerline of pipe. If coming off another 22.5 elbow use this option and the Osnap, INT of, and select the corner of the existing 22.5 elbow.

Straight Tee

Command line: TESW **(also** HTESW **or** VTESW**)**

See Appendix D type 5 (see page 638) prompts for this component.

Reducing Tee

Command line: TERW **(also** HTERW **or** VTERW**)**

See Appendix D type 5 (see page 638) prompts for this component.

4-Way Straight Cross

Command line: CRSW **(also** HCRSW**)**

See Appendix D type 6 (see page 639) prompts for this component.

4-Way Reducing Cross

Command line: CRRW **(also** HCRRW **or** VCRRW**)**

See Appendix D type 6 (see page 639) prompts for this component.

Wye

Command line: YB **(also** HYB **or** VYB**)**

See Appendix D type 8 (see page 640) prompts for this component.

Lateral

Command line: LATW **(also** HLATW **or** VLATW**)**

See Appendix D type 7 (see page 640) prompts for this component.

Flanged Straight Tee

Command line: TEFL **(also** HTEFL **or** VTEFL**)**

See Appendix D type 5 (see page 638) prompts for this component.

Concentric and Eccentric Reducer

Command line: CONC **(also** ECC**)**

See Appendix D type 1 (see page 635) prompts for this component. This routine will reverse the main and reduction sizes automatically after execution.

Cap

Command line: CAPW

See Appendix D type 1 (see page 635) prompts for this component.

Coupling

Command line: CPS **(also** VCPS**)**

See Appendix D type 2 (see page 636) prompts for this component.

Slip-On Flange

Command line: FLGO

See Appendix D type 1 (see page 635) prompts for this component.

Butterfly Valve

Command line: BFYF

See Appendix D type 1 (see page 635) prompts for this component. Special options included with this component include an alternate *Center* starting location.

Check Valve

Command line: CHKF

See Appendix D type 1 (see page 635) prompts for this component. Special options available in this command include *Reverse* direction that allows the user to select the direction of the valve that will position the flow indicator pointing in the opposite direction.

Ball Valve

Command line: BALF

See Appendix D type 1 (see page 635) prompts for this component. Special options included with this component include an alternate *Center* starting location.

Plug Valve

Command line: PLUF

See Appendix D type 1 (see page 635) prompts for this component. Special options included with this component include an alternate *Center* starting location.

Threaded/Socket

Plant menu: ➤ Toolbars ➤ Thrd/SW Fittings



Note: The arrow in the lower right corner of the button indicates a flyout for more options.

In this section, socket weld components are shown on the command line. Most of these commands end with an S (for ex. 90S). There will be an equalivent threaded component (90T). Utilize the socket weld and threaded (see page 176) icon on the Settings Toolbar to set the desired mode.

Threaded/Socket Pipe

Pipe

Command line: PIPF

Command: **PIPF**
1" PIPE, S/XS SMLS, ASTM A-106 GR B
Pick start point or [**Ends/TOP/BOP**] <last point>:

Pick start point

This option allows the user to just pick a start and ending point with the default end types. If the drawing environment is set to threaded, the piping component will have threaded ends. If the drawing is set to a socket weld environment, the piping component will have plain ends.

Ends

This option allows the user to apply any end type to the pipe to be drawn. Bevel end types will show up even with the center line while Plain end type will be set back and is intended to be used with socket weld fittings. The end types will be truncated to the end of the description if the PipeDescriptionCat (see page 61) variable is on (PBE, TOE, TBE, TOE-BOE, etc.).

Starting end option [**Threaded/Bevel/Plain**] <Threaded>:
Ending end option [**Threaded/Bevel/Plain**] <Threaded>:

Threaded

This option is to be used when working with threaded pipe. The threaded default option will appear when the environment is set to threaded mode (see page 176).

Bevel

This option is to be used when working with welded components.

Plain

This option is to be used when working with threaded pipe. The threaded default option will appear when the environment is set to socket weld mode (see page 176).

After the previous prompts, the user would then have other options similar to the base options such as TOP and BOP or a start and ending point could be picked. The user can also enter to pick the last point of the last placed component.

Pick start point or [TOP/BOP] <last point>:
Pick end point:

TOP

This option allows the user to start the pipe at the top of pipe.

Pick start point:
Pick end point:

BOP

This option allows the user to start the pipe at the bottom of the pipe.

Pick start point:
Pick end point:

Nipple

Command line: NIP

See Appendix D type 1 (see page 635) prompts for this component. Please review the Pipe function topic for more information on the available end types.

Two other useful options located within this command include the *Length* and *Close* option. *Length* allows the user to enter an exact length. The *Close* option reads the close distance as specified within the data file.

Tube

Command line: TUBE

This function provides the user the capability of attaching tubing to any 2D or 3D polyline. The function allows different radii to be placed at each vertex of the polyline or the user can enter a common radius for all corners. Use the ROUTER (see page 293) command within CADWorx Plant for useful 3D polyline routing capabilities.

The tubing uses the pipe component entry within the CADWorx Plant specifications. Pipe and tubing can be used concurrently without any problem. Tubing could occupy 1/8" – 1/2" range where regular pipe could occupy the 3/4" and above. Tubing could be set up as an optional component - see Specification (see page 254) editing for more information. The specification could also be set up to only handle tubing.

Once placed in the drawing, tubing can have other components inserted into it. It will automatically break just as normal piping. Even after a component is inserted, the tubing will remain as one piece. When edited with CEDIT, the tubing attached to a single polyline will act as one piece. Grouping control by the AutoCAD environmental variable PICKSTYLE, must be turn on for the tubing to act as one piece. CADWorx GROUPTOG (see page 368) command can also be used to turn AutoCAD grouping control on.

The following prompts indicate how the tubing functionality works.

Command: TUBE
Enter corner radius or [**Manual**]:

Manual

This option allows individual radii to be placed at each corner of the selected polyline.

Select polyline or 3dpoly: *select 2D or 3D polyline*

When used, the function will stop and pause at each vertex asking for a radius. It will show a rubber-banded line back to the actual vertex itself.

Enter corner radius

The prompts will quit after the last vertex has been selected.

Enter corner radius: *enter a radius (i.e. 3")*

This prompt asks for the radius one time. This chosen radius will be placed at each vertex.

Select polyline or 3dpoly: *select 2D or 3D polyline*

Note: Tubing is exported to a database only when using a Live Database.

Note: The tubing will appear in a CADWorx BOM (see page 386) as one piece.

Note: Tubing is exported to the CADWorx ISOGEN (see page 214) isometric generator and will appear in the ISOGEN material list.

Note: Tubing is not exported to CADWorx AutoISO (see page 185) isometric generator.

Note: Tubing is not exported to CAESAR II (see page 193).

Threaded/Socket 90 Degree Elbow

Command line: 90S **(also** H90S**)**

See Appendix D type 2 (see page 636) prompts for this component.

Threaded/Socket 90 Degree Street Elbow

Command line: 90SS **(also** H90SS**)**

See Appendix D type 2 (see page 636) prompts for this component. Other options included with this function are the *Male* and *Female* ends options.

Threaded/Socket 45 Degree Elbow

Command line: 45S **(also** H45S **or** V45S**)**

See Appendix D type 3 (see page 637) prompts for this component. Special options available in this command include *Opposite* direction which allows the user to continue from an existing 45 degree run. If coming off a 45 degree rotated pipe use this option and the Osnap, NEArest of, and select back along the centerline of pipe. If coming off another 45 elbow use this option and the Osnap, INT of, and select the corner of the existing 45 elbow.

Threaded/Socket 90 Degree Reducing Elbow

Command line: 90RS **(also** H90RS**)**

See Appendix D type 2 (see page 636) prompts for this component. Other options included with this function are the *Small* and *Large* end options.

Threaded/Socket Straight Tee

Command line: TESS **(also** HTESS **or** VTESS**)**

See Appendix D type 5 (see page 638) prompts for this component.

Threaded/Socket Reducing Tee

Command line: TERS **(also** HTERS **or** VTERS**)**

See Appendix D type 5 (see page 638) prompts for this component.

Swage

Command line: SWGC **(also** SWGE**)**

Command: **SWGC**
2"x1" SWAGE, ECC. S/80, ASTM A-234 GR WPB
Pick large end point or [**Small end**] <last point>:

Pick large end point

This is the default start point. The user can pick a point or enter to automatically connect to the last placed component.

Pick small end direction: *pick a direction*

The next prompt allows the user to add any desired end type connection. This is a alpha description and anything can be entered here.

Large end type <PLE>: *type end connection*
Small end type <PSE>: *type end connection*

If SWGE is used, then an additional prompt would require the location of the flat side of the eccentric swage.

Pick direction of flat side or [**Up/Down**]:

Down

Select if elbow is desired in the downward or the negative Z direction.

Up

Select if elbow is desired in the upward or positive Z direction.

Small end

This allows the user to start the swage at the small end.

Pick small end point:
Pick large end direction:

See explanation above for the following prompts.

Large end type <PLE>:
Small end type <PSE>:

Note: This routine will reverse the main and reduction sizes automatically after execution.

Threaded/Socket Reducers

Command line: SWGCS **(also** SWGES**)**

See Appendix D type 1 (see page 635) prompts for this component.

Threaded/Socket Cap

Command line: CAPS

See Appendix D type 1 (see page 635) prompts for this component.

Threaded/Socket Straight Cross

Command line: CRSS **(also** HCRSS**)**

See Appendix D type 6 (see page 639) prompts for this component.

Threaded/Socket Lateral

Command line: LATS **(also** HLATS **or** VLATS**)**

See Appendix D type 7 (see page 640) prompts for this component.

Threaded/Socket Misc.

Plug

Command line: PLGT

See Appendix D type 1 (see page 635) prompts for this component.

Bushing

Command line: BUST (**also** BUSS)

See Appendix D type 1 (see page 635) prompts for this component.

Threaded/Socket Straight Coupling

Full Coupling

Command line: CPS **(also** VCPS**)**

See Appendix D type 1 (see page 635) prompts for this component.

Half

Command line: HCPS **(also** VHCPS**)**

See Appendix D type 1 (see page 635) prompts for this component.

Threaded/Socket Reducing Coupling

Command line: CPRS **(also** VCPRS**)**

See Appendix D type 1 (see page 635) prompts for this component.

O-lets

Welded

Command line: WOL **(also** VWOL**)**

See Appendix D type 1 (see page 635) prompts for this component.

Threaded

Command line: TOL **(also** VTOL**)**

See Appendix D type 1 (see page 635) prompts for this component.

Socket

Command line: SOL **(also** VSOL**)**

See Appendix D type 1 (see page 635) prompts for this component.

Ell-O-Lets

Welded

Command line: EOLW **(also** HEOLW **or** VEOLW**)**

See Appendix D type 2 (see page 636) prompts for this component.

Threaded

Command line: EOLT **(also** HEOLT **or** VEOLT**)**

See Appendix D type 2 (see page 636) prompts for this component.

Socket

Command line: EOLS **(also** HEOLS **or** VEOLS**)**

See Appendix D type 2 (see page 636) prompts for this component.

Lateral-O-Lets

Welded

Command line: LOLW **(also** HLOLW**)**

See Appendix D type 2 (see page 636) prompts for this component.

Threaded

Command line: LOLT **(also** HLOLT**)**

See Appendix D type 2 (see page 636) prompts for this component.

Socket

Command line: LOLS **(also** HLOLS**)**

See Appendix D type 2 (see page 636) prompts for this component.

Nip-O-Lets

Welded

Command line: NOLP **(also** VNOLP**)**

See Appendix D type 1 (see page 635) prompts for this component.

Threaded

Command line: NOLT **(also** VNOLT**)**

See Appendix D type 1 (see page 635) prompts for this component.

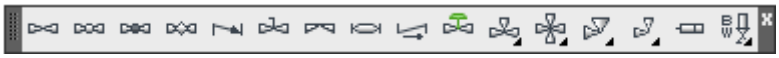
Threaded/Socket Union

Command line: UNS

See Appendix D type 1 (see page 635) prompts for this component.

Threaded/Socket Valves

Plant menu: ➤ **Toolbars** ➤**Thrd/SW Valves**



Note: The arrow in the lower right corner of the button indicates a flyout for more options.

In this section, socket weld components are shown on the command line. Most of these commands end with an S (for ex. GATS). There will be an equalivent threaded component (MATT). Utilize the socket weld and threaded (see page 176) icon on the Settings Toolbar to set the desired mode.

Threaded/Socket Gate Valve

Command line: GATS

See Appendix D type 1 (see page 635) prompts for this component.

Threaded/Socket Ball Valve

Command line: BALS

See Appendix D type 1 (see page 635) prompts for this component.

Threaded/Socket Globe Valve

Command line: GLBS

See Appendix D type 1 (see page 635) prompts for this component.

Threaded/Socket Plug Valve

Command line: PLUS

See Appendix D type 1 (see page 635) prompts for this component.

Threaded/Socket Check Valve

Command line: CHKS

See Appendix D type 1 (see page 635) prompts for this component. Special options available in this command include *Reverse* direction that allows the user to select the direction of the valve which will position the flow indicator pointing in the opposite direction.

Threaded/Socket Needle Valve

Command line: NEDS

See Appendix D type 1 (see page 635) prompts for this component.

Threaded/Socket Diaphragm Valve

Command line: DIAS

See Appendix D type 1 (see page 635) prompts for this component.

Threaded/Socket Rotary Valve

Command line: ROTS

See Appendix D type 1 (see page 635) prompts for this component.

Threaded/Socket Dual Check Valve

Command line: DCHS

See Appendix D type 1 (see page 635) prompts for this component.

Threaded/Socket Control Valve

Command line: CNTS

See Appendix D type 1 (see page 635) prompts for this component. Other available options include the *Length* option that will allow an exact length to be entered.

Threaded/Socket 3-Way Valve

Command line: 3VS **(also** H3VS **or** V3VS**)**

See Appendix D type 5 (see page 638) prompts for this component.

Threaded/Socket 4-Way Valve

Command line: 4VS **(also** H4VS**)**

See Appendix D type 6 (see page 639) prompts for this component.

Threaded/Socket Relief Valve

Command line: RELS **(also** HRELS **or** VRELS**)**

See Appendix D type 5 (see page 638) prompts for this component. Please refer to the flanged relief valve (see page 86) for more information on this component.

Threaded/Socket Angle Valve

Command line: ANGS **(also** HANGS**)**

See Appendix D type 2 (see page 636) prompts for this component.

Threaded/Socket Gage Valve

Command line: GAGS

See Appendix D type 1 (see page 635) prompts for this component. The outlet locations of the gage valve can be picked by using OSnap, MIDpoint of, and picking the sides on the outlet end. In 3D there is no way to locate the outlets perfectly. We suggest drawing the component in 2D and then converting it to 3D solids to accurately locate the outlets.

Threaded/Socket X-Body Valve

Command line: EGAB **(also** EGAT **or** EGAS**)**

See Appendix D type 1 (see page 635) prompts for this component.

Gaskets etc.

Plant menu: ▶ **Toolbars** ▶ **Gaskets, etc.**



Note: The arrow in the lower right corner of the button indicates a flyout for more options.

Tick Mark

Command line: TICK

This routine is used to symbolically show which side of the extension line (of a dimension) the gasket is on. This routine is dependent upon the dimscale being set properly (select Setup from the pulldown menu and set the proper scale). For example; if the gasket thickness was set to 1/8" and the scale of the drawing was set to full; the distance from the extension line to the tick mark would be 1/8". If the scale of the drawing was set to 3/4"=1'-0", the distance from the tick mark to the extension line would be 3/8". This will insure that the tick mark will be visible on a plotted drawing.

The initial prompt allows the user to set the desired thickness, but will not appear again during the single drawing session. Changing the thickness is allowed in the next prompt.

Command: **TICK**
Enter gasket thickness <1/8">: *enter real number or enter*
Pick dimension extension line [**Change**]:

Change

Select this option if the user needs to change the thickness. After changing the thickness, this routine will proceed to insert a tick mark.

Enter new thickness: *enter real number*
Pick dimension endpoint: *pick point*
Pick direction: *pick direction parallel with extension line*
Pick side: *pick gasket side*

Pick dimension extension line

Pick direction: *pick direction parallel with extension line*
Pick side: *pick gasket side*

Stud Bolt

Standard

Command line: PLACEBOLT

This routine allows the insertion of stud bolt sets into the drawing environment for bill of materials extraction. The standard option allows the user to insert a stud bolt set anywhere. This option requires that the size and specification are set accordingly.

Command: **PLACEBOLT**
3/4" X 3 1/2" STUD BOLTS W/ NUTS
Hit enter when finished...

Select placement point for bolts: *pick point*
Select text tag direction: *pick direction*

Select placement point for bolts: *pick another point*
Select text tag direction: *pick another direction*

Select placement point for bolts: *enter to end*

Note: The "B" that is inserted for the bolt symbol uses the current AutoCAD Text Style setting.

Non Standard

Command line: NONSTDBOLT

This non-standard allows the user to insert a stud bolt set anywhere. The prompts below will allow for the insertion of special bolts. This option requires that the size and specification are set accordingly.

Command: **NONSTDBOLT**
Enter abbreviated description: *(ex: NON STD STUD BOLTS)*
Enter description: *(ex: STUD BOLTS W/ DOUBLE NUTS & WASHERS)*
Enter diameter of stud: *enter real number (ex: 5/8")*
Enter length of stud: *enter real number (ex: 7.25 or 7-1/4")*
Enter number of stud in set: *enter integer number (ex: 12)*
Enter weight for set: *enter real number (ex: 23)*
Enter code <0041>: *enter string, real number or enter*
Enter sort location number <41>: *enter integer number or enter*

3/4" X 7 1/4" STUD BOLTS W/ DOUBLE NUTS & WASHERS
Hit enter when finished...

Select point of placement for bolts: *pick point*
Select tag direction: *pick direction*

Select point of placement for bolts: *pick another point*
Select tag direction: *pick another direction*

Select point of placement for bolts: *enter to end*

Automatic

Command line: AUTOBOLT

This will assign stud bolt sets with similar sizes and specifications to all gaskets in the crossing or window. This option will insert a stud bolt set at every gasket located within the chosen window or crossing. Gaskets of different sizes and specifications will be automatically handled.

Command: **AUTOBOLT**
Pick piping components (or gaskets) to assign bolts:
Select objects: **C** *by crossing or window*
First corner: *pick point* Other corner: *pick another point*
57 found *(not necessarily the number of gaskets)*

Select objects: *select more or hit enter to end*

Gasket

Command line: GAS

The gasket is composed of two different entities. There will be an inserted block that is located at the insert point and another inserted block that will be at the other side of the gasket. The “GASR” will contain xdata information for bill of materials extraction. The “GASL” will be a plain insert without information. When “Mode Converting” is used to convert components to another mode in a Z plane position, the “GASL” insert will be destroyed. This is not important, since all the information is contained in the “GASR” insert. Converting the single “GASR” insert, in the proper X and Y planes, will provide both sides of the gasket again. The shape of the gasket allows other uses which are explained next.

The half circle on the outer portion of both inserts allow for easy picking; for dimensioning (ex: when dimensioning, select Osnap, CENter of, and pick the center of circle which is the exact corner of the face of flange). The user will find that it is relatively easy to select the center of the circle from a zoomed out view.

For placing a flange face against the gasket, the user can select Osnap, ENDpoint of, and select the desired side of the gasket to place the flange perfectly in position. This will provide a method of placing the flange when the last point option is no longer available.

This routine is also used for ring type joints. The data files for rings will have an extra column indicating the ring number. The gasket routine will sense this additional column and add this ring number to the bill of materials description. The distance between the faces of adjoining flanges (under compression) will be used in this file instead of the actual depth of the ring.

Command: **GAS**
4" GASKET, 150LB, FLEX TYP CG, SPIRAL WD
Pick start point <last point>: *pick point or enter for last*
Pick direction: *pick direction*

Weld Gap

Command line: WELD

This routine provides weld gaps for detailed pipe fabrication. These weld gaps are control by the specifications (also by means of multiple entries) and are placed according to size and/or schedule. Each specification (see page 254) will have data files which provide different thickness. The AutoWeldGap (see page 51) variable is available for setting the default startup settings.

Starting a new pipe or component from the weld gap can be done by selecting Osnap, INSert of, and then selecting any portion of the weld gap symbol. The last point is also available by simply hitting a return.

Command: **WELD**
Pick start point <last point>: *pick point or enter for last*
Pick direction: *pick direction*

There is also a function to extract all weld gaps for an accurate count for estimating purposes. This option can be executed at the command prompt.

Command: **WELDTOTAL**
Enter an option [**Entire drawing/Individual**]:

Entire drawing

This will produce a count of every weld gap on the entire drawing. There will not be any additional prompts.

Individual weld gaps

This option allows individual selection of weld gaps. Selection can be achieved by the AutoCAD window, crossing, etc.

Select objects: *select by crossing*
Select objects: *select more or enter to end*

After the preceding prompts the user will be given the total number of welds, the thickness and the total amount of linear weld required for each pipe size. Shown below is a typical screen output. This information can be copy from the AutoCAD text screen and pasted into a text file or an Excel spreadsheet. The ability to export the drawing to a database using the DBFGEN or LIVEDB commands can also help manage weld gaps.

```
Q      Sp      Nomi      Act      Thk.      Linear
t      ec      nal       ual                Lg.
y      .
.
```

5	A0	2.000	2.375	0.154	37.310
4	A0	3.000	3.500	0.216	43.980
14	A1	10.000	10.750	0.365	472.810
1	A1	14.000	14.000	0.375	43.980
150	A1	2.000	2.375	0.154	1119.190
51	A1	3.000	3.500	0.216	560.770
79	A1	4.000	4.500	0.237	1116.840
71	A1	6.000	6.625	0.280	1477.730
26	A1	8.000	8.625	0.322	704.500
13	A2	2.000	2.375	0.218	97.000
10	A2	3.000	3.500	0.216	109.960
12	A2	4.000	4.500	0.237	169.650
2	C1	1.500	1.900	0.200	11.940
79	C1	10.000	10.750	0.365	2668.000
1	C1	16.000	16.000	0.500	50.270
126	C1	2.000	2.375	0.154	940.120
68	C1	6.000	6.625	0.280	1415.290
12	C1	8.000	8.625	0.322	3034.780
3	C2	2.000	2.375	0.218	22.380
19	C2	3.000	3.500	0.300	208.920
13	C2	6.000	6.625	0.432	270.570
6	F1	10.000	10.750	0.593	202.630
1	F1	16.000	16.000	0.843	50.270
39	F1	2.000	2.375	0.218	290.990
5	F1	3.000	3.500	0.300	54.980
1	F1	4.000	4.500	0.337	14.140

```
1       F1      8.00      8.6       0.50      27.100
                  0         25        0
2       F2      3.00      3.5       0.30      21.990
                  0         00        0
```

The following information only applies to ISOGEN and weld gap components: If "Misc" BOM Item Type (see page 303) is set in CEDIT (see page 303) for weld gaps, ISOGEN will represent this weld gap as a "Field Fit Weld". Use ISOGEN Option Switch 22 to increase cut piece add on allowances for field fit welds.

Pipe Ends

Plant menu: ▶ **Toolbars** ▶ **Pipe Ends**



Flange O.D.

Command line: ENDFLG

This feature draws a circle around the diameter of a flange at the current size and specification. The circle (representing the end view of a flange) is placed at the center point wherever picked. Hit enter or space bar to end. The circle will have centerlines that will be on the default “CL” layer.

Command: **ENDFLG**
Hit enter to end...

Pick point for 4"-150 flange O.D.: *pick point*
Pick point for 4"-150 flange O.D.: *pick another point*
Pick point for 4"-150 flange O.D.: *enter*

Plain End

Command line: ENDPLAIN

This feature draws a circle around the diameter of a pipe at the current size and specification. The circle (representing the end view of a pipe) is placed at the center point wherever picked. Hit enter or space bar to end.

Command: **ENDPLAIN**
Hit enter to end...

Pick point for 4"-150 plain end: *pick point*
Pick point for 4"-150 plain end: *pick another point*
Pick point for 4"-150 plain end: *enter*

Hidden End

Command line: ENDHIDDEN

Same as PLAIN END except with hidden linetype

Command: **ENDHIDDEN**
Hit enter to end...

Pick point for 4"-150 hidden end: *pick point*
Pick point for 4"-150 hidden end: *pick another point*
Pick point for 4"-150 hidden end: *enter*

Pipe End

Command line: ENDPIPE

This feature places a baseball at the center point wherever picked. Hit enter or space bar to end. The size and the layer of the baseball are determined by the current size and specification.

Command: **ENDPIPE**
Hit enter to end...

Pick point for 4"-150 baseball: *pick point*
Pick point for 4"-150 baseball: *pick another point*
Pick point for 4"-150 baseball: *enter*

Angle

Command line: SETANGLE

This command sets the rolled angle for rolled flanges and pipe ends.

Command: **SETANGLE**
Enter placement angle for accessories: *enter angle (ex. 45)*

Rolled Flange

Command line: ROLLEDFLG

This feature draws an ellipse the diameter of a flange at the current size and specification. The ellipse (representing the end view of a rolled flange) is placed at the center point wherever picked. Hit enter or space bar to end.

Command: **ROLLEDFLG**
Hit enter to end...

Pick point for 4"-150 rotated flange O.D.: *pick point*
Pick rolled direction: *pick direction*
Pick point for 4"-150 rotated flange O.D.: *pick another point*
Pick rolled direction: *pick direction*
Pick point for 4"-150 rotated flange O.D.: *enter*

If rolled angle (SETANGLE) is not set, the user will get the following message.

Please enter placement angle for accessories...

Rolled Plain

Command line: ROLLEDPLAIN

This feature draws an ellipse the diameter of a pipe at the current size and specification. The ellipse (representing the end view of a rolled pipe end) is placed at the center point wherever picked. Hit enter or space bar to end.

Command: **ROLLEDPLAIN**
Hit enter to end...

Pick point for 4"-150 rotated plain end: *pick point*
Pick rolled direction: *pick direction*
Pick point for 4"-150 rotated plain end: *pick another point*
Pick rolled direction: *pick direction*
Pick point for 4"-150 rotated plain end: *enter*

Rolled Hidden

Command line: ROLLEDHIDDEN

Same as ROLL PLAIN except hidden linetype

Command: **ROLLEDHIDDEN**
Hit enter to end...
Pick point for 4"-150 rotated hidden end: *pick point*
Pick rolled direction: *pick direction*
Pick point for 4"-150 rotated hidden end: *pick point*
Pick rolled direction: *pick direction*
Pick point for 4"-150 rotated hidden end: *enter*

Restraints

Plant menu: ➤ Toolbars ➤ Restraints



There are six different types of restraints used with CADWorx Plant. These restraints are drawn in a symbolic form and can be changed by the user. The drawing files for these restraints are H1-H6.DWG and are located in the <INSTALL-DIR>\SUPPORT directory.

These drawing are drawn at 1 unit which allows them to be drawn any size (say 17”) with the functions prompt. These drawing could be redrawn at a different length (say 17”) and the length prompt could be answered with 1 each time. Each results with a 17” long restraint. As previously mentioned, the drawings are in a symbolic form. They can be redrawn in 3 dimensional if desired. Make sure to maintain the base (0,0,0) and the direction of the restraint (0 degrees).

Regardless of the shape or the length, the restraint is used in the bi-directional link with CAESAR II (pipe stress analysis software). The position of the restraint will be important for the evaluation of the stress model within CAESAR II. For example, a Rotational restraint can be positioned at any vector.

Note: The CAESAR Tolerance (see page 55) variable is useful when the bi-directional has problems locating these restraints along a pipe line (especially sloped lines).

These restraints are also use in the CADWorx Plant AutoISO. Wherever located in the model is where they will be placed in the isometric. The automatic dimensioning will also locate them appropriately.

All supports drawn using the Restraints toolbar are exported as guides to ISOGEN. To have a different symbols exported to ISOGEN, change the SKEY in the CEDIT -> ISOGEN Data dialog. A support direction can also be set from the ISOGEN Data dialog. The possible values for the SKEY are: ANCH, DUCK, GUID, SKID, SPRG, HANG. See ..\CADWorxPlant\ISOGEN\Isogen_Utils\Skeys.pdf for sample ISO shapes used with these SKEYs.

Translational

Command line: TRANSLATIONAL

Command: **TRANSLATIONAL**
Pick location on component for restraint: *pick a piping component*

If a valid piping component is not located by the restraint function, it will not have the necessary information (size, specification, etc.) for CAESAR II to make proper analysis or for the automatic isometric. If this happens, the next prompt will appear asking for a component selection. This would be the component that the restraint is attached to.

Select component or enter for none:

Pick a component centerline so that the program can extract the information needed to place into the restraint. If enter is hit, then it will not place any information within the restraint, therefore it will not transfer to CAESAR II.

Regardless of what is chosen above, the following prompts require the direction to be entered.

Enter or pick rotation [**Up/Down**]:

Enter or pick rotation

This option allows the user to enter a degree for the restraint to be rotated or the ability to pick two points.

Up

Chose this option if the restraint is needed in the positive Z direction.

Down

Chose this option if the restraint is needed in the negative Z direction.

These last prompts allow the entry of the depth and length of the restraint (basically the X and Y scaling of the block).

Enter depth: 18
Enter length or diameter <1'-6">:

If a component is found, then the user can enter a description if desired. This can be changed at anytime with the CEDIT command.

Enter assembly identification number <18.00-1 TRANSLATIONAL>:

Anchor

Command line: ANCHOR

Command: **ANCHOR**

Same as the Translational.

Spring

Command line: SPRING

Command: **SPRING**

Same as the Translational.

Rotational

Command line: ROTATIONAL

Command: **ROTATIONAL**

Same as the Translational.

Snubber

Command line: SNUBBER

Command: **SNUBBER**

Same as the Translational.

Guide

Command line: GUIDE

Command: **GUIDE**

Same as the Translational.

Misc

Plant menu: ▶ **Toolbars** ▶ **Misc**



Note: The arrow in the lower right corner of the button indicates a flyout for more options.

Strainer

Flanged

Command line: STRF **(also** HSTRF**)**

Command: **STRF**
4" STRAINER, 150LB FLG.
Pick start point or [**Length**] <last point>:

Length

This option provides different lengths for the strainer. Instead of reading information from the specified strainer data file, it prompts the user for the face/face and centerline to the bottom of the basket dimensions. The bottom of the basket location is the extreme lower part of the basket, not at the centerline.

Enter length: *enter real number (ex: 12, 10-1/4")*
Enter length of basket side: *enter real number (ex: 8, 8-3/8")*

Pick start point <last point>: *pick point*
Pick main end direction: *pick direction*
Pick branch end direction: *pick direction*

Pick start point

The user can hit enter here to connect to the last point on the previous placed component.

Pick start point <last point>: *pick point*
Pick main end direction: *pick direction*
Pick branch end direction: *pick direction*

Threaded/Socket

Command line: STRS **(also** HSTRS**)**

This command is similar to the STRF command above.

User Shapes

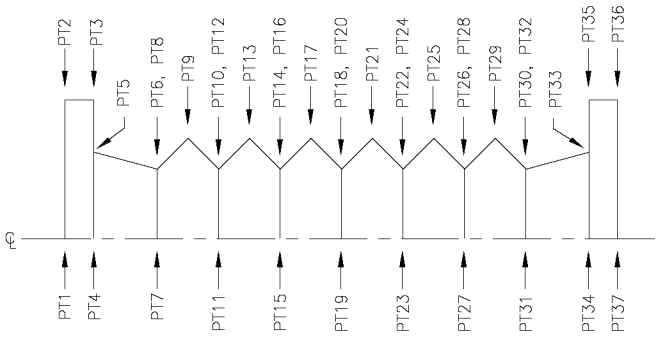
Command line: USERCREATE

Misc toolbar:

This routine allows the user to create any symmetrical or non-symmetrical user shape component. Symmetrical shapes require an outline to be drawn that can be used to create the special user component. Non-symmetrical shapes required a block or an object completely drawn in 2D and 3D that represents the complete component. A symmetrical object can be converted from 2D to 3D Mode, or vice versa, using the mode convert (see page 295) routines. It does not allow double-line to single-line conversion in the 2D mode. Non-symmetrical shapes can only convert between isometric and solids (CONVERTISO or CONVERTSOLID).

The user must first create the component and then modify the specification. Once that is complete, simply come back to the following toolbar and select the user shape program button. User shape components can also be tagged as optional components in the specification. This allows for an indefinite number of additional components.

For symmetrically shaped components, draw a 2D polyline indicating the shape's outline. The symmetrical shape is limited to straight lines between vertices in the polyline. In other words, there cannot be a fillet radius on any corner of the polyline. If they exist, the routine will draw them with chamfered corners. Draw only half of the shape on one side of the centerline. We recommend preparing an outline before trying to draw the polyline. For example, draw the component using standard AutoCAD methods or CADWorx Plant components and then draw a polyline around the contour.



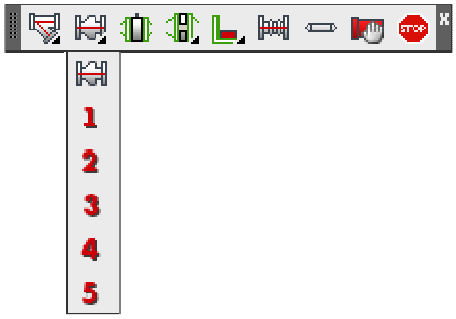
Note: For symmetrical shapes, the polyline may overlap as shown in the example below for 2D (drawn to each indicated point, PT1, PT2, etc.). For use with 3D solids, the polyline cannot overlap (there cannot be a PT4 or PT7). If the polyline overlaps, an ACIS error will occur indicating the object cannot be revolved.

For non-symmetrically shaped components, a 2D and 3D representation must be created first. It needs to show all the required details of the component. The 3D representation will be used when in the 3D drawing mode and it can be used for orthographic and model type representation. The 2D representation will be used when in the 2D isometric drawing mode. This mode was primarily designed for use with automatic isometrics.

After the polyline or components are complete, run the USERCREATE function (toolbar under pulldown Plant>Toolbars>Misc) to initiate the routine. This will create a data file in the required directory with the required file name. For non-symmetrically shaped componets, it will also create two drawing files – a 2D and 3D drawing. The drawing names will have append in square brackets, the size and drawing mode (i.e. [4.0-3D]). The file name must be unique. A sub-directory called \USE in the <INSTALL-DIR>\LIB_I, <INSTALL-DIR>\LIB_M, and <INSTALL-DIR>\LIB_MM is provided for this purpose. This function will also create a directory if it does not exist.

Make sure to have a default size and specification set. USERCREATE will have to be run multiple times for creating components with different sizes. From the toolbar under Plant>Toolbar>Misc in the pulldown, select *Create* icon to start the process.

Note: With the use of optional components, a specification file can have an unlimited number of user shapes.



The following information will be prompted.

Command: **USERCREATE**
[**Append existing file/Create new file**] <Create new file>:

Append existing file

This option is used to append to an existing file. The user will not be asked to “Enter short description” because the file and description already exist. For example, it can be used to create a 6" flexible coupling after the 4" in this example. This is true for symmetrical and non-symmetrical shapes.

Create new one

This option creates a new component and file. As mentioned above, non-symmetrical shapes will have three associated files while symmetrical shapes will have only one.

Enter name of library directory <USE>: *enter USE or hit enter*
Enter name of file, no extension <COMP>: *enter FLEX*

At this point, if the specification is not set an error will occur. This specification is used only as a default and does not have to be the same specification that the component is going to be placed with.

Specification or size is not set, please set and try again.

If a specification is set, it will ask for confirmation. Just accept it.

Enter specification <150>: *enter to accept the specification*

The next prompt will ask for the long description for the component. It is only a description for use within the data file. Further explanation on entering the component into the specification with a more complete description will be covered in the next section.

Enter long annotation: *enter FLEXIBLE COUPLING*

Next, the user will be prompted whether to create reducing components.

Make component reducing [Yes/No] <No>: *hit enter for no or type Y for yes*

When yes is entered, additional prompts appear later. For example, creating a 4”x3” flexible coupling is possible.

Note: Do not mix reducing and non-reducing components in the same file.

Enter weight: *enter real number (ex: 130)*
Enter density: *enter 0.283 for steel*

This next prompt allows points to be picked, selection of a polyline outline or selection of a pre-drawn block with assembled entities. The *Block* option can be constructed from individual entities or it can be an inserted block with attributes. Pick the *Select polyline* option for the example below.

Enter an option [**Pick points/Select polyline/Block**]:

Pick points

This option is used with symmetrical components. It asks for the selection of each point required to form the shape. First, reference points (starting and ending points) must be selected.

Pick reference point: *pick point pt1*
Pick end point: *pick point pt37*

Hit enter when finished...
Pick first point: *pick point pt1*
To point: *pick point pt2*
To point: *pick point pt3*
To point: ...
To point *pick point pt36*
To point: *pick point pt37*
To point: *enter to end*

Select polyline

This option is also used with symmetrical components. It is very similar to the prompt above except it asks for the selection of a polyline. It will automatically extract all the points from the selected polyline. First, reference points (starting and ending points) must be selected.

Pick reference point: *pick point pt1*
Pick end point: *pick point pt37*

Select polyline: *select polyline drawn through the points*

Note: For the *Pick points* and *Select polyline* option above, the points picked or the polyline selected must lay in an X-Y plane. This will be clearly visible when the reference points are pick above. If the X and Y UCS icon indicators are not showing, then the object needs to be drawn in an X-Y plane. The easiest way is to draw your polyline or pick your points in a world coordinate system.

Block

This option is used with non-symmetrical components. If the component is created on layer 0, it will inherit the specification color when inserted. This option requires a fully developed component drawn to the correct size with AutoCAD or CADWorx Plant. Even if the assembled components used have extended entity data attached, this function will remove it.

Enter an option [**2d/3d/Both**] <Both>:

2d

This option is similar to the *Both* option below except it creates the 2D portion of the component only.

3d

This option is similar to the *Both* option below except it creates the 3D portion of the component only.

Both

This option allows both 2D and 3D non-symmetrical components to be built. The components must be fully constructed, ready for use, and viewable on the screen before entering this function. The next prompt requires start and end reference points. These points do not necessarily need to be connection points, which will be discussed later.

Pick 2D reference point: *pick start point*
Pick end point: *pick ending point*

Connection points are where other equipment or components attach to the non-symmetrical user shape. These points are very important when exporting the model to an automatic isometric and will affect the scaling if not properly selected. Enter to finish the connection point process.

Pick connection point: *pick point*
Next connection point: *pick point*
Next connection point: *pick point*
Next connection point: *enter to finish*

Start with 2D representation...

Select objects: *select the 2D objects*
Select objects: *enter to end the selection process*

At this point, the reference point can be reselected. This is useful when drawing a 2D and 3D component. It allows the two components to be separated for easier selection.

Reselect reference point [Yes/No] <Yes>: *enter to reselect*

Pick new reference point: *pick start point*

3D representation...
Select objects: *select the 3D objects*
Select objects: *enter to end the selection process*

The next prompt asks whether the present size is set correctly. If not, change it to the required size at this prompt.

6" main size [Yes/No] <Yes>: *enter for yes, type N for this example*

By entering no, the size can be changed at this point. If a reducing component was requested above, an additional prompt regarding the reduction size would be requested here.

Nominal main size: *4*
Finished...

This symmetrical example created a file in <INSTALL-DIR>\LIB_I\USE\FLEX.USE. The contents of the file will be similar to the following, where pt1 would be the x,y locations of the first point and pt37 would be the x,y location of the last point.

```
FLEXIBLE COUPLING
=================================================
(4.0 4.5 130.0 (pt1)(pt2)(pt3).....(pt35)(pt36)(pt37))
```

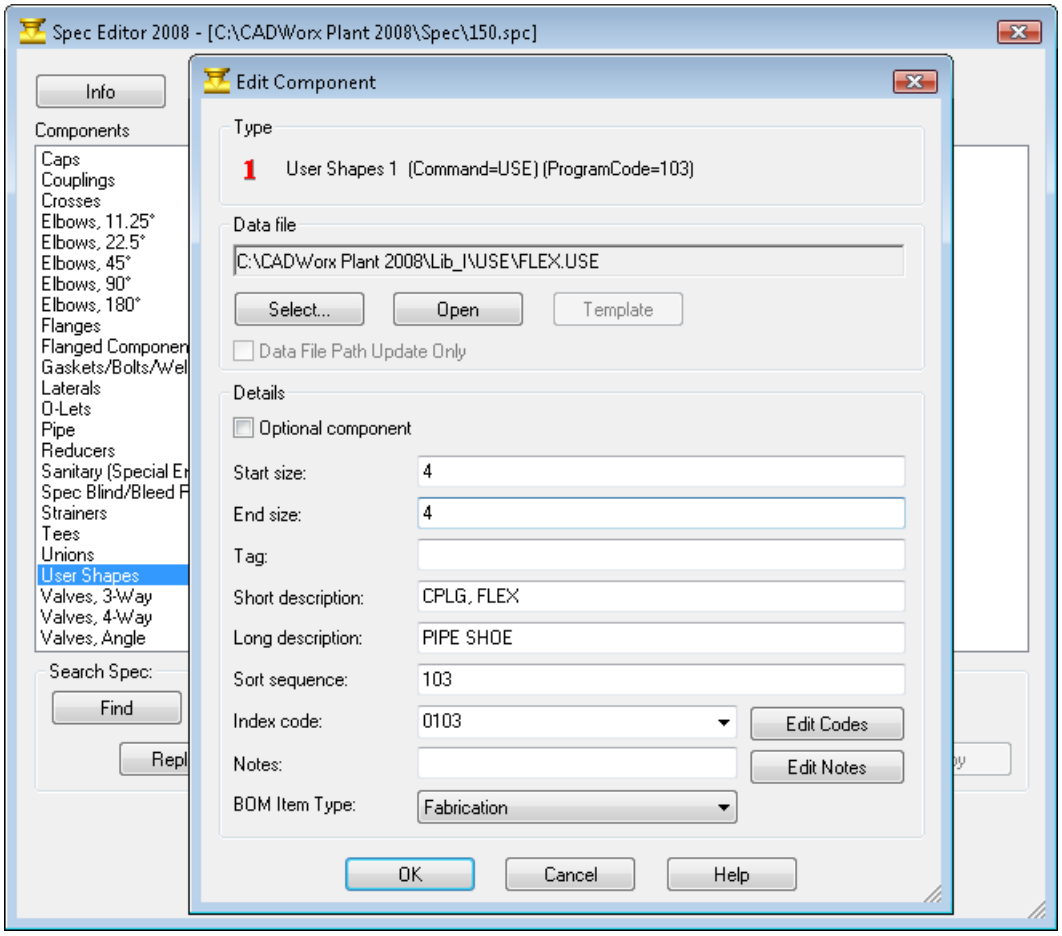
If the component is less than 1/16" long (2 mm when in metric mode), the associated xdata will be in an inserted block named “XATB”. It will be located on the default layer named “SYSTEM” instead of the “CL” layer so that it will not plot or be visible while this layer is turned off.

If the component is a non-symmetrical type, located at the start point will be an inserted block named “NATB”. This block, like the “XATB” block will contain all the associated xdata and will act as any other component. When a 2D representation of a non-symmetrical component is not present (only the 3D representation option above was used) and the component is converted to the 3D mode, the “NATB” will remain. This block can be converted to 2D but will not have any graphics in the 3D mode. This “NATB” and “XATB” block can be moved and controlled just as any other CADWorx component. If the 2D representation of a non-symmetrical user shape is not present, all that appears in an automatic isometric will be this “NATB” block. The bill of material will still treat it as a CADWorx Plant component.

Note: With the use of optional components, a specification file can have an unlimited number of user shapes.

The next step required is to add the user shape to the desired specification with the Specification Editor (see page 254).

Command: **SPECEDIT**


Spec Editor 2008 - [C:\CADWorx Plant 2008\Spec\150.spc]
Info
Components
Caps
Couplings
Crosses
Elbows, 11.25°
Elbows, 22.5°
Elbows, 45°
Elbows, 90°
Elbows, 180°
Flanges
Flanged Componen
Gaskets/Bolts/Wel
Laterals
O-Lets
Pipe
Reducers
Sanitary (Special Er
Spec Blind/Bleed F
Strainers
Tees
Unions
User Shapes
Valves, 3-Way
Valves, 4-Way
Valves, Angle
Search Spec:
Find
Repl
Edit Component
Type
1
User Shapes 1 (Command=USE) (ProgramCode=103)
Data file
C:\CADWorx Plant 2008\Lib_I\USE\FLEX.USE
Select...
Open
Template
Data File Path Update Only
Details
Optional component
Start size:
4
End size:
4
Tag:
Short description:
CPLG, FLEX
Long description:
PIPE SHOE
Sort sequence:
103
Index code:
0103
Edit Codes
Notes:
Edit Notes
BOM Item Type:
Fabrication
OK
Cancel
Help

From this dialog, find the USER SHAPES group on the left side. With the *Add* button, add USER1 to the list box on the right. Use the *Edit* button from the bottom of the main dialog box and browse for the <INSTALL-DIR>\LIB_I\USE\FLEX.USE file. Enter the *Short description:* CLPG, FLEX and the *Long description:* COUPLING, FLEXIBLE, AMCE MODEL #356. The starting and ending sizes could also be modified in the appropriate boxes. CADWorx Plant created the library <INSTALL-DIR>\LIB_I\USE in the installation process. After making the modifications, select the *OK* button to end the specification editing process and to return to the drawing.

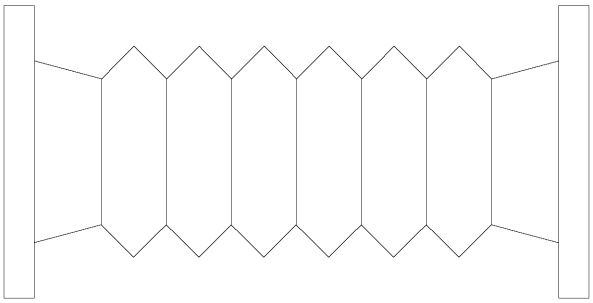
Next prepare the specification for use.

Command: **SPEC**
Specification <>: *150*
Prepared E:\CADWORX PIPE\RELEASE 15\SPEC\150.SPC specification...

Reselect User Shapes from the toolbar shown previously, and select the button with the number 1 on it. This button coincides with the USER1 entry just made in the specification. For a symmetrical shape, a simple start point and direction is all this is required.

Command: **USE** *(note: this command must be run from the pulldown menu)*
Pick start point <last point>: *pick point*
Pick direction: *pick direction*
4" COUPLING, FLEXIBLE ACME MODEL #356

The example in this section provides a 4" flexible coupling as shown. The component can be used just like any other component. It allows material extraction, automatic labeling, conversion from 2D to 3D modes, etc. The user could repeat this process to create more user shapes with any other sizes.



For non-symmetrical shapes, the prompts are slightly different. Prior to picking or entering the rotation, select any of the other options to orient the non-symmetrical user shape.

Command: **USE** (note: this command must be run from the menu)
4" COUPLING, FLEXIBLE ACME MODEL #356
Pick location: pick point
[**Up/Down/Roll/mirrorX/mirrorY/mirrorZ**] <Pick rotation>:

Up

This option rolls the component by 90 degrees. It will turn the component up if the non-symmetrical component has a branch like extrusion.

Down

This option rolls the component by –90 degrees. It will turn the component down if the non-symmetrical component has a branch like extrusion.

Roll

This option allows the non-symmetrical user shape to be rolled at any angle. Please note that when the component has been flipped with one of the options below, the angle might need to be negative. If the rolled angle input is incorrect, just re-type the angle as a negative to correct it. In addition, when the component has been turned up or down with one of the options above, rolling will be from that location.

mirrorX

This option flips or mirrors the non-symmetrical user shape along the X axis. Once a shape has been flipped or mirrored, it will have an affect on all the options above. Simply running this option again can place the component back to its prior position.

mirrorY

This option flips or mirrors the non-symmetrical user shape along the Y axis. Once a shape has been flipped or mirrored, it will have an affect on all the options above. Simply running this option again can place the component back to its prior position.

mirrorZ

This option flips or mirrors the non-symmetrical user shape along the Z axis. Once a shape has been flipped or mirrored, it will have an affect on all the options above. Simply running this option again can place the component back to its prior position.

<Pick rotation>

This option requires a picked location or an angle to be entered. This option causes the command to exit and the options above would become un-available. This should be the last option chosen.

Note: Non-symmetrical shapes can only convert between isometric and solids (CONVERTISO or CONVERTSOLID)

Spacer/Bleed Ring

Command line: BLD

See Appendix D type 1 (see page 635) prompts for this component.

Spectacle Blind

Command line: SPCT **(also** VSPCT**)**

Command: **SPCT**
4" BLIND, SPECTACLE, 150LB, ASTM A-105
Pick start point or [**Thickness**] <last point>:
Pick direction:

Thickness

This option allows the user to enter a thickness needed for the spectacle blind. If this option is not used, the program will get the thickness from the data file associated with this component.

Enter an option [**Open/Closed**]:

This option allows the user to show the spectacle blind open or closed. This will be important when an automatic isometric is generated using this component. It shows the proper spectacle symbol indicating the component is open or closed.

Closed

Select to close the flow and show an open spectacle blind on the tie bar end.

Open

Select to open the flow and show a closed spectacle blind on the tie bar end.

Enter degree direction [**0 90 180 270**] <0>:

Use these options to locate the spectacle blind about the piping.

270

Select to place the tie bar direction away from the user.

180

Select to place the tie bar direction in the down position.

90

Select to place the tie bar direction toward the user.

0

Select to place the tie bar direction in the up position.

Reinforcing Pad and Saddle

Command line: REINFSAD **(also** REINFPAD**)**

This function allows the attachment of reinforcing pads and saddles to different branch connections. This function places a symbolic letter ("P" or "S") onto the drawing which will be used in bill of material generation and with the CAESAR II (pipe stress analysis software) bi-directional link. This symbolic letter will have xdata attached which is used by both these facilities.

Command: **REINFSAD**

Select centerline of main: *select a centerline of the main run*
Select centerline of branch: *select the centerline of the branch run*

Enter thickness: *enter a pad or saddle thickness (i.e.: 0.25)*
Enter O.D. of reinforcing sad: *enter an approximate distance (i.e.: 12)*
Enter weight: *enter weight (i.e.: 12.5)*

Note: To show reinforcing pads in ISOGEN isometric drawings, review the topic on the OTAP (see page 145) component.

Expansion Joint

Command line: EXPF

Command: **EXPF**
2" JOINT, EXPANSION, 150LB, FLGD
Pick start point or [**Length**] <last point>: *pick start point*
Pick direction: *pick a direction*

Length

This option allows the length to be entered at the command line instead of using the length from the data file listed in the specification.

Enter length: *enter a real (ex. 12)*

Rigid Element

Command line: ELEMENT

This component can be used with CAESAR II (pipe stress analysis software) when a rigid component is required. A good example would be using a rigid from the base of a pump to the pump discharge piping. Another good example would be placing a rigid as a structural member that might be supporting the pipe.

When using the CAESAR II bi-directional link or importing a stress input file, rigids may be automatically placed. These rigids are used by the bi-directional link and should not be modified in any manner. They can be moved with a piping system, but they need to remain in their relative location. They are often used when the length of a CAESAR II component does not match the components contained within CADWorx Plant.

These elements are represented with a polyline that has pointed ends. They are placed on the "System" layer, therefore they can be turned off when plotting.

Command: **ELEMENT**
Pick start point for CAESAR II element: *pick a point*
Pick second point: *pick a point*

Generic Attachment

Command line: GENERIC

This routine attaches intelligence to any graphics within the drawing. It later can be used in bill of material extraction, labeling, etc. for various equipment or components not provided with CADWorx Plant. After manually constructing a piece of equipment or a component, use this routine to attach xdata information.

To pinpoint the center of gravity locations use the Osnap commands. For example, to place information on a pump and have it reflect the CG correctly, draw a centerline and pick MIDpoint or NEArest to the end of the centerline toward the electrical motor or the exact known location.

Command: **GENERIC**

Enter abbreviated description: *enter string (ex: PUMP 2X1X8)*
Enter complete description: *enter string (ex: PUMP, INGERSOLL-RAND, MODEL HOC, 2X1X8)*
Enter weight: *enter real (ex. 180)*
Enter index code <0000>: *enter part number*
Enter sort location number <0>: *for bill of material schedule location*
Assign line number [Yes/No] <Yes>: *enter for yes and N for no*
Enter main size: *enter real (ex. 4)*
Select object: *select any object*

Stop Sign

Command line: STOPSIGN

This symbol is used for designating the starting and ending locations for automatically generated isometric output. Stop signs are used with the CADWorx Plant AutoISO (see page 183) isometric generator and the ISOGEN isometric generator. When running the AutoISO automatic isometric generator, ISOOUT (see page 185), and using the *Database* option, stop signs break the isometrics into different systems starting and ending anywhere there is a stop sign. Stop signs must be at a weld or connection point on the piping system. The user places these designators on the drawing and then exports them to the database. This could occur manually with DBFGEN (see page 236) or automatically with LIVEDB (see page 231) (the live database ability incorporated into CADWorx Plant).

If stop signs are present when using the *Database* option of the ISOOUT command, the user is asked if stop signs should be used in generating the ISOs.

Use Stop Signs [Yes/No] <Yes>:

If YES is selected, continuity is checked and the ISOs are broken as a stop sign is encountered. If there are no line numbers or stop signs present in the database, the following prompt is displayed.

Database does not have any line number or stop signs...

Note: When exporting with ISOGEN isometric generator (ISOGENOUT (see page 214) or ISOGENBATCH) (see page 214), stop signs are used if selected.

Tap

Command line: TAP

The TAP command creates tap connections for use by ISOGEN. The Tap connection are for use with small branches. After creating a tap connection for a component, piping can be routed from that point. The CADWorx ISOGEN interface automatically processes numerous types of tap without needing a tap connections. Only use a tap connection if ISOGEN reports discontinuity.

Note: Verify that main size is set to the size of the tap before drawing this component.

Command: **TAP**
Pick point: *pick a point for the tap. This point is normally the endpoint of the tapping component.*
Select entity: *select the center line of the component that is being taped.*

Offset Tap

Command line: OTAP

The OTAP command creates offset tap connections for use by ISOGEN. Offset taps are for use with large branches. After creating a offset tap connection for a component, piping can be routed from that point. Use offset tap for all major branches of the main pipeline. Offset tap is more robust at processing branches then tap connection. Use a offset tap connection if ISOGEN reports discontinuity or tap connection does not produce the desired results.

Note: Verify that main size is set to the size of the tap before drawing this component.

Command: **OTAP**
Pick branch end point: *pick a point for the tap. This point is normally the inner most endpoint of the tapping component.*
Pick second point: *pick a point on the center line of the component that is being taped.*

The OTAP command allows several types of pipe-to-pipe connections. The default connection for OTAP is TANGENTIAL (TTSO). To change to another type of connection, change the SKEY value in CEDIT -> ISOGEN Data dialog for the OTAP component.

SKEY	Connection Type
TTSO	TANGENTIAL
TTRF	TANGENTIAL REINFORCED
TOSO	OFFSET
TORF	OFFSET REINFORCED
TSSO	STUB IN
TSRF	STUB IN REINFORCED
TESO	SET ON
TERF	SET ON REINFORCED

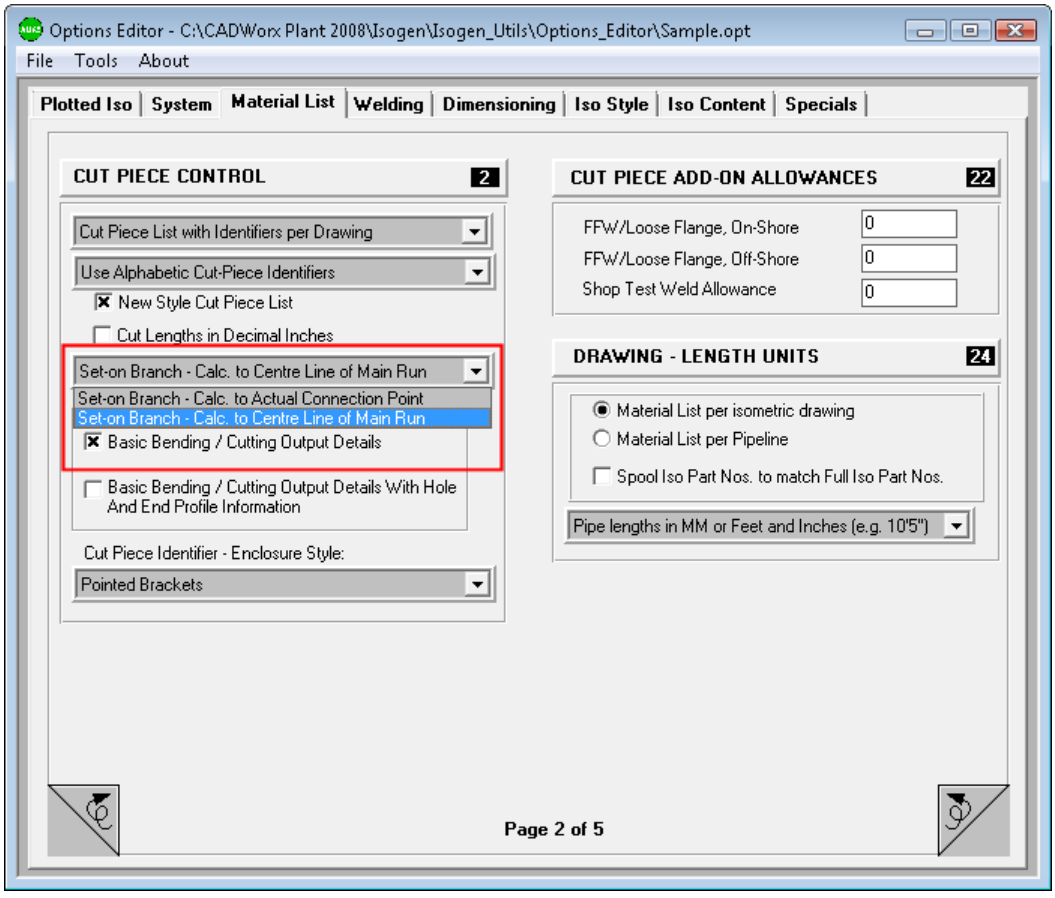
If the OTAP connects two pipes that are the same size, using the following SKEY will produce these results:

- TERF, TESO, TSRF, TSSO:
- The length of pipe for the cut list will be increased to include the OTAP length.
- The dimension will include OTAP length

If the OTAP connects two pipes that are the same size, using the following SKEY will produce these results:

- TORF, TOSO, TTRF, TTSO:
- The actual length of pipe drawn in the model will be used for cut list.
- The dimension will not include OTAP length

If the OTAP connects two pipes that are of different sizes (reduction), ISOGEN Option Switch 2 is used for cut length calculation.



"Set-on Branch - Calc. to Centre Line of Main Run" setting with SKEYs: TERF, TESO, TSRF, TSSO:

- The length of pipe for the cut list will be increased to include the OTAP length.
- The dimension will include OTAP length

"Set-on Branch - Calc. to Centre Line of Main Run" setting with SKEYs: TORF, TTRF, TTSO, TOSO:

- The actual length of pipe drawn in the model will be used for cut list.
- The dimension will not include OTAP length

"Set-on Branch - Calc. to Actual Connection Point" setting with SKEYs: TERF, TESO, TSRF, TSSO:

- The actual length of pipe drawn in the model will be used for cut list.
- The dimension will include OTAP length.

"Set-on Branch - Calc. to Actual Connection Point" setting with SKEYs: TORF, TTRF, TTSO:

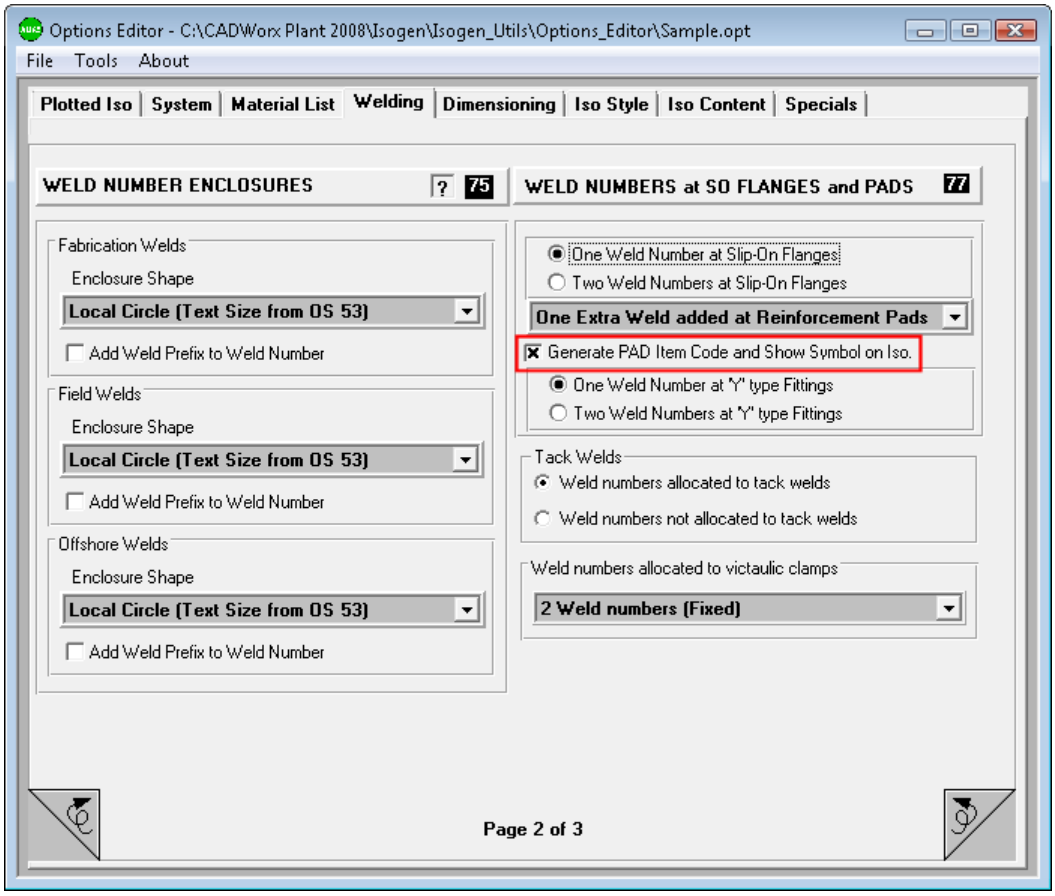
- The length of pipe for the cut list will be adjusted based on the diameter of the main and reduction pipe sizes.
- The dimension will not include OTAP length.

"Set-on Branch - Calc. to Actual Connection Point" setting with SKEYs: TOSO:

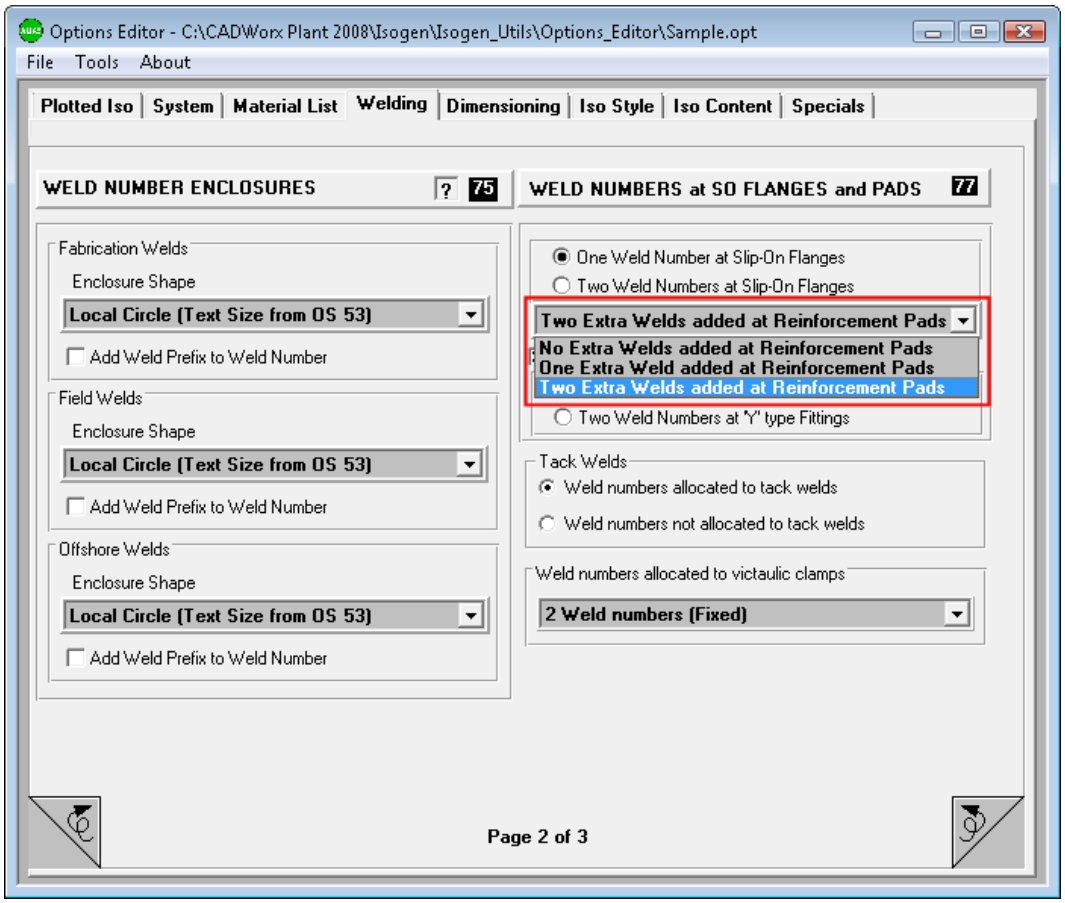
- The actual length of pipe drawn in the model will be used for and cut list.
- The dimension will not include OTAP length.

Option switch 77 "Generate PAD Item Code and Show Symbol on Iso" setting will show a reinforcement pad on OTAP connections with SKEY of: TTRF, TORF, TERF. A CADWorx reinforcement pad does not have to be drawn in the model for these three SKEYs.

For all other SKEYs, a CADWorx reinforcement pad has to be drawn in the model for it to appear in the ISO drawing.


Options Editor - C:\CADWorx Plant 2008\Isogen\Isogen_Utils\Options_Editor\Sample.opt
File
Tools
About
Plotted Iso
System
Material List
Welding
Dimensioning
Iso Style
Iso Content
Specials
WELD NUMBER ENCLOSURES
?
75
WELD NUMBERS at SO FLANGES and PADS
77
Fabrication Welds
Enclosure Shape
Local Circle (Text Size from OS 53)
Add Weld Prefix to Weld Number
Field Welds
Enclosure Shape
Local Circle (Text Size from OS 53)
Add Weld Prefix to Weld Number
Offshore Welds
Enclosure Shape
Local Circle (Text Size from OS 53)
Add Weld Prefix to Weld Number
One Weld Number at Slip-On Flanges
Two Weld Numbers at Slip-On Flanges
One Extra Weld added at Reinforcement Pads
Generate PAD Item Code and Show Symbol on Iso.
One Weld Number at 'Y' type Fittings
Two Weld Numbers at 'Y' type Fittings
Tack Welds
Weld numbers allocated to tack welds
Weld numbers not allocated to tack welds
Weld numbers allocated to victaulic clamps
2 Weld numbers (Fixed)
Page 2 of 3

ISOGEN option switch 77 "No Extra Welds added at Reinforcement Pads", "One Extra Weld added at Reinforcement Pads", "Two Extra Welds added at Reinforcement Pads", setting will control weld numbering at reinforcement pads. Extra welds are only shown for the following SKEYs: TTRF, TORF, TERF



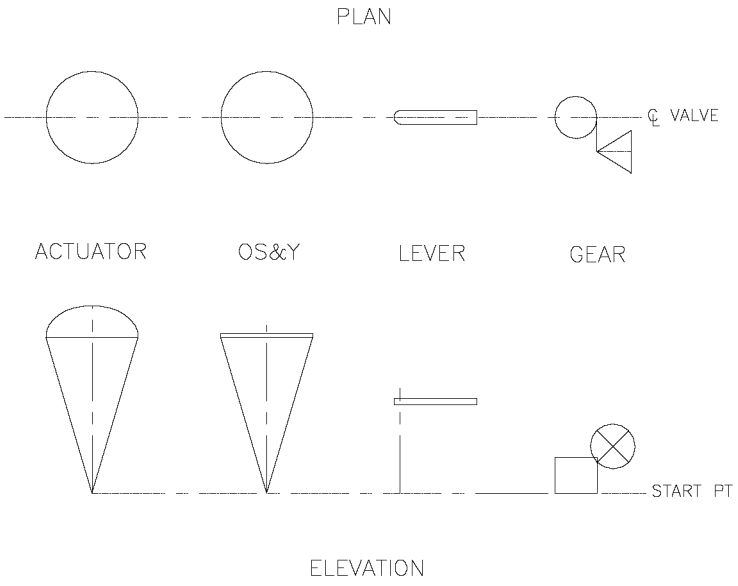
Operators

Plant menu: ▶ Toolbars ▶ Operators



These routines are for drawing various valve accessories. These accessories will be drawn according to the present mode (2D or 3D). There are two main options to choose from, *Plan* and *Elevation*. *Plan* is for drawing the valve accessories in the Z direction of the present UCS (upward or downward). The *Elevation* option is for drawing in the present X and Y directions of the UCS.

Some icons have a small arrow in the lower right corner that indicates a flyout for more options. Commands appearing in braces appear as flyouts.



Actuator

Command line: OPERELEV **(and** OPERPLAN**)**

Elevation

2D and 3D mode...

Command: **OPERELEV**
Pick center of valve: *pick point, Osnap, INSert of, a valve*
Pick direction: *pick direction*
Enter actuator outside diameter: *enter real number*
Enter distance, center of valve to top of actuator: *enter real number*

Plan

3D mode...

Command: **OPERPLAN**
Pick center of valve: pick point, Osnap, INSert of, a valve
Enter actuator outside diameter: enter real number
Enter distance, center of valve to top of actuator: enter real number

2D mode...

Command: **OPERPLAN**
Pick center of valve: *pick point, Osnap, INSert of , a valve*
Enter actuator, or handwheel, outside diameter: *enter real number*

Outside Stem & Yoke

Command line: OSYELEV **(and** OSYPLAN**)**

Elevation

2D and 3D mode...

Command: **OSYELEV**
Pick center of valve: *pick point, Osnap, INSert of, a valve*
Pick direction: *pick direction*
Enter handwheel outside diameter: *enter real number*
Enter distance, center of valve to handwheel: *enter real number*

Plan

3D mode...

Command: **OSYPLAN**
Pick center of valve: *pick point, Osnap, INSert of, a valve*
Enter handwheel outside diameter: *enter real number*
Enter distance, center of valve to handwheel: *enter real number*

2D mode...

Command: **OSYPLAN**
Pick center of valve: *pick point, Osnap, INSert of, a valve*
Enter actuator, or handwheel, outside diameter: *enter real number*

Lever

Command line: LEVERELEV **(also** LEVERPLAN**)**

Elevation

2D and 3D mode...

Command: **LEVERELEV**
Pick center of valve: *pick point, Osnap, INSert of, a valve*
Pick stem length-direction or enter @x,y: *pick point*
Pick lever length-direction or enter @x,y: *enter @x,y (ex: @4<45)*

Length-direction allows the user to pick a designated length and direction at the same time. The user can also give these prompts a polar distance such as @8<45 or @8,4.

Plan

3D mode...

Command: **LEVERPLAN**
Pick center of valve: *pick point, Osnap, INSert of a valve*
Enter stem length, from center valve to lever: *enter real number*
Pick lever length-direction or enter @x,y: *enter @x,y (ex: @5,5)*

2D mode...

Command: **LEVERPLAN**
Pick center of valve: *pick point, Osnap, INSert of, a valve*
Pick lever length-direction or enter @x,y: pick point

Length-direction allows the user to pick a designed length and direction at the same time. The user can also give these prompts a polar distance, such as, @8<45 or @8,4.

Gear

Command line: GEARELEV **(also** GEARPLAN**)**

Elevation

2D mode...

Command: **GEARELEV**
Pick bottom of gear: *pick point above centerline of valve*
Enter gear O.D.: *enter real number*
Enter height of gear: *enter real number*
Pick horizontal direction to handwheel: *pick direction*
Enter horizontal distance to handwheel: *enter real number*
Pick vertical direction to handwheel: *pick direction*
Enter vertical distance to handwheel: *enter real number*
Enter handwheel O.D.: *enter real number*

3D mode...

Command: **GEARELEV**
Pick center of valve: *pick point, Osnap, INSert of, a valve*
Enter distance, center of valve to bottom of gear: *enter real number*
Enter gear O.D.: *enter real number*
Enter gear height (bottom-top): *enter real number*
Pick vertical direction of gear: *pick direction*
Pick horizontal direction to handwheel: *pick direction*
Enter horizontal distance, center of valve to handwheel: *enter real number*
Enter vertical distance, bottom of gear to handwheel: *enter real number*
Enter z distance, center of valve to handwheel face: *enter real number*
Enter handwheel O.D.: *enter real number*

Z distance can be described as the distance in a vector extending out of the present X, Y plane (toward or away from the user). Horizontal and vertical directions given in the prompts are for advice only. They can be reversed where horizontal can be vertical and vertical can be horizontal. If the directions are equivalent, the user might get an error message as shown below and the routine will abort.

Angle to handwheel and gear are same, aborted...

Plan

2D mode...

Command: **GEARPLAN**
Pick center of valve: *pick point, Osnap, INSert of, a valve*
Enter gear O.D.: *enter real number*
Pick direction to handwheel stem: *pick direction*
Enter distance to handwheel stem: *enter real number*
Pick direction to handwheel face: *pick direction*
Enter distance to handwheel face: *enter real number*
Enter handwheel O.D.: *enter real number*

3D mode...

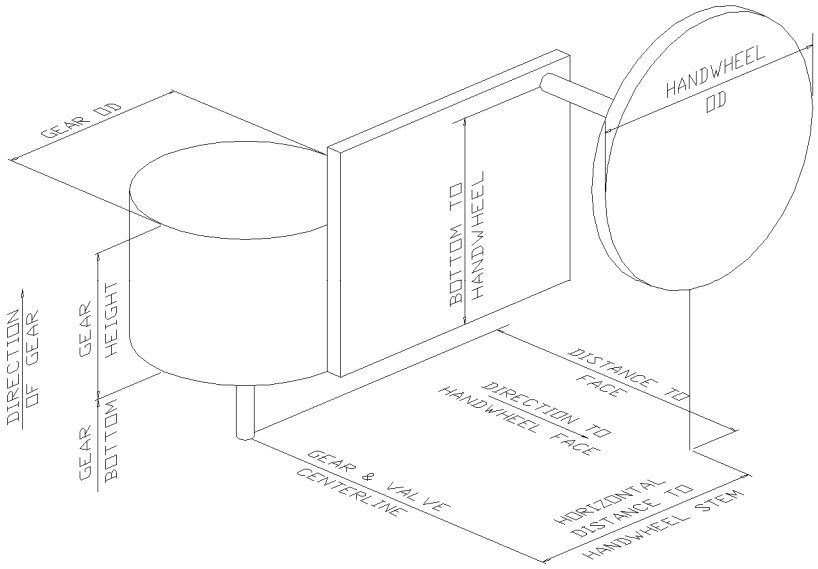
Command: **GEARPLAN**
Pick center of valve: *pick point, Osnap, INSert of, a valve*
Enter vertical distance, center of valve to bottom of gear: *enter real number*
Enter gear O.D.: *enter real number*

Enter gear height (bottom-top): *enter real number*
Pick direction to handwheel stem: *pick direction*
Enter distance to handwheel stem: *enter real number*
Pick direction to handwheel face: *pick direction*
Enter distance to handwheel face: *enter real number*
Enter handwheel O.D.: *enter real number*
Enter vertical distance, bottom of gear to center handwheel: *enter real number*

Distances to the stem or face are from the center of the valve. The directions, or the angle of the handwheel stem and the handwheel face should be 90 degrees apart. This will be true for both 2D and 3D modes. If these directions are the same, the following error message will appear and the routine will abort.

Angle to handwheel face and stem are same, aborted...

Shown below are the distances and locations for various prompts required by the gear operator.



Line Numbers

Plant menu: ▶ **Toolbars** ▶ **Line Numbers**



Line Number Setup

Command line: NUMBERSETUP

Line Numbers toolbar:

Please refer to Line number setup (see page 289) for more information regarding this command.

Line Number Assign

Command line: NUMBERASSIGN

Line Numbers toolbar:

Please refer to Line number assign (see page 284) for more information regarding this command.

Line Number Annotate

Command line: NUMBERANNOTATE

Line Numbers toolbar:

Please refer to Line number annotate (see page 285) for more information regarding this command.

Line Number Set By

Command line: NUMBERSET

Line Numbers toolbar:

Please refer to Line number set by (see page 286) for more information regarding this command.

Line Number Count

Command line: NUMBERCOUNT

Line Numbers toolbar: 113

Please refer to Line number count (see page 288) for more information regarding this command.

Line Number Count Increment

Command line: NUMBER+

Line Numbers toolbar: +

Please refer to Line number count increment (see page 288) for more information regarding this command.

Line Number Count Decrement

Command line: NUMBER-

Line Numbers toolbar:

Please refer to Line number count decrement (see page 288) for more information regarding this command.

UCS

Plant menu: ▶ **Toolbars** ▶ **UCS**



View Flat

Command line: VIEWFLAT

UCS toolbar:

Please refer to Point and Shoot UCS (see page 506) for more information regarding this command.

Note: If running CADWorx Equipment, run the command EQUIPVIEWFLAT

View North

Command line: VIEWNORTH

UCS toolbar:

Please refer to Point and Shoot UCS (see page 506) for more information regarding this command.

Note: If running CADWorx Equipment, run the command EQUIPVIEWNORTH

View South

Command line: VIEWSOUTH

UCS toolbar: 

Please refer to Point and Shoot UCS (see page 506) for more information regarding this command.

Note: If running CADWorx Equipment, run the command EQUIPVIEWSOUTH

View East

Command line: VIEWEAST

UCS toolbar: 

Please refer to Point and Shoot UCS (see page 506) for more information regarding this command.

Note: If running CADWorx Equipment, run the command EQUIPVIEWEAST

View West

Command line: VIEWWEST

UCS toolbar: 

Please refer to Point and Shoot UCS (see page 506) for more information regarding this command.

Note: If running CADWorx Equipment, run the command EQUIPVIEWWEST

UCS Next

Command line: UCSNEXT

UCS toolbar: 

Please refer to UCS Next (see page 509) for more information regarding this command.

UCS Object

Command line: UCS

UCS toolbar: 

Please refer to UCS Object (see page 509) for more information regarding this command.

Compass

Command line: COMPASS

UCS toolbar:

Please refer to Compass (see page 510) for more information regarding this command.

Steel

Plant menu: ▶ **Toolbars** ▶ **Steel**



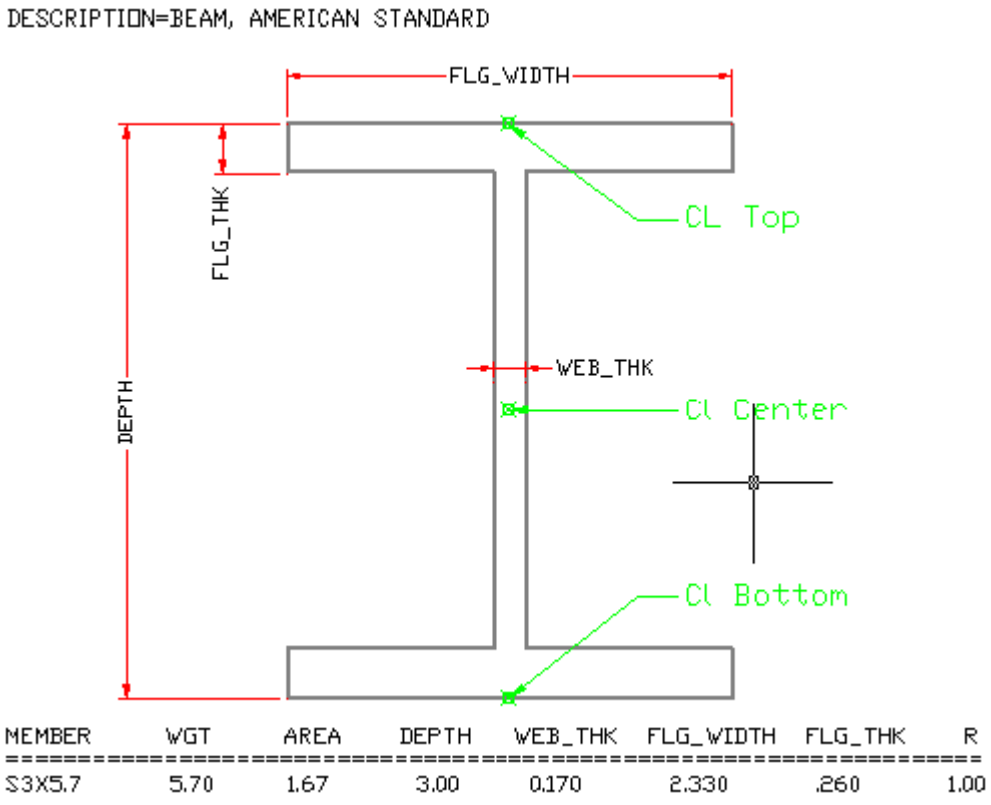
Wide Flange Shapes

Command line: WSHAPE

Steel toolbar: 

Please refer to Steel Shapes (see page 426) for more information regarding this command.

Data file sample (TYPE=W)



MEMBER	WGT	AREA	DEPTH	WEB_THK	FLG_WIDTH	FLG_THK	R
S3X5.7	5.70	1.67	3.00	0.170	2.330	.260	1.00

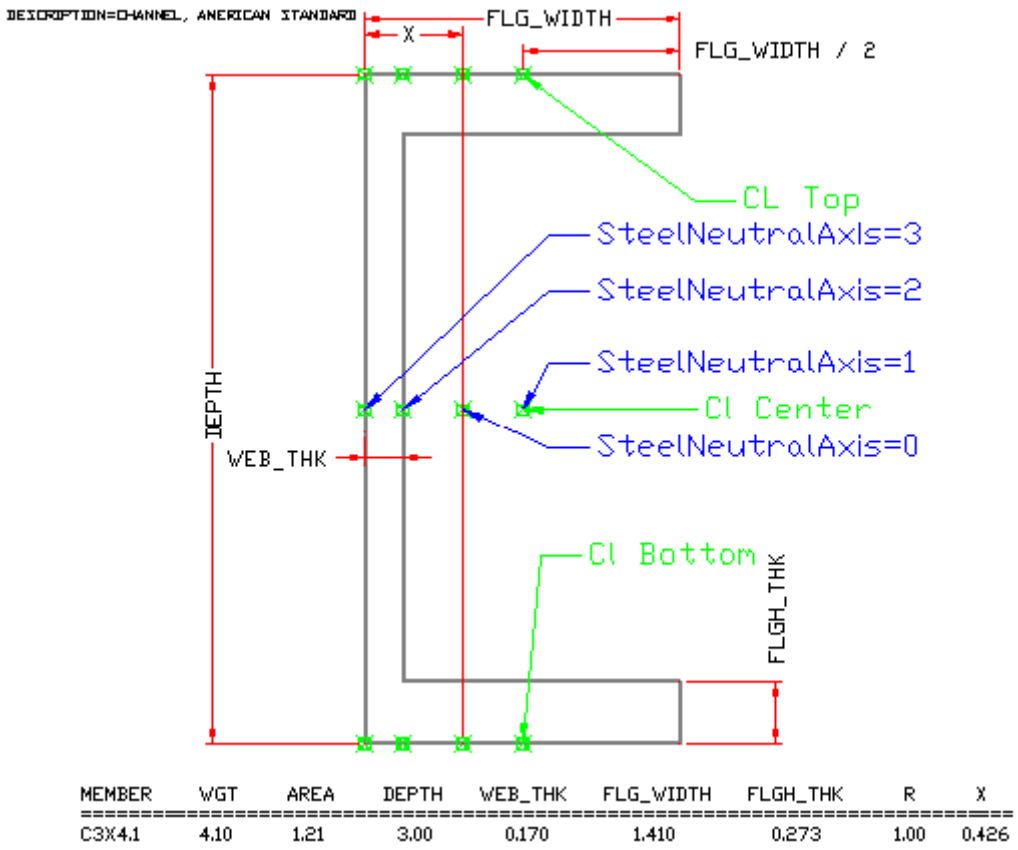
Channel Shapes

Command line: CSHAPE

Steel toolbar:

Please refer to Steel Shapes (see page 426) for more information regarding this command.

Data file sample (TYPE=C)



MEMBER	WGT	AREA	DEPTH	WEB_THK	FLG_WIDTH	FLGH_THK	R	X
C3X4.1	4.10	1.21	3.00	0.170	1.410	0.273	1.00	0.426

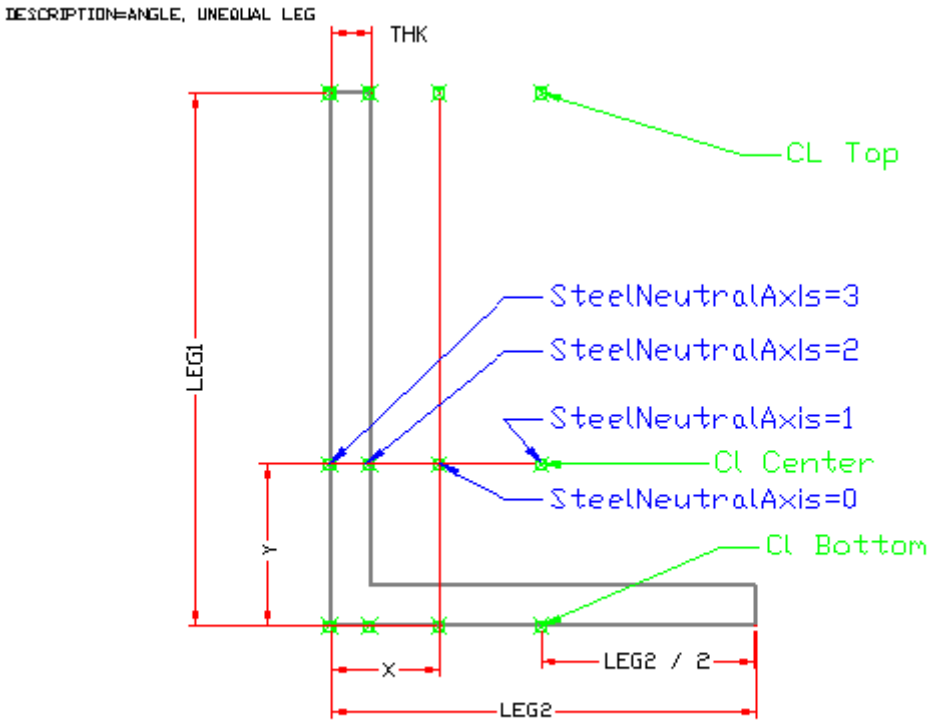
Angle Shapes

Command line: LSHAPE

Steel toolbar:

Please refer to Steel Shapes (see page 426) for more information regarding this command.

Data file sample (TYPE=L)



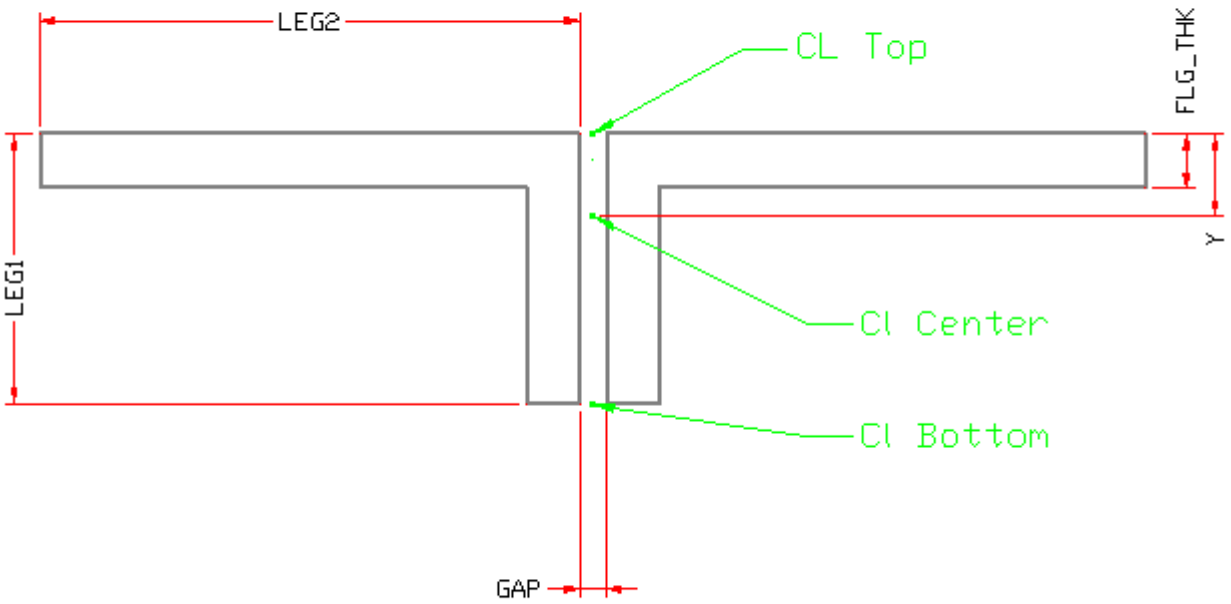
Angle Double Shape

Command line: **2**LSHAPE

Please refer to Steel Shapes (see page 426) for more information regarding this command.

Data file sample (TYPE=2L)

```
DESCRIPTION=Double Angle
```



MEMBER	WEIGHT	AREA	LEG1	LEG2	THK	GAP	Y
Un_Equal_Legs	1.0	1.0	100	200	20	10	10

Tee Shapes

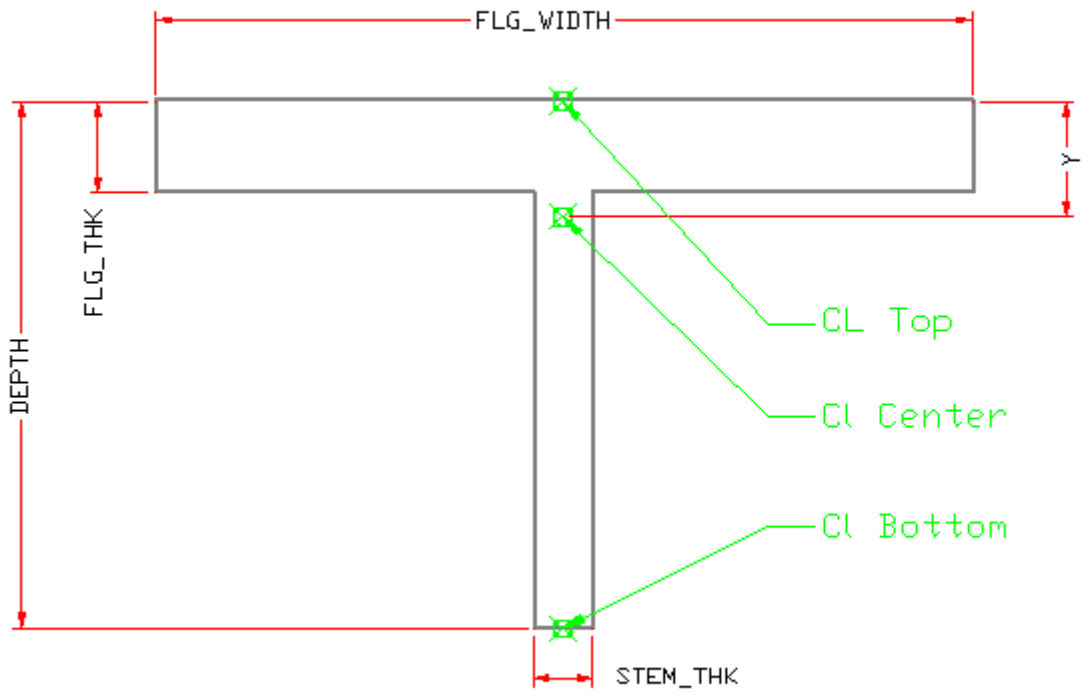
Command line: TSHAPE

Steel toolbar:

Please refer to Steel Shapes (see page 426) for more information regarding this command.

Sample data file (TYPE-WT):

```
DESCRIPTION=TEE, AMERICAN STANDARD BEAN
```



MEMBER	WGT	AREA	DEPTH	STEM_THK	FLG_WIDTH	FLG_THK	R	Y
ST15X2.85	2.850	0.835	1.50	0.170	2.330	0.260	1.00	0.329

Flat Bar Shapes

Command line: FBSHAPE

Steel toolbar:

Please refer to Steel Shapes (see page 426) for more information regarding this command.

Round Bar Shapes

Command line: RBSHAPE

Steel toolbar:

Please refer to Steel Shapes (see page 426) for more information regarding this command.

Pipe Shapes

Command line: **P**SHAPE

Steel toolbar:

Please refer to Steel Shapes (see page 426) for more information regarding this command.

Tube Shapes

Command line: TSSHAPE

Steel toolbar:

Please refer to Steel Shapes (see page 426) for more information regarding this command.

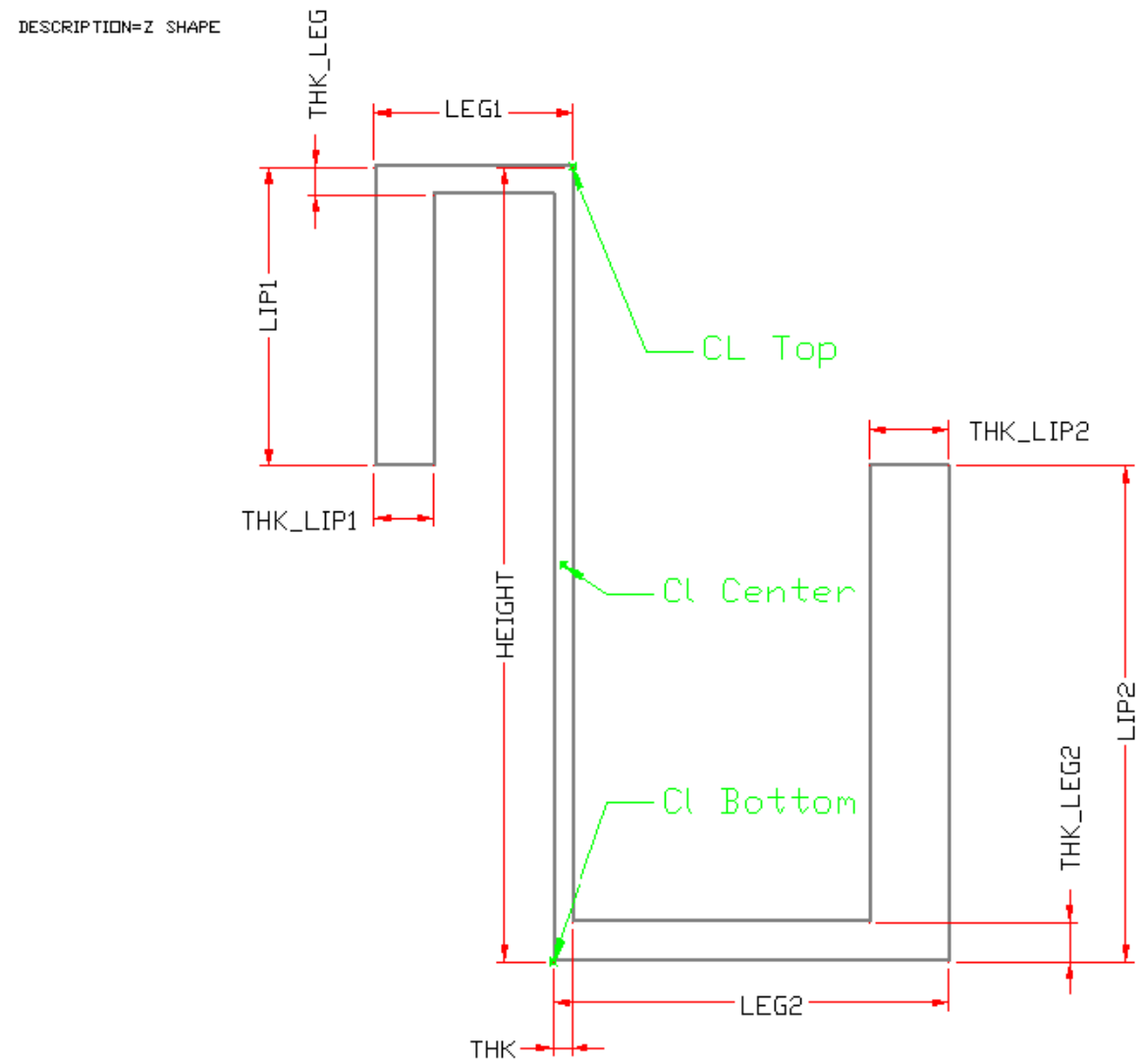
Z Shapes

Command line: ZSHAPE

Please refer to Steel Shapes (see page 426) for more information regarding this command.

Data file sample (TYPE=Z):

DESCRIPTION=Z SHAPE



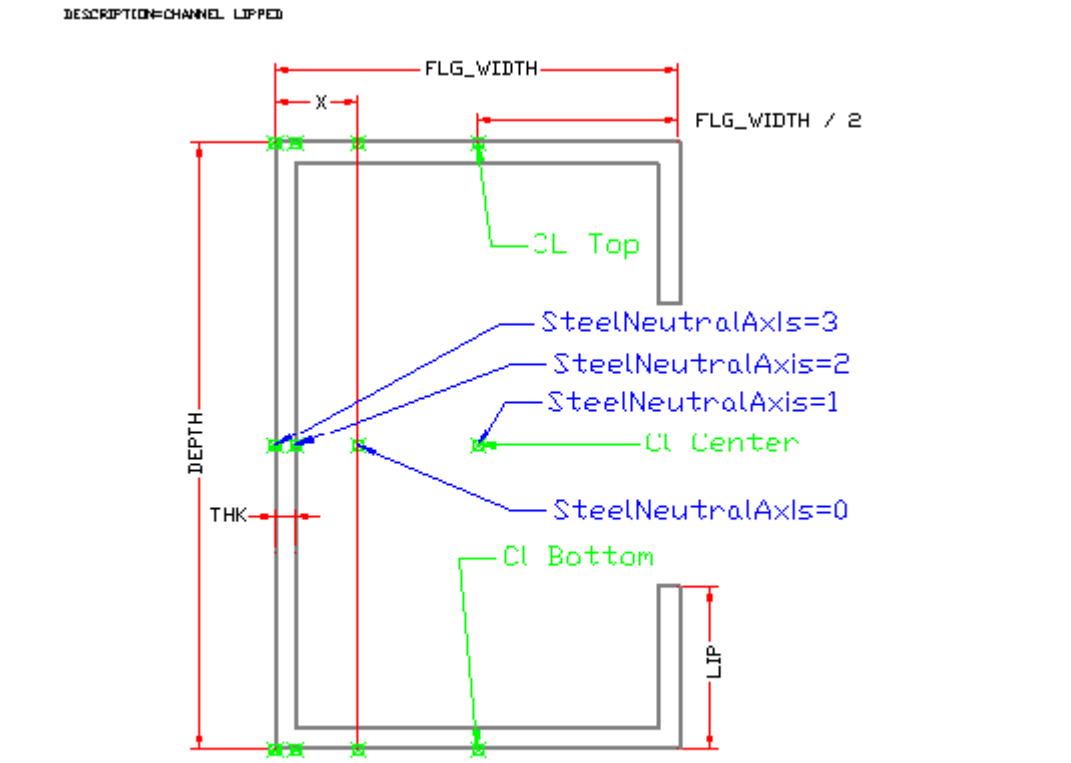
MEMBER	MAT	AREA	HEIGHT	THK	LEG1	LIP1	LEG2	LIP2	THK_LEG1	THK_LEG2	THK_LIP1	THK_LIP2
Variable_Lip_Thickness	1.00	1.0	200.0	5.0	50.0	75.0	100.0	125.0	7	10	15	20

Channel Shape with Lip

Command line: CLIPSHAPE

Please refer to Steel Shapes (see page 426) for more information regarding this command.

Data file sample (TYPE=C_LIP):

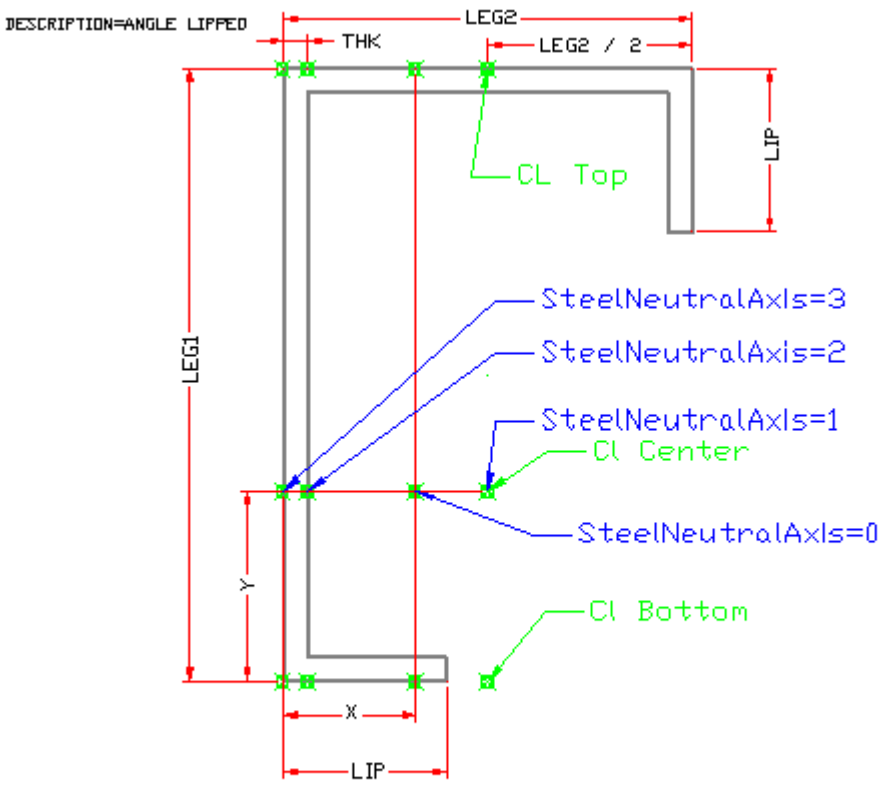


Channel Shape with Lip

Command line: LLIPSHAPE

Please refer to Steel Shapes (see page 426) for more information regarding this command.

Data file sample (TYPE=L_LIP):



MEMBER	WGT	AREA	LEG1	THK	LEG2	LIP	X	Y
75x50x20x2.0	1.00	1.00	75.0	2.0	50.0	20.0	20	30

Plate

Command line: SPLATE

Steel toolbar:

Please refer to Plates (see page 432) for more information regarding this command.

Base Plate

Command line: SBPLATE

Steel toolbar: 

Please refer to Base Plate (see page 453) for more information regarding this command.

Cope

Command line: SCOPE

Steel toolbar: 

Please refer to Cope (see page 435) for more information regarding this command.

Cut

Command line: SCUT

Steel toolbar: 

Please refer to Cut (see page 436) for more information regarding this command.

Miter

Command line: SMITER

Steel toolbar: 

Please refer to Miter (see page 437) for more information regarding this command.

Union

Command line: SUNION

Steel toolbar: 

Please refer to Union (see page 438) for more information regarding this command.

Edit

Command line: SCEDIT

Steel toolbar: 

Please refer to Edit (see page 455) for more information regarding this command.

CG Generator

Command line: SCG

Steel toolbar: 

Please refer to CG Generator (see page 439) for more information regarding this command.

Auto Cope

Command line: SAUTOCOPE

Steel toolbar: 

Please refer to Automatic Coping (see page 454) for more information regarding this command.

Frame

Command line: SFRAME

Steel toolbar: 

Please refer to Frame for more information regarding this command.

Ladder

Command line: SLADDER

Steel toolbar: 

Please refer to Ladder (see page 440) for more information regarding this command.

Stair

Command line: SSTAIR

Steel toolbar: 

Please refer to Stair (see page 446) for more information regarding this command.

Hand Rail

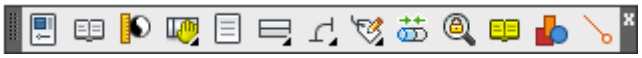
Command line: SHRAIL

Steel toolbar: 

Please refer to Hand Rail (see page 450) for more information regarding this command.

Settings

Plant menu: ▶ Toolbars ▶ Settings



Note: The arrow in the lower right corner of the button indicates a flyout for more options.

Setup

Command line: SETUP

Settings toolbar:

Please refer to Setup (see page 18) for more information regarding this command.

Specification

Command line: SETSPEC

Settings toolbar:

Please refer to setting the specification (see page 21) for more information regarding this command.

Size

Command line: SETSIZE

Settings toolbar:

Please refer to Setup (see page 19) for more information regarding this command.

Component to All

Command line: COMP2SIZE **(also** COMP2SPEC**,** COMP2SIZESPEC**,** COMP2LINE**, and** COMP2ALL**)**

Settings toolbar: **, , , , and**

Please refer to Set Component (see page 300) for more information regarding this command.

Current Settings

Command line: CURRENT

Settings toolbar:

Please refer to Current Settings (see page 300) for more information regarding this command.

Mode Convert

Command line: CONVERTDOUBLE **(also** CONVERTSINGLE**,** CONVERTSOLID**, and** CONVERTISO)

Settings toolbar: **,** **,** **and**

Please refer to Mode Convert (see page 295) for more information regarding this command.

Socket Weld/Threaded

Settings toolbar: **and**

Please refer to threaded/socket weld modes (see page 18) for more information regarding this command.

CEdit

Command line: CEDIT **(also** XCEDIT, **and** GCEDIT**)**

Settings toolbar: **,** **, and**

Please refer to Component Edit (see page 303) for more information regarding this command.

Auto Connect

Command line: CONNECTTOG

Settings toolbar:

Please refer to Auto Connect (see page 366) for more information regarding this command.

Zoom Lock

Command line: ZOOMLOCK

Settings toolbar: 

Please refer to Zoom Lock (see page 370) for more information regarding this command.

Specification Option Toggle

Command line: SPECOPTION

Please refer to Specification option toggle (see page 274) for more information regarding this command.

Grouping

Command line: GROUPTOG

Settings toolbar: 

Please refer to Grouping (see page 368) for more information regarding this command.

Select Control

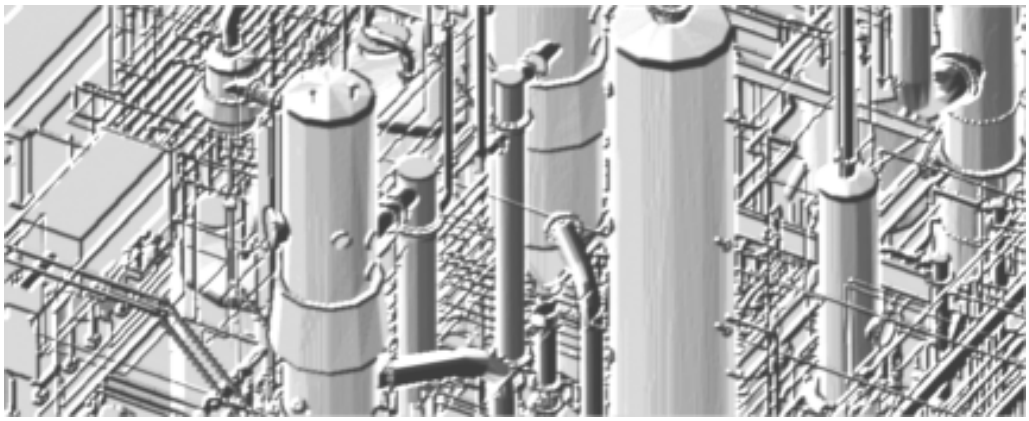
Command line: SELECTCONTROL

Settings toolbar: 

Please refer to Select Control (see page 368) for more information regarding this command.

CHAPTER 4

Accessory



In This Chapter

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Plan to Elevation

Plant menu: ▶ **Accessory** ▶ **North, South, East & West**

Command line: NORTHELEV**,** SOUTHELEV**,** EASTELEV **&** WESTELEV

This option allows automatic elevation creation from existing 2D plans. There are four different elevations to choose from: north, south, east, and west. Any objects selected out of these four planes or views will not convert in the elevation. The components in the plans should be drawn at their proper elevation for them to be placed correctly. There are no references to elevation within the components of this program. All components must be drawn at the desired elevation. If the plans are created with the UCS left at a 0' elevation, the elevation will have all components at the same elevation. The plans can be created at a 0' elevation and later the components can be elevated to their proper elevation by using the CHGELEV (see page 324) command from the pulldown *Pipe>Utility>Elevation>Change*.

Command: **NORTHELEV** (**SOUTHELEV**, **EASTELEV**, & **WESTELEV**)
Select objects: *pick point* Other corner: *pick other* 20048 found
Select objects: *enter to end*

Pick reference point: *pick a reference point to be used as base point later*

The next prompt provides an option allowing the component information (in the elevation) to be on or off. If the plan is being used for Bill of Material generation, the elevation material should not be repeated. Therefore, this option should be off. If the elevation is to be placed back in the plan, type on. If off, many automatic functions such as component labeling will not work.

Component information [ON/Off] <Off>: *type* **ON** *or enter for Off*

This function only draws graphic ends for all pipes that are open within the view or elevation.

Draw pipe end baseballs [Yes/No] <No>: *type* **Y** *(for yes) or enter for No*

This next option is useful for deleting the plan and replacing it with the ELEV2PLAN (see page 181) command. With this option a user can make an elevation, delete the plan, make modifications to the elevation, then place it back into the plan.

Delete selection for future replacement [Yes/No] <No>: *type* **Y** *(for yes) or enter for No*
Pick base point for elevation: *pick a base point for the elevation*

Elevation to Plan

Plant menu: ▶ **Accessory** ▶ **Elevation to Plan**

Command line: ELEV2PLAN

This option allows for the creation of plans from elevations. This function is a complement to the NORTHELEV, SOUTHELEV, EASTELEV and WESTELEV (see page 180) functions. This function is useful when an elevation has been changed and the user wishes to place it back into a plan.

Command: **ELEV2PLAN**
Select objects: *select objects*
Select objects: *hit enter*

Pick reference point: *pick point to be used by the lower base point location*
Component information [ON/Off] <Off>: *type* **ON** *or enter for Off*
Draw pipe end baseballs [Yes/No] <No>: *type* **Y** *or enter for No*
Delete selection for future replacement [Yes/No] <No>: *type* **Y** *or enter for No*
Pick base point and rotation: *pick base point, then rotation*

2D Representation

Plant menu: ▶ **Accessory** ▶ **2D Representation**

Command line: 2DREP

This command allows the user to create a flat 2D representation from a 3D model. In addition, it removes hidden lines and retains layer and color information. It can work both in model or paper space. In model space, the 2D representation is created as a block.

Note: An alternate command called **2DREPH** is also available. This command will export the hidden lines from solids to the 2D representation created. All other features of this command are similar to the existing 2DREP command.

Note: If running CADWorx Equipment, run the commands EQUIP2DREP or EQUIP2DREPH

When the command is run in model space, the user is prompted to select objects.

Command: **2DREP**
Select objects: *select objects*

If the command is run in paper space, the user can select one or more viewports or select objects from within a viewport.

Command: **2DREP**
Select [**Viewports/Objects**] <Viewports>:

Once the objects are selected, the user is prompted to indicate what they would like to include in the 2D representations.

Enter a filter option [**Solids/sOlids and centerlines/soLids and lines/Centerlines/cEnterlines and lines**] <sOlids and centerlines>:

Solids: This option generates 2D representations of a native AutoCAD 3D solids.

Centerlines: This option will copy CADWorx Piping and Steel center lines selected to the 2D representation.

Lines: This option will copy AutoCAD 3dPolyline, Arc, Circle, Ellipse, Leader, Line, Polyline, Spline and Helix selected to the 2D representation. This includes CADWorx Piping and Steel center lines.

After the option above is picked, the command will generate the 2D representation and insert it into model or paper space.

If the user is in paper space, the user will be prompted if they want the viewport(s) to be turned off.

Turn off Viewport(s) [**Yes/No**] <Yes>:

This option is useful when there are changes made to the model and the 2D representation needs to be updated. The user can turn the viewport back on using the AutoCAD properties dialog and and run the command again.

Note: This command only processes native AutoCAD Solid and Line entities that are typically created by CADWorx. This command will not process any custom solid or line entities that are created by other add on AutoCAD applications.

Auto Isometric

There are only a few simple steps to prepare automatic isometrics for use. The first of the requirements is the user border preparation. The second would be the actual configuration setup. After these two steps are finished, isometrics can easily be created by exporting the orthographics or 3D models to an export file and then importing the same file.

Border Preparation

As an example, the drawing _ANSI_BI.DWG (_ISO_A3I.DWG, metric) has been provided in the <INSTALL-DIR>\SUPPORT directory for use with the automatic isometrics within CADWorx Plant. The user may modify this drawing or start a new one.

Note: This drawing must be set up in Model space with nothing in Paperspace (no view ports created with MVIEW, etc).

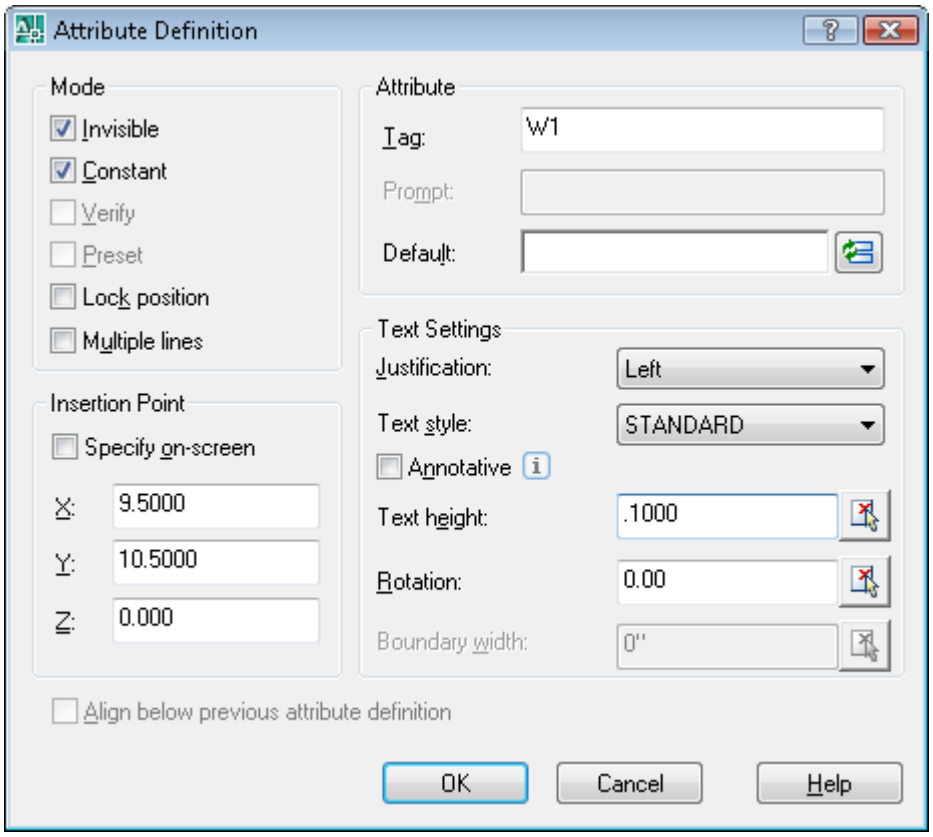
There are several attributes on this drawing that have a useful purpose within AutoISO. The attributes are as follows.

Attribute	Mode	Purpose
N1	Invisible, Constant	North Arrow location
W1,W2	Invisible, Constant	Mview window location
B1	Invisible, Constant	BOM heading location
User	Normal	Any user requirements

The N1attribute is used by AutoISO to determine the location of the north arrow for the isometric. If AutoISO does not find this invisible attribute, a prompt will appear inside of AutoISO asking for the insertion point.

The method for inserting the N1 attribute is provided by AutoCAD with the ATTDEF command as shown in the dialog below. Please consult the AutoCAD documentation for further instructions on using the ATTDEF command.

Command: **ATTDEF**



The W1 and W2 attributes are used by AutoISO to determine the location and size of the Mview which will be used by AutoISO. The piping configuration will be scaled to fit this window. Suggested locations are the lower left corner of the border drawing and the left side of the Bill of Material schedule location. See the sample _ANSI_BI.DWG (or _ISO_A3I.DWG, metric) in the <INSTALL-DIR>\SUPPORT directory of CADWorx Plant. Use the same procedure shown above for the insertion of the W1, W2, and B1 attribute.

If the automatic isometric program does not find these attributes, the program will pause and ask for the window location.

Mview...
First point: *pick lower left corner of window.*
Second point: *pick upper right corner of window.*

B1, the last attribute use by the automatic isometric program is the Bill of Material location. This is located in the upper right corner of the Bill of Material. The suggested location is the upper right corner of the border drawing. Use the procedure above for insertion of this attribute.

In the configuration for the automatic isometric program, options to tie constants within the drawing environment and the title block are available. For example, attribute DWG_REF could be a link to the drawing number that the isometric was created from. This will be explained in further detail in the configuration section. For these attributes, make sure all the *Mode* toggles are off. Under the *Tag:* enter DWG_REF. Under *Prompt:*, enter the desired prompt (ex: Drawing reference). Utilize the *Pick Point* button to locate the attribute within the border and enter any desired *Text Options*.

The last attributes that might be required are those necessary for the user requirements.

ISO Out

Plant menu: ▶ **Accessory** ▶ **Auto Isometric** ▶ **Iso out**

Command line: ISOOUT

The isometric out command supplies the standard file selection dialog prompt for a file name. This same file will be used later in the AUTOISO (see page 188) command. Once the file name is provided, the command prompts the user to review the isometric configuration (see page 189) file.

Note: The isometric configuration file is displayed only once during a drawing session.

Command: **ISOOUT**

The next prompt appears if the standard CADWorx Plant north arrow is not in the drawing. If company standards dictate another north arrow, then copy it to this drawing (N1.DWG in the <INSTALL-DIR>\SUPPORT directory).

Enter north direction <90>:

The selection process in the drawing environment is provided in three ways:

1. With the standard selection process in AutoCAD. This can include selection by crossing, window or any other available method.
2. Selecting one or more line numbers from a dialog or selecting a component with an existing line number.
3. Selecting an external database (see page 236) to create isometrics.

Enter an option **[Database/Line number/Select components]** <Select components>:

Database

This option allows the user to create isometrics or export components from an external database that can contain line numbers that span across different drawing files. The user is first asked to select the external database using a standard file open dialog.

Note: If a live database (see page 231) is used, the user will not be prompted to select the external database and the live database will be used for creating the isometrics or external files.

Without Line Numbers

If no line numbers or stop signs are detected in the database, the following prompt is displayed.

Database does not have any line number or stop signs...

If the database contained line numbers but no stop signs, the user will be prompted as to whether the stop signs should be used or not.

Use Stop Signs [**Yes/No**] <Yes>:

Yes

If YES is selected, continuity is checked and the ISOs are broken as a stop sign is encountered.

No

Ignores all stop signs and process all components checking for continuity.

With Line Numbers

If line numbers are found in the database, the following prompt is displayed:

Enter an option [**Line number/All components**] **<All components>**:

Line number

Selecting this option displays the common line list dialog (see page 647). One or more line numbers can be selected.

Generate files based on [**Continuity/Line number**] <Continuity>:

Continuity

This option checks the continuity of the line numbers selected. The function will combine line numbers that are connected to each other.

Line number

This option ignores continuity between the line numbers selected and will create separate files for each line number selected. Continuity within the same line number is checked. The file names will include the line number as part of it.

All components

All components will be processed without regard to line numbers.

If stop signs are detected, the user will be prompted as to whether the stop signs should be used or not.

Use Stop Signs [**Yes/No**] <Yes>:

Yes

If YES is selected, continuity is checked and ISOs are broken as a stop sign is encountered.

No

Ignores all stop signs and process all components checking for continuity.

When the function is completed, the number of files generated is shown.

Generated 12 AutoISO files:
E:\Drawings\COADE_001.pmi
to
E:\Drawings\COADE_012.pmi

Line Number

This option allows the user to select a component with an existing line number or lists all line numbers in the model.

Enter an option [**List/Select component**] <Select component>:

Once the line number(s) are selected by either one of the above methods, the program accesses all the components with matching line numbers.

List

See Appendix F: Common List Dialog (see page 647) for more information about this option.

Select component

This option allows the user to select a component with an existing line number. Select the desired component by crossing, window, or any other preferred method.

Select a CADWorx entity: *select any CADWorx entity*
Accessing entities with line number of "CS1-6"-305"

Once the entity is selected, it's line number will be displayed in the command line as shown above.

Select component

This option allows the user to select the desired components by crossing, window, or any other preferred method. The user can select components that are from an XREF if the UseObjectsFromXrefs (see page 67) is set to 1.

ISO In

Plant menu: ▶ **Accessory** ▶ **Auto Isometric** ▶ **Iso In**

Command line: AUTOISO

The AUTOISO process initializes by selecting this option. After the prompt, the standard file selection dialog will request a file that initializes the AUTOISO process.

Command: **AUTOISO**
Drawing preparation...OK

Select the previously exported isometric file. If the presently opened drawing is not new, a prompt will appear indicating that the isometric must be started with a new drawing. The components will then be drawn into the model world of the isometric drawing.

Reading and listing component number: 31...finished

After drawing a skeleton each component in its' new scaled position, the user will be asked to select a view.

Select desired view...
*** Southeast view shown ***
Enter an option [eXit/southEast/Northeast/northWest/Southwest] <next view>: *enter for the next view*
*** Southwest view shown ***
Enter an option [eXit/southEast/Northeast/northWest/Southwest] <next view>: **X** *to finish selection*
Enter an option [**Pan/+/-/Accept**] <Accept>: *enter*

Pan

Displacement: *pick a point*
Second point: *pick a second point*

This function responds identically to the PAN command used by AutoCAD.

+ (or =)

This simply increases the size of the model within the view port created with the MVIEW command.

-

This simply decreases the size of the model within the view port created with the MVIEW command.

Accept

After the view is selected, there will be no other prompts unless the border drawing was not set up according to the guidelines outlined earlier. If manual placement of BOM marks are requested, the user will also have to manually place the bubbles. Refer to the BOM documentation for more about manual material marks.

Converting components to Isometric single line...finished
Auto Dimensioning...finished
Auto Bolts...finished

If Bill of Material is requested in the isometric configuration file, it is generated as well.

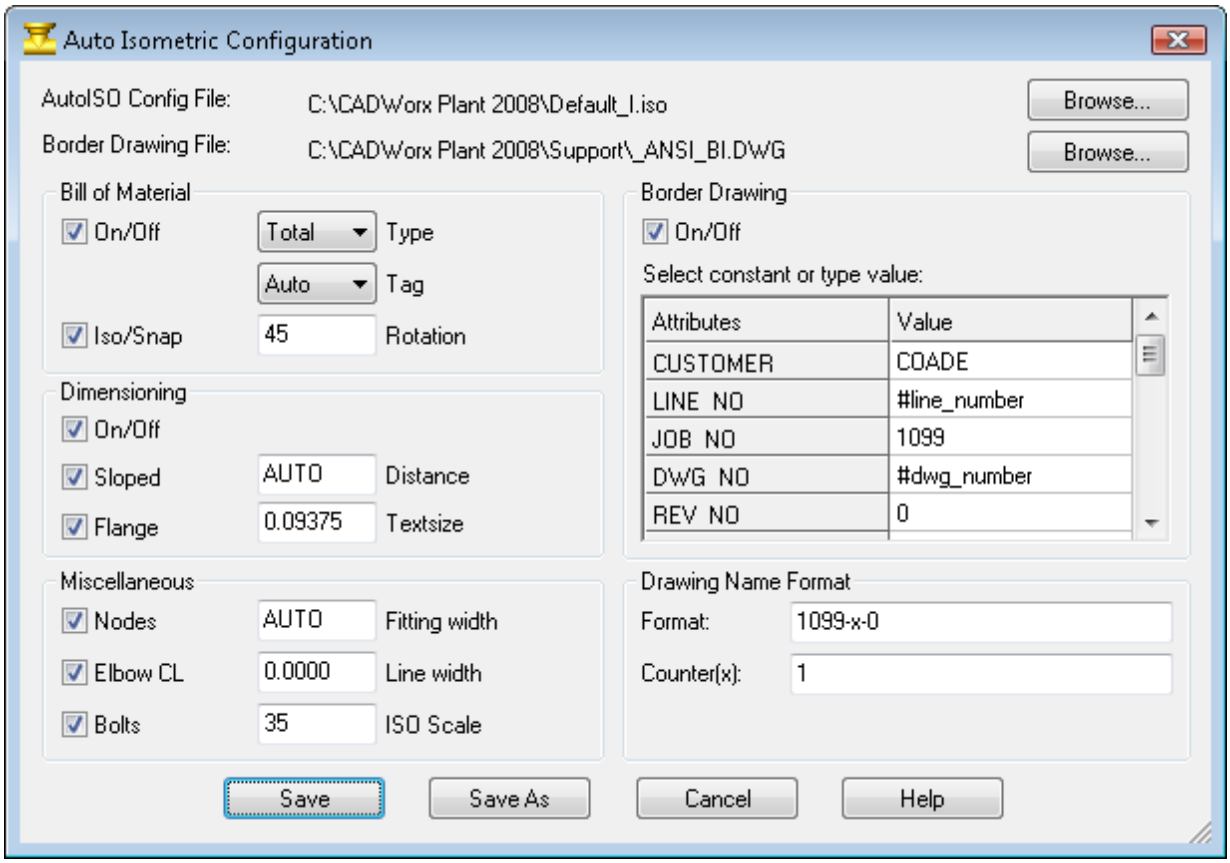
Generating Bill of Material...finished
Paperspace zoom lock toggled on...

ISO Config

Plant menu: ➤ **Accessory** ➤ **Auto Isometric** ➤ **Config**

Command line: ISOCFG

The first time the isometric configuration is run, a default file will be created. This file will be named DEFAULT_I.ISO for imperial or DEFAULT_M.ISO for metric systems. This configuration file can be renamed and relocated to any directory desired (prior to use). The most logical placement might be the same directory as the isometric drawing files.



Border Drawing File

This browse button provides selection of the border drawing to be associated with the configuration file. This will be saved to the iso file listed in the configuration box. As mentioned earlier, this border can have preset attributes that will automate the iso sequence even further.

Bill of Material - ISO Config

The On/Off check box will indicate whether Bill of Material generation will or will not be provided. If the BOM is turned off, the Type, Tag, Rotation list boxes and ISO/Snap check box will be disabled.

The *Iso/Snap* toggle is useful when the manual button is used above. This will set the crosshairs in an isoplane and the snap will be set on. Placement of bubbles and leaders can be arranged in an exact parallel location to the components in the isometric. Check for one.

The *Type* list box offers three types of bill of materials. Cut lengths are used when individual cut lengths for pipe are required in the Bill of Material. Select Total for pipe cuts to be combined into totals. Single is for placing each individual component into a single row in the BOM schedule.

The *Tag* list box offers three choices providing automatic, manual or no material tags. Automatic will place the tags in the angle direction as listed below in the Tag rotation edit box. The user will have to rearrange the tags as required. This can be assisted with the Tag location option in the pulldown menus. Manual placement will stop each time a component needs to be tagged and ask the user for the bubble location of the tag. The leader from the bubble to the component will automatically be drawn when the bubble location is selected. Off provides no material tags at all.

The *Rotation* edit box is used to provide an angle for the leader from the component to the bubble. This angle is used only when the automatic button is selected above.

Dimensioning

The *On-Off* buttons in this box determine whether dimensioning in the isometric will or will not be provided.

The *Sloped* toggle places dimensions in the vertical legs of sloped systems regardless of the DimensionsExcludedSlope (see page 56) variable setting in the startup configuration file.

The *Flange* option allows the dimensioning routine to dimension to and from the face of the flanges. Flange faces on the ends will automatically be dimensioned.

The *Distance* edit box allows control of the dimension line offset from the components that are being dimensioned. This is a true distance in the model of the isometric. For example, the dimensioning on a 12"-150# flanged system would require this factor to be greater than 9.5 since the radius of this flange is 9.5". The default is 25, which will work in most cases. By typing “AUTO” in this box, the program will automatically base this distance on the size of the text within the dimension. This is the preferred method and default.

The *Textsize* edit box shows the present plotted or paper space textsize. This size should be slightly larger than the users normal textsize, because dimensioning placed in the model environment is at a slanted angle (approximately 30 degrees).

Miscellaneous

The *Nodes* toggle can be selected to indicate whether nodes placed on the “SYSTEM” layer will or will not be provided.

Elbow CL can be toggled to provide square centerlines on the elbows in the isometric.

Bolts can be toggled on for the addition of bolts. If bolts were added in the orthographic drawing, then they will not be required here. If this occurs, there will be duplicates in the drawing.

Fitting width is used to adjust the width (hub od or flange od) of various components. Large flanges can be adjusted to where their od/length ratio can better represent what would be desired in a true isometric. Socket weld fittings can be decreased to 70% in order for their hub not to overlap. By typing “AUTO” in this box, the program uses the values listed in the ISORATIO.TBL in the <INSTALL-DIR>\SYSTEM sub-directory. These values can be changed to any user preferences.

The Line width edit box provides adjustment for the plotted line width of the components. Normally, setting this width to 0 and using the Layers command to assign a width is best. Line width can then be set for individual color or layers in the plotting sequence.

The ISO scale can be used to adjust the scaled size of the isometric. In all isometrics the actual pipe component (not elbows, flanges, etc.) is scale. The original length will be preserved in the component for the Bill of Material generation. When a pipe and a valve occupy the same location along an x axis, the pipe will not be scaled in the way of the valve. Any overlap to each side of the valve can then be scaled. This factor determines what the scale factor should be. Setting this option to 100% will not scale any portion of the isometric. A suggested factor of 35% seems to work in all general cases. One slope pipe will act as a rigid component - it will not be scaled in the X, Y, and Z of the sloped pipe. Multiple sloped piping will turn the scaling algorithm off.

Border Drawing

This option will place the selected border (from the border button above) in the isometric drawing or will leave it out. When the *On/Off* option is off, the entire right side of the dialog will be disabled which disallows selection on this side.

Attributes and Values

The attributes section provides a list of attributes that were in the drawing selected above from the border drawing button. When selecting a new drawing for the first time, this list will provide the attribute list on the left side and “na” on the right of the list box. After selecting one item in the list the Constants box below will become available and allow selection.

Note: Constant and Preset attribute definitions will be ignored.

After selecting one item in the list box above, the value can be changed and then replaced with any user requirement. The constants list will provide options that can be searched for during the AUTOISO process. For example, the “#name” constant will search for the AutoCAD login name and place that in the attribute specified in the attributes section above. The “#line_size”, when associated with an attribute in the drawing, will search through the components and use the first line-number found. All constants will be searched for that are specified, but not all will be found. Listed below are the constant names and search procedures.

#line_number	searches components
#dwg_number	name specified below in the name box
#name	AutoCAD setvar LOGINNAME
#date	AutoCAD setvar CDATE
#dwg_reference	AutoCAD setvar DWGNAME from source dwg
#line_size	searches components
#pipe_material	searches pipe components, matches “-”
#fit_material	searches fitting components, matches “-”
#forg_material	searches forged components, matches “-”
#specification	searches components
#schedule	searches components, matches directory
#rating	searches components, matches directory

Name

Drawing Name Format edit box provides a method of naming the present isometric drawing and then other consecutive drawings will be incremented by a value of 1. The name supplied in the box will need an “x” in the description any place. The counter edit box will need a value in the “x” variable for the name. As isometrics are automatically generated the counter (x) edit box will be incremented by 1.

2D ISO

Command line: 2DISO

This command converts a CADWorx Plant Modelspace/Paperspace isometric drawing to a 2D flat drawing.

Note: This command requires AutoCAD Express Tools to be installed on the user's machine.

Once the conversion is completed, all CADWorx Plant dimensions and the bill of material is moved to Model Space. The components no longer retain any CADWorx Plant intelligence once this command is run.

```
Command: 2DISO

Creating flat isometric drawing...
467 object(s) changed from MODEL space to PAPER space.
Make model space drawing [Yes/No] <Yes>:
Regenerating 546 modified entities.
```

CAESAR II

The following functions provide a seamless bi-directional link to CAESAR II (pipe stress analysis). By selecting components in the model environment of the drawing, a CAESAR II input file can be created. After the stress analysis has been performed, the data can be transferred back into the drawing environment. Components that are modified in CAESAR II will replace or update the current components in the drawing. Components that are unchanged will not be disturbed. All CAESAR II information will then be stored in the drawing providing future analysis capabilities based on the previous information (node numbers, temperature, pressure, etc.). This capability can also be used for the creation of a stress isometric that will be explained below.

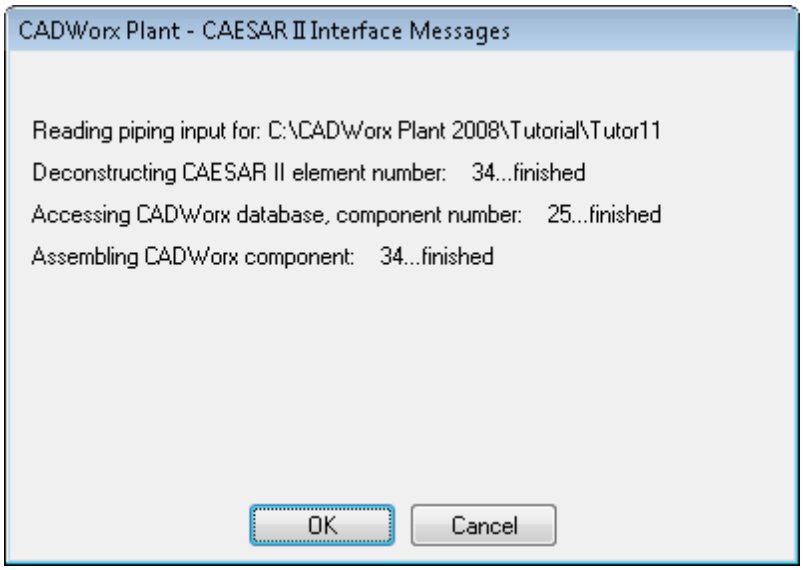
System In

Plant menu: Accessory ▶ CAESAR II ▶ System In

Command line: C2IN

Input requires the selection of the necessary “*._A” or "*.C2" file. This is the input file created by CAESAR II or the output facility in CADWorx Plant. By selecting Plant>Accessory>CAESAR II>System In from the pulldown menu, the user will be presented with the standard file selection dialog. In this dialog, select the desired “*._A” or "*.C2" file. Selecting any other file will result in an error message. After selecting the “*._A” or "*.C2" file, a dialog indicating that the file was read will appear.

Note: Make sure that the CAESARVersion (see page 55) startup variable is set to match your CAESAR II installation. If this variable is set to 3.2 and the C2IN command is issued on a file created or modified with CAESAR II 4.40, then an error will occur indicating that the valid file is corrupt.



The next prompt requires that a specification be set. Although, there might be an existing specification set, confirmation or new specification will be required. The specification might not be used. The input facility in CADWorx Plant will attempt to determine the proper specification for the component being input from data in the the CAESAR II input file. The input facility or the data in the input file will not always indicate a specification, therefore a default specification needs to be set for these components.

Please set a default Specification <150>: *enter or type desired specification*
Prepared <INSTALL-DIR>\SPEC\150.spc specification...

After the specification has been set, the last prompt before data is actually transferred requires the user to determine how the components should be populated.

New components populated by [**Caesar**/**Environment**] <Environment>:

Caesar

The descriptive information within any new components will be derived from the material passed back from CAESAR II and the name of the purposed component located within the PRGM.TBL which is in the <INSTALL-DIR>\System directory. The system will automatically assign all the required paths for data files which might be use later for reconstruction (i.e. mode converting). If the material was not supplied in CAESAR II, then this option would use the description from the specification but would not populate the line number.

Environment

This option assigns the component information based on the default specification chosen above. It also assigns a line number if there is a current line numbering system setup. The user is responsible for the correct line number being set prior to importing. This options overrides any material sent from CAESAR II. For example, if A-133-B material type was chosen in CAESAR II, and the default specification above dictated A-106-B as the material type, this option would override the A-333-B grade and place A-106-B in the component. It's full description would be from the default specification set above.

System Out

Plant menu: ▶ **Accessory** ▶ **CAESAR II** ▶ **System Out**

Command line: C2OUT

This option creates a CAESAR II input file. This input file can then be used in CAESAR II without any modification. Components selected in the drawing will convert to the appropriate elements for the stress model. For example, a tee connected to pipe in all directions will be represented with one node at the intersection instead of four (three at the ends and one at the intersection). This provides the stress analyst a model that is ready for use. All pertinent information will also be transferred (weights, wall thickness, etc.).

The selection process in the drawing environment is available in two ways. The first requires selecting the components with the standard selection processes in AutoCAD. This can include selection by crossing, window or any other preferred method. The second selection process allows the selection by line number or selecting a component with an existing line number. The database option shown is not available with this command. The following prompts outline both methods.

Command: **C2OUT**
Enter an option **[Database/Line number/Select components]** <Select components>:

Database

This option is not available at this time.

Line Number

This option allows the user to select a component with an existing line number or lists all line numbers in the model.

Enter an option [**List/Select component**] <Select component>:

Once the line number(s) are selected by either one of the above methods, the program accesses all the components with matching line numbers.

List

See Appendix F: Common List Dialog (see page 647) for more information about this option.

Select component

This option allows the user to select a component with an existing line number. Select the desired component by crossing, window, or any other preferred method.

Select a CADWorx entity: *select any CADWorx entity*
Accessing entities with line number of "CS1-6"-305"

Once the entity is selected, it's line number will be displayed in the command line as shown above.

Select component

This option allows the user to select the desired components by crossing, window or any other preferred method.

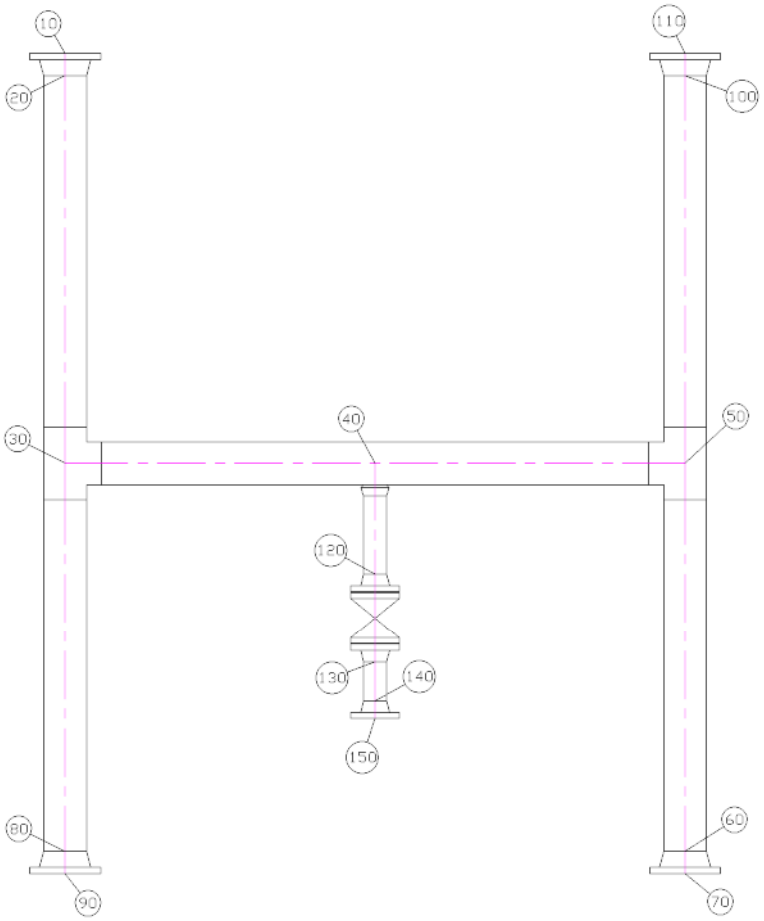
The next prompt allows the user to select the node number system to be sent to the CAESAR II input file. This prompt allows the user to specify the starting location and then the next node number direction.

Provide start locations [Yes/No] <No>: *type* **Y** *for yes or enter for no*

Enter to finish...

Pick point: *pick a point using an OSnap*
Pick direction: *pick anywhere using the OSnap NEArest to*
Pick point: ...
Pick direction: ...
Pick point: *enter to finish*

To accomplish the following example, the user would pick the starting point at node 10. Picking anywhere along the centerline toward node 90 would be adequate for a direction. The next point and direction would be picked at the intersection at 30 and then picking a direction toward 50. To continue along the same direction, a pick at 40 and a direction toward 50 would be required. The next point would be the intersection at 50 and would require a direction toward 70. The user could let CAESAR II automatically node the rest of the model or the user could finish by manually selecting the next point at 30 and then picking a direction toward 90. The next point and direction would be 50 toward 110 and the last point and direction required would be 40 with a direction toward 150. Make sure Ortho is off and use the OSnap NEArest to to pick along centerlines for directions.



After the selection process, a prompt appears indicating the status of the output from the drawing environment.

Writing component number:

A final dialog indicates that a CAESAR II input file has been created.

Stress ISO

Plant menu: ▶ **Accessory** ▶ **CAESAR II** ▶ **Stress Iso**

Command line: C2ISO

This option is designed to provide the stress analyst with the capabilities of creating stress isometric drawings. Importing the "*._A" or "*.C2" file is similar to the C2IN (see page 194). After importing the "*._A" or "*.C2" file, dialogs and questions similar to the automatic isometric will be presented. The dialogs provide options to adjust or control dimensioning, textsize, etc. The selection of a border drawing and a dialog requesting attribute values will be presented.

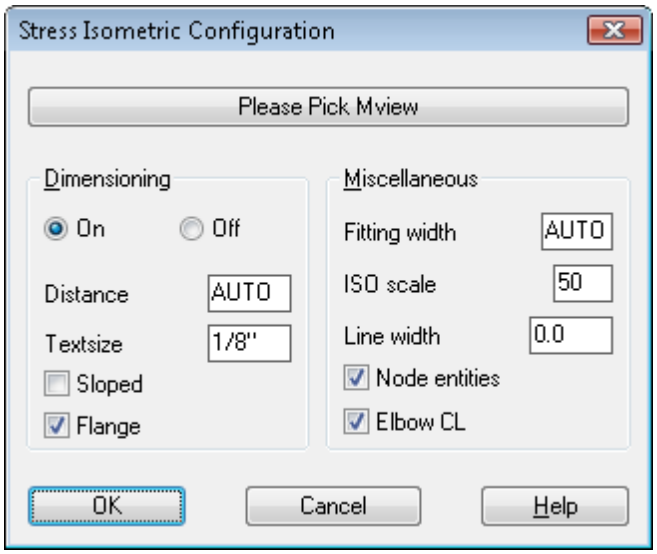
Similar to AUTOISO (see page 183), the isometric will be drawn in the model environment of AutoCAD (tilemode = 1). All annotations (except dimensioning) will be placed in the paper space (tilemode = 0). This option must be used with a new drawing. Upon initialization the following prompt will appear.

```
CAESAR II stress ISO preparation...finished
Drawing preparation...OK
```

As with the C2IN option above, the next prompt indicates the selection of the input file. The standard selection dialog will be presented for this purpose.

The next step would require the selection of a border drawing from the standard selection dialog. The option to choose any border is available. The border drawings supplied with CADWorx Plant have attributes attached that will present another dialog asking for values to be supplied. The values can be supplied at this time or at a later date by using the DDATTE command supplied with AutoCAD. Selecting OK at the bottom of the attribute dialog will continue the program execution.

The last dialog presents different options for the drawing appearance. It can control various options such as textsize, fitting width, isometric scale, and many others. These options are explained below.



After selecting OK from this dialog, the drawing will generate automatically. The following prompts will indicate the progress.

Reading and listing component number: 31...finished
Working component number: 31...finished

After drawing each component in its' new scaled position, the user will be asked to select a view.

Select desired view...
*** Southeast view shown ***

Enter an option [eXit/southEast/Northeast/northWest/Southwest] <next view>: *pick a view and then type X to exit*
Enter an option [Pan/+/-/Accept] <Accept>: *enter to accept*
Converting components to Isometric single line...finished
Auto Dimensioning...finished

The next prompt requires the location of a Stress ISO arrow. This step is very important. It stores the information about the Mview scaling within the Stress Iso arrow. When applying annotation, the program gets information about placing the text. If this Stress Iso arrow is not on the drawing, the annotation will be placed at 100 percent.

Pick stress ISO arrow location:
Paperspace zoom lock toggled on...

Paperspace zoom lock is applied and this completes the stress isometric. Any other modifications required can be performed at this time.

Please Pick Mview

This button is mandatory. The Mview is the opening in the paper space that shows the components in the model environment (Mspace). The components will be presented and pre-viewed in this view. Suggested locations are in the lower left and upper right of the border drawing. If this button is not activated and an Mview is not selected, an error dialog will appear and not allow an exit from this dialog. By picking Cancel at the bottom of any dialog, the current process will be aborted. For further information regarding the AutoCAD Mview, please refer to the applicable AutoCAD manuals.

Dimensioning

The *On/Off* button allows dimensioning to be turned on or off. The dimensioning, as in AUTOISO, will be entirely automatic.

The *Distance* option determines the offset distance of the dimension lines from the component. All dimension lines will be placed uniformly from the components at this distance. By typing “AUTO” in this box will force the program to automatically base this distance on the size of the text within the dimension. This is the preferred method.

Textsize controls the actual size of the dimension text that will be in the model space. This size will also be controlled by the actual size of the Mview. For example; dimension textsize could be 2’-4” in the model, but would plot at the textsize set here. A factor is also added to the dimension text to compensate for the dimension text being placed at an angle in the isometric view. For further discussion regarding dimension placement in an Mview, please refer to the AutoCAD manuals.

The *Sloped* toggle places dimensions in the vertical legs of sloped systems regardless of the DimensionExcludedSlope (see page 56) variable setting in the startup sequence.

Set the *Flange* toggle to on if dimensioning to flange faces is desired. Off will by-pass all flange faces with the exception of flange faces at starting or ending points of runs.

Miscellaneous

The *Fitting width* option controls the width of flanged, socket weld and threaded components. One good use of this control is to set socket weld components to 70 percent. This will provide a good representation by not having overlapping hubs in a 90 degree component. Large flanges can be reduced to show a more pleasing length to O.D. ratio. By typing “AUTO” in this box will force the program to use the values listed in the ISORATIO.TBL in the <INSTALL-DIR>\SYSTEM directory. These values can be changed to any user preferences.

The *ISO scale* edit box controls the scaled appearance of the isometric. All components in the drawing are not scaled. Piping in general is the only component which is scaled. This controls how much the pipe is scaled and will affect the over all appearance of the ISO. A 100 percent scale factor will draw the isometric full size (no reduction).

Line width controls the plotted width of the components. The line width of components is similar to the textsize above. In the model space line width will be much greater than in the paper space. The end result will provide the desired line width for plotting.

The *Node entities* toggle controls placement of nodes at all components ends. This option does not have anything to do with the node descriptive annotation used by CAESAR II.

Elbow CL (centerline) toggle provides square elbow centerlines along with the radius centerline.

Multiple ISO

Plant menu: Accessory ▶ CAESAR II ▶ Multiple Iso

Command line: C2MULTI

This option provides a method for breaking large CAESAR II models into multiple stress isometric drawings. Selecting this option imports a CAESAR II input file, draws a skeleton of the model and displays the dialog below. This dialog provides all the functions required to create multiple stress isometrics from one CAESAR II input file. Similar to the C2IN (see page 194) and the C2ISO (see page 199) functions, the selection of a CAESAR II "*._A" or "*.C2" file is required. This function requires that a new drawing be used. The program asks for a default specification and provides an input intervention level. Please refer to the C2IN for further information regarding the intervention level and importing a stress isometric.

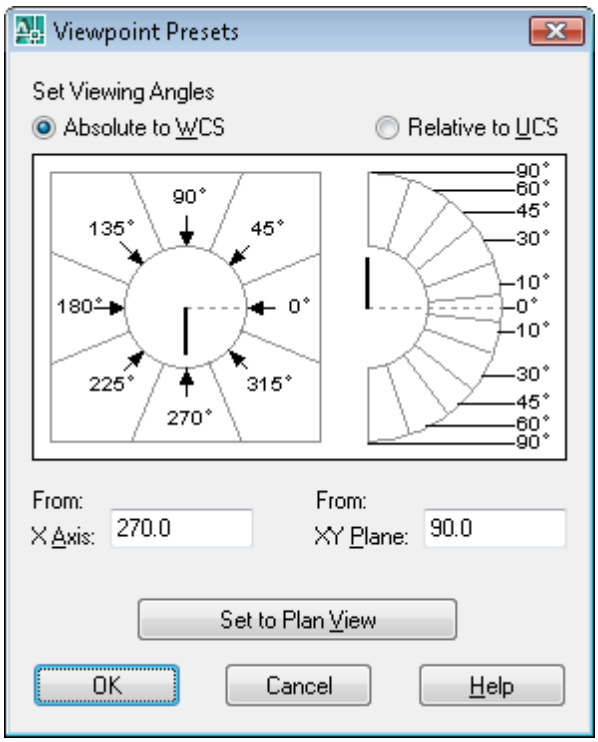
The steps to obtain multiple isometric drawings are outlined below.

1. Import a CAESAR II model by selecting the C2MULTI command from the menu. This creates a skeleton of the complete CAESAR II input file and displays the dialog shown below. The model will not be scaled (scaling will be available in the individual isometrics created by the this function).
2. The dialog shown below can be used to select different components for each individual isometric group. Group names can be created (*List Add* button) or deleted (*List Delete* button) as desired. Any name can be given to a isometric group. After creating a group, components should be selected for inclusion in it using the *Select* button.
3. Each group can have any number of components associated with it. This will be displayed at the very top of the dialog as the components are selected. Components can also be associated with multiple groups. An elbow on a corner could be in the ISO-01 group and in a ISO-02 group (the color of the component will be that of the last selected group).
4. Each group can be given a different color to allow easy viewing of the different groups created (*Color* button under the selection section). Long pieces of pipe can be broken for use in multiple isometrics (*Break* button under the selection section).
5. After all desired groups are created and components are selected for each, the different groups can be written to individual isometrics. This is achieved with the *Write Drawing File* button. A label below this button will indicated that the file has or has not been written.
6. After writing the different isometric drawing files, the complete CAESAR II skeleton file (current drawing) can be saved with any name (utilizing the standard AutoCAD save command). Returning to this model at any later date is permitted and C2MULTI can be used again and again on the same skeleton model in this saved drawing file. Re-importing the CAESAR II input file is not required; modifying or adding more isometric groups from this drawing file can be accomplished with the same skeleton drawing file.

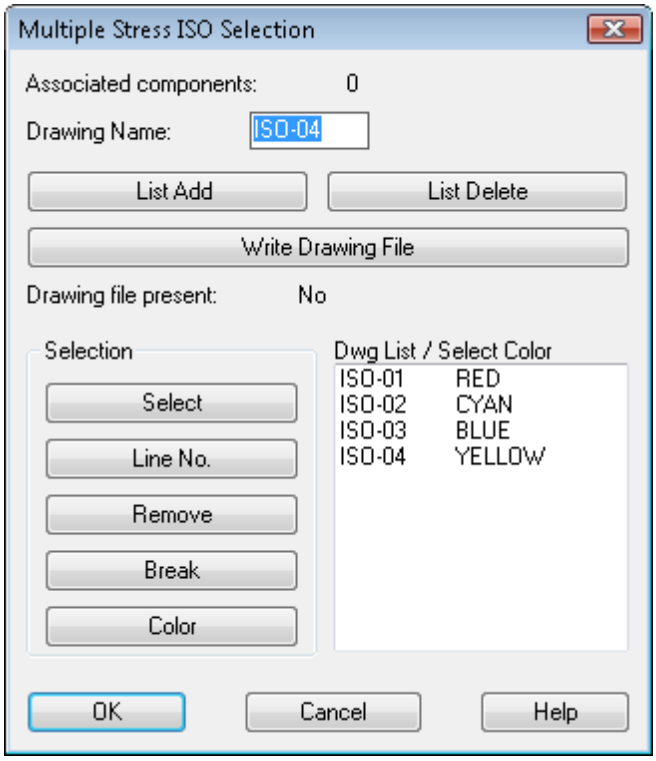
7. Opening each individual isometric drawing file created with the C2MULTI (for example ISO-01.DWG, ISO-02.DWG) and re-running C2MULTI is all that is required to start the automatic sequence that will generate the desired isometric. From here on, all commands and options proceed exactly like those for C2ISO (Do not run C2ISO, it will not generate the individual isometric - C2MULTI must be used to generate these special isometrics).

   Command: **C2MULTI**

After executing the command, a dialog will appear requesting selection of the "*._A" or "*.C2" file. After this file is chosen, and the skeleton is imported, a DDVpoint dialog appears requesting a view to be selected. Refer to the AutoCAD documentation for more information on this command and dialog. Generally, 315 degrees from the 0 degrees (east) and 30 degree up from the xy plane (grade) will provide a good view. This dialog is shown next.



After this, the C2MULTI dialog allows the selection of the different groups. This dialog can be ended and returned to as required.



Associated Components

This is the number of components associated with the current group that were selected with the *Select* button. It maintains an exact count of components associated with each individual group listed in the *Drawing Name* edit box.

Drawing Name

This edit box shows the name of each isometric group. These groups can then be added, deleted, or selected from the list box below. If the name is in this box, it is the current group being modified.

List Add

The name in the edit box can be added to the list box below with this button.

List Delete

This deletes the name in the edit box from the list below.

Write Drawing File

This creates a drawing file with the same name listed in the edit box (or highlighted in the list box below). All components associated with the current group in the edit box are written to a drawing file.

Drawing File Present

This label indicates whether there is a current drawing file written with the listed name. All drawings are written to the local or startup directory.

Drawing List/Select Color

This list box supplies names and colors of all current created groups.

Select

This button hides the dialog and allows selection of components to be added to the current group. As more components are selected, the number of *Associated components*: increases.

Line No.

This button provides a method of selecting components by typing in a line number. The type in value can be part of a line number. For example, CWS would select 4"-CWS-400.

Remove

This button hides the dialog and allows the removal of components from the current selected group. As more components are removed, the number of *Associated components*: decreases.

Break

This button provides a method of breaking a pipe component (pipe only), allowing it to be included in multiple groups.

Color

This button shows the standard color dialog for color coding the groups. This provides a visual difference within the master skeleton drawing file.

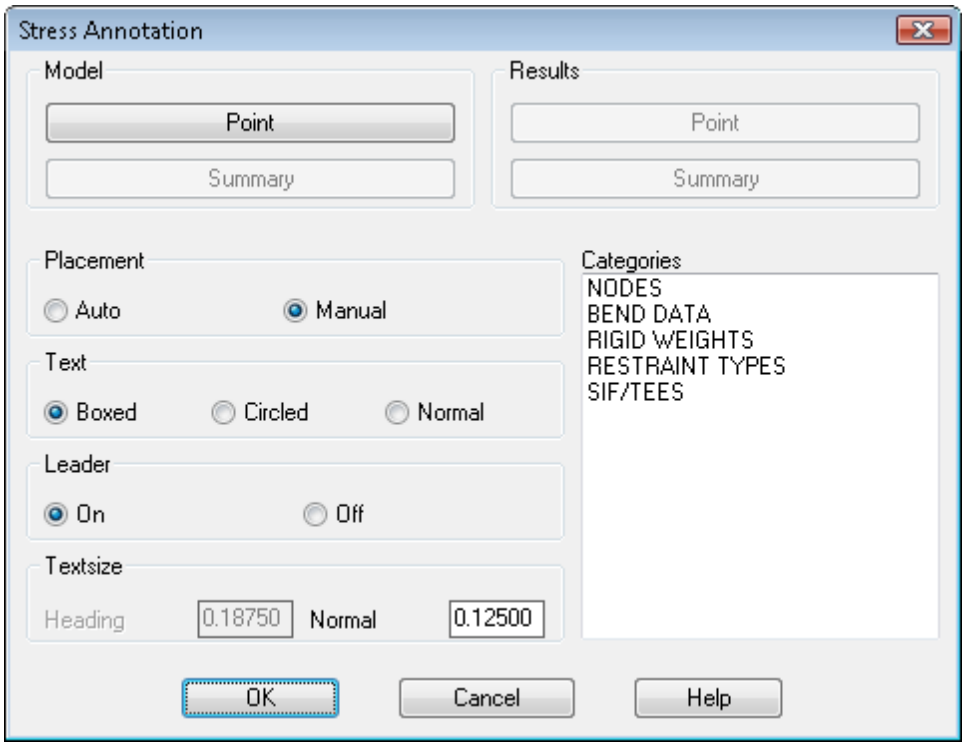
Annotation

Plant menu: ▶ **Accessory** ▶ **CAESAR II** ▶ **Annotation**

Command line: C2TEXT

This option provides utilities for model and result summary text placement. When a stress analysis is performed, CAESAR II generates different output files where this information can be extracted. The static output file ("_P" file) will be required to utilize this utility. Running statics inside of CAESAR II can generate this. The spectrum ("_S" file), time history ("_T" file) and the harmonic ("_Q file) output files are not available for text placement summaries at this time.

By selecting this option, the standard file selection dialog will display. Selection of the desired "*._A" or "*.C2" file will be required. Any other selection will generate an error. Accepting the desired file will generate a prompt indicating that the file is being indexed. After indexing the following dialog will appear.



After selecting the desired category and the desired options, the first question requiring response indicates whether the AutoCAD Monotext text style is desired. This style is very useful in summary cases (characters are equally spaced, thus, aligning in a vertical fashion).

Monotext style <Yes>: **N** *for no or enter for yes*

As indicated above the *Summary* cases will prompt for a location only.

Pick left justified summary location: *pick point*

The *Point* list will have some other options at the command line. These prompts will repeat for each item chosen in the point case.

Pick point: *pick point*
[**Skip/Ok**] <Ok>:

Skip

This option will skip (erase) the currently placed item.

Ok

This option will leave the currently placed item and proceed as normal.

Model

The *Point* button will list summaries of various categories from the model analysis. This list will be displayed in the *Categories* list box in this dialog. Each point will be prompted for placement as explained below.

The *Summary* button is not used at this time. It will be disabled. Future use will provide general summaries from the model analysis.

Results

The *Point* button will list summaries of various categories from the results analysis. This list will be displayed in the Categories list box in this dialog. Each point will be prompted for placement as explained below.

The *Summary* button will list the general summaries of various categories from the results analysis. This list will be displayed in the *Categories* list box in this dialog. One location pick will be required.

Placement

These buttons provide either manual or automatic placement of the text. *Auto* (automatic) will place the text (possibly in boxes or circles if the next option has been chosen) on the drawing at the designated 3 dimensional point. *Manual* allows each text group to be placed individually. A ghost of the text will be display for placement guidance. If summary is selected from either the *Model* or *Results* boxes, this boxed group will be grayed out and rendered inoperable.

Text

Three buttons provide either boxed text, circled text, or just text alone. This option will grey out when the *Summary* button is selected.

Leader

These buttons turn leaders on or off. The leader will be drawn from the corner of the boxes. The program will automatically determine which corner (based on the closest) to draw it from. This option will grey out when the *Summary* button is selected.

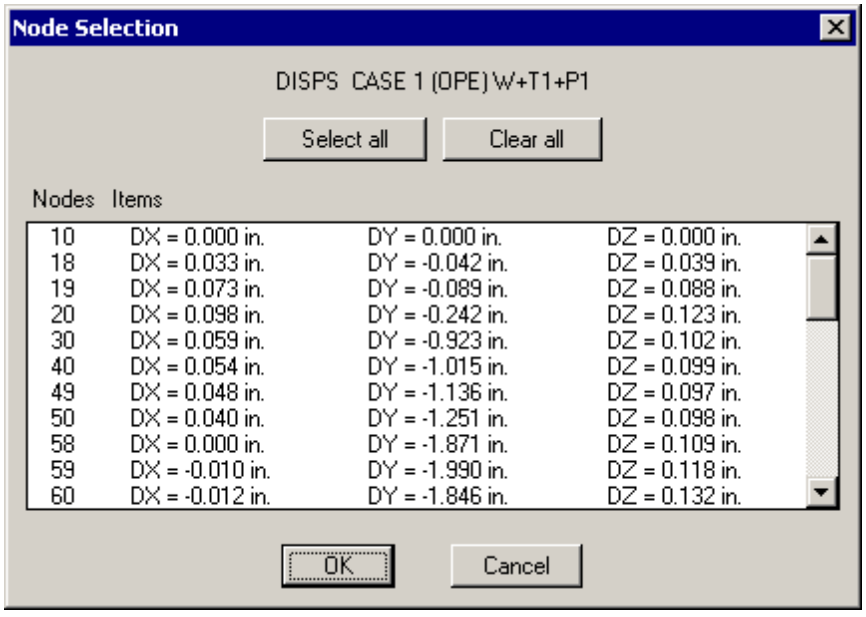
Textsize

This will set the size of the text in the paper space environment of the drawing. The *Heading* size will be operable for summaries only.

Categories

This is a list of all cases that were generated by CAESAR II. As the *Point* or *Summary* buttons are selected this list box will provide the cases in the particular category. Selecting one of the cases while the *Summary* button has been chosen will close the dialog and prompt the user for a location point.

When viewing a point list, double clicking on a case, causes the following dialog to display. This is a list of all cases at each point in the file.



Select all will highlight every item associated with the particular case. Selection can be done by individual picking also. Multiple items can also be selected by using the CTRL-Key and Shift-Key combinations.

Clear all will eliminate highlighting of every item in the particular case.

This dialog will be labeled with the case at the top. Each item in this list is associated with a point in the drawing. Hitting the *Cancel* button allows returning to the previous dialog and selecting another case if desired.

ISOGEN

CEDIT ISOGEN Information

ISOGEN Project Manager / I-Configure

ISOGEN is an automatic isometric generation program that can be used with CADWorx Plant. CADWorx Plant exports the necessary information and then runs ISOGEN to automatically generate isometrics. ISOGEN is the industry standard for creating isometrics. Normally there is not any clean up required for these isometrics.

ISOGEN can be an alternative to CADWorx Plant's own automatic isometric generation (see page 183) program. Both methods of automatic generation can be maintained and run on the same machine.

Warning: CADWorx Plant Professional must be authorized on the hardware lock to run ISOGEN. When a hardware lock has been authorized to run ISOGEN, only that workstation will be able to generate ISOGEN isometrics. The models created with un-authorized workstations will not be able to export to ISOGEN even if the model is later opened on a workstation that is authorized to run ISOGEN. Each component generated with an ISOGEN authorized workstation lock is marked. Any machine not authorized will not be able to generate a model for export or generate the ISOGEN isometric. To avoid ISOGEN authorization issues, COADE recommends purchasing CADWorx Plant Professional for all workstations if you are planning on using ISOGEN for isometric generation.

The ISOGEN programs are all very well documented. Each of the programs included with ISOGEN are supplied with an Adobe Acrobat Portable Document Format (PDF) file. This PDF file can be printed and/or viewed from the computer. Some PDFs can be accessed from the program group that is created during installation. We suggest reading the PDF manual prior to contacting for technical support.

ISOGEN is installed in a sub-directory of CADWorx Plant. There is not an option of installing it anywhere else. The program group created during installation provides shortcuts to all the programs within these directories. It also provides shortcuts for all the documentation as well.

The information supplied in the CADWorx help file is for general use only.

Under the ISOGEN directory, there are several sub-directories.

The Isogen_Util directory contains all the editors that are used to control the format of the automatically generated isometrics. These utilities include the options editor, symbols editor, and others. These programs are required to control the appearance of the ISOGEN isometric. As mentioned above, all the programs have shortcuts from the programs group.

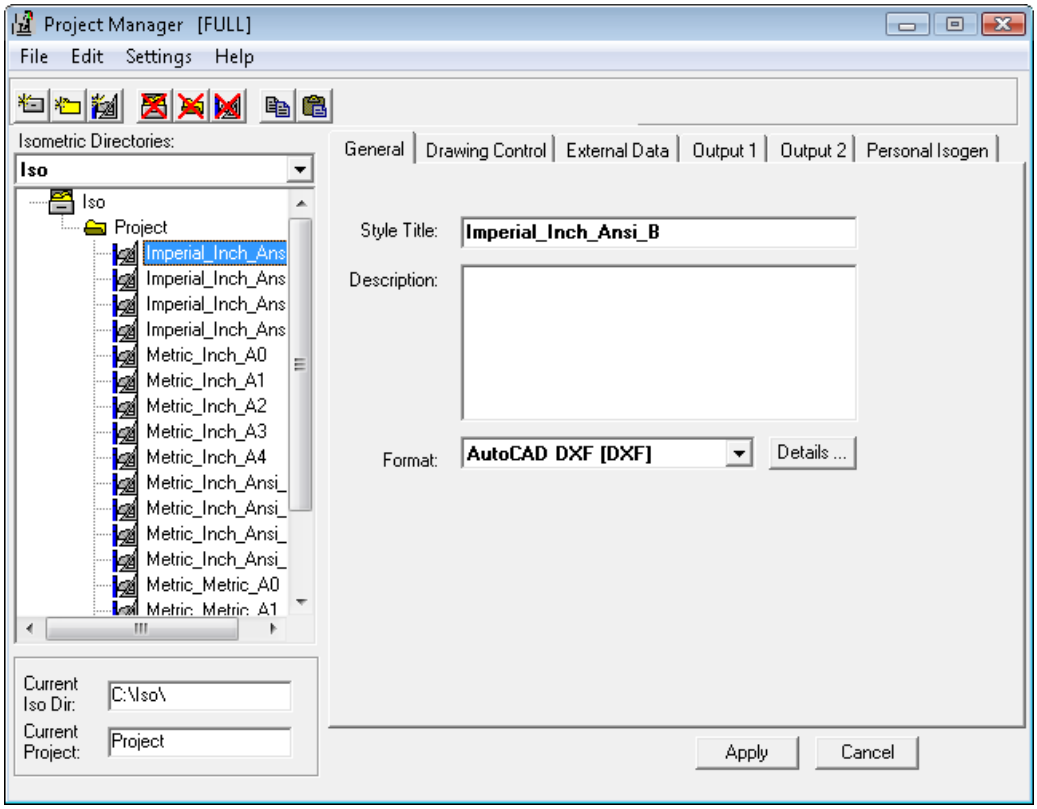
There are two application available for creating and maintaining ISOGEN Styles: Project Manager and I-Configure.

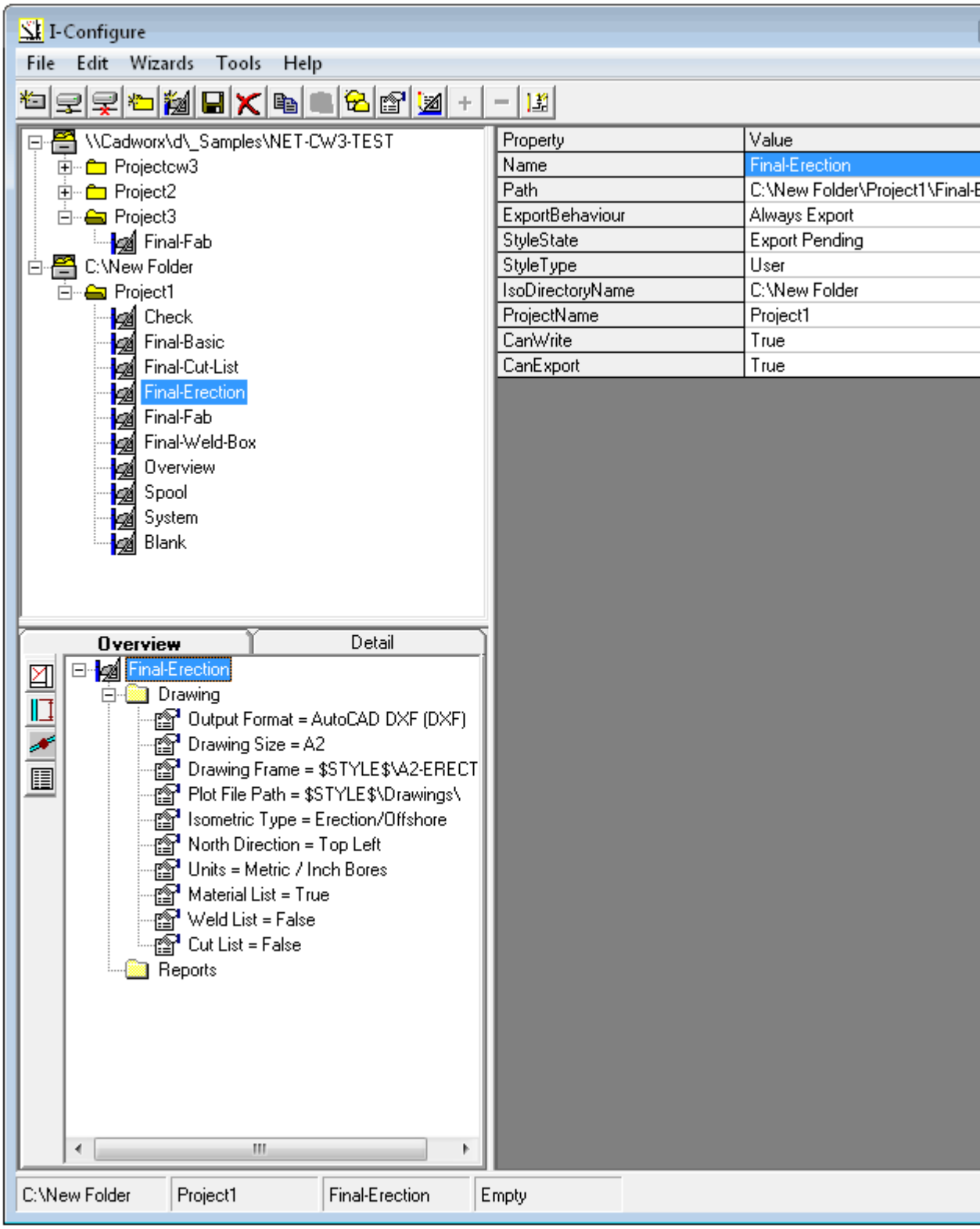
Project Manager in installed with CADWorx ISOGEN and is located at .\CADWorx\ISOGEN\Project_Manager\PIPMAN.EXE

I-Configure is not installed with CADWorx ISOGEN. The install for I-Configure is located at .\CADWorxISOGEN\I-Configure\Setup.exe

CADWorx can use ISOGEN Style created by Project Manager or I-Configure.

To fully understand how ISOGEN works, the user must read the documentation. We will not try to explain every individual concept of ISOGEN within this manual. Please read the ISOGEN PDF documentation.




I-Configure
File Edit Wizards Tools Help
\\Cadworx\d_Samples\NET-CW3-TEST
Projectcw3
Project2
Project3
Final-Fab
C:\New Folder
Project1
Check
Final-Basic
Final-Cut-List
Final-Erection
Final-Fab
Final-Weld-Box
Overview
Spool
System
Blank
Property
Value
Name
Final-Erection
Path
C:\New Folder\Project1\Final-E
ExportBehaviour
Always Export
StyleState
Export Pending
StyleType
User
IsoDirectoryName
C:\New Folder
ProjectName
Project1
CanWrite
True
CanExport
True
Overview
Detail
Final-Erection
Drawing
Output Format = AutoCAD DXF (DXF)
Drawing Size = A2
Drawing Frame = $STYLE$\A2-ERECT
Plot File Path = $STYLE$\Drawings\
Isometric Type = Erection/Offshore
North Direction = Top Left
Units = Metric / Inch Bores
Material List = True
Weld List = False
Cut List = False
Reports
C:\New Folder
Project1
Final-Erection
Empty

ISOGEN Out

Plant menu: ▶ **Accessory** ▶ **ISOGEN** ▶ **ISOGEN Out**

Command line: ISOGENOUT

This function requires the user to select components that are required in an automatically generated isometric. These components must be generated with an authorized ISOGEN hardware lock or this function will inform the user that it could not process the components. The selection process has different options that allow the user to perform selections that are more detailed.

Command: **ISOGENOUT**
Enter an option [**Line number/Select components**] <Select components>:

Line Number

This option allows the user to select a component with an existing line number or lists all line numbers in the model.

Enter an option [**List/Select component**] <Select component>:

Once the line number(s) are selected by either one of the above methods, the program accesses all the components with matching line numbers.

List

See Appendix F: Common List Dialog (see page 647) for more information about this option.

Select component

This option allows the user to select a component with an existing line number. Select the desired component by crossing, window, or any other preferred method.

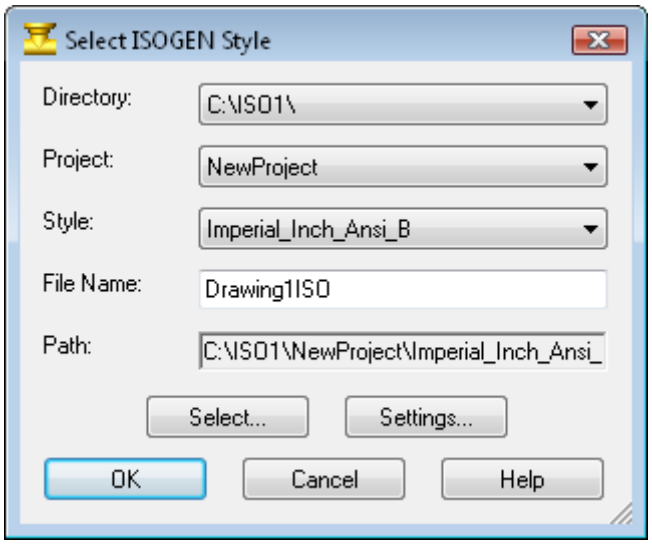
Select a CADWorx entity: *select any CADWorx entity*
Accessing entities with line number of "CS1-6"-305"

Once the entity is selected, it's line number will be displayed in the command line as shown above.

Select component

This option allows the user to select the desired components by crossing, window, or any other preferred method. The user can select components that are from an XREF if the UseObjectsFromXrefs (see page 67) is set to 1.

This dialog allows the user to select different projects, styles and names of the automatically generated isometrics. Please review the beginning of this section regarding the ISOGEN Program Manager (see page 210) for more in information on creating and managing these projects and styles using Project Manager or I-Configure. See the ISOGEN Configuration dialog for explanations of the edit boxes.


Select ISOGEN Style
Directory:
C:\ISO1\
Project:
NewProject
Style:
Imperial_Inch_Ansi_B
File Name:
Drawing1ISO
Path:
C:\ISO1\NewProject\Imperial_Inch_Ansi_
Select...
Settings...
OK
Cancel
Help

The *Directory and Project* edit boxes display all the available projects within the ISOGEN environment. Click on the down arrow key to see a list of all available projects. Use the Program Manager or I-Configure mentioned above to create these directories.

The *Style* edit box displays all the available styles within a project. Click on the down arrow key to see a list of all available projects.

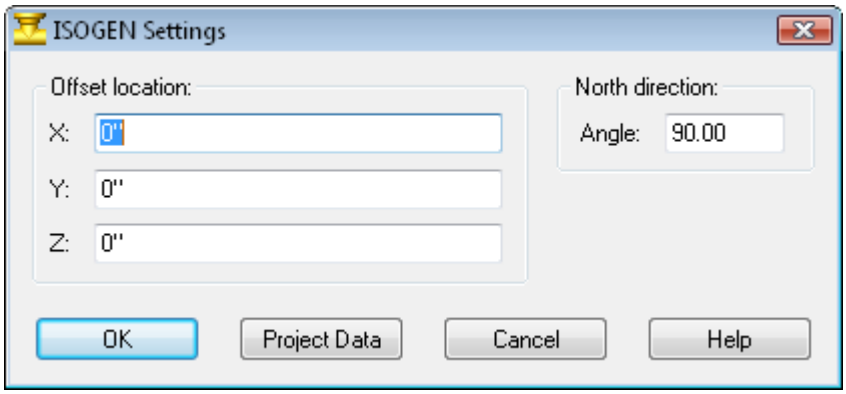
The *File Name* edit box allows the user to name an isometric file if none of the components selected contain a line number. If the selected components contain line number information, the line number from the last component selected is used as the isometric file name.

The *Path* box show the Isometric directory that is used by the ISOGEN Style.

The *Select* button allows the user to select an ISOGEN Style (ISOGEN.FLS file). Use this button to select an ISOGEN style that may be setup on a shared network directory.

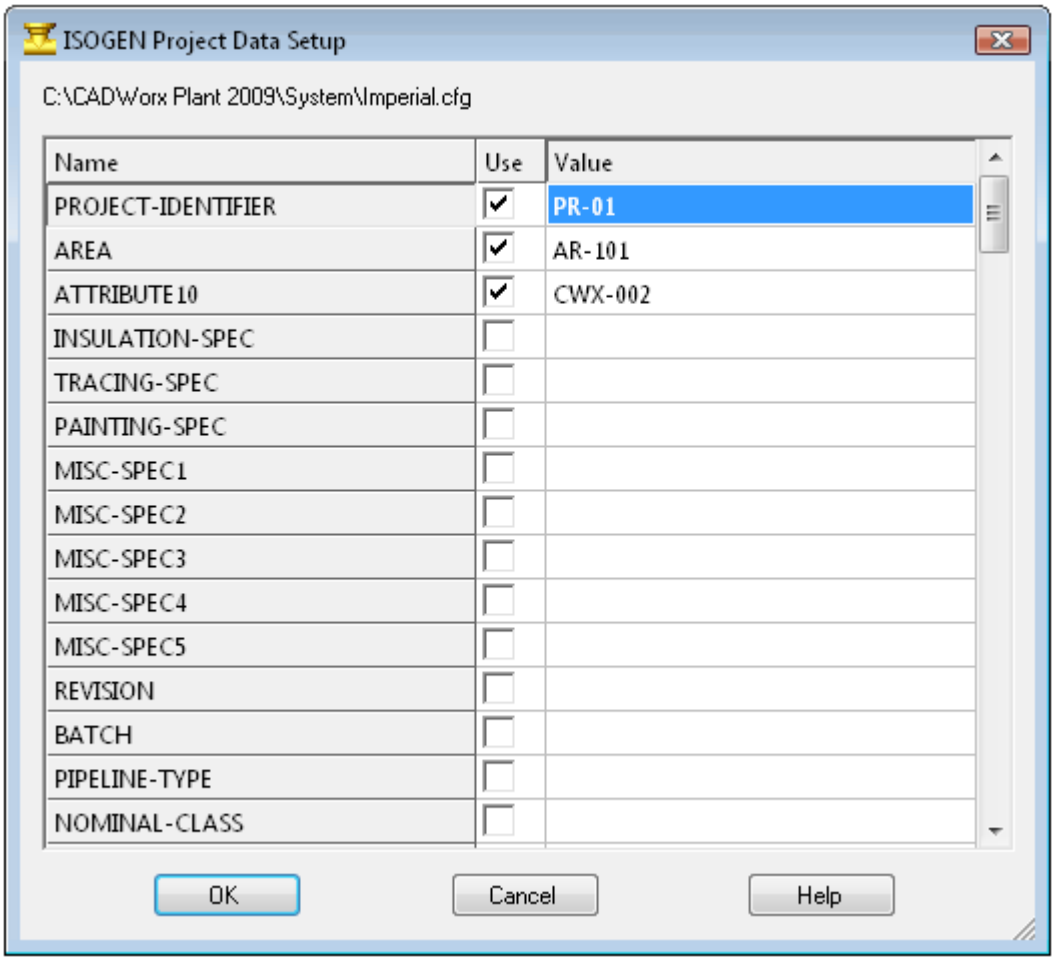
The *Settings* button provides the following dialog which allows the system to be transposed via an X, Y, Z offset distance. In the same dialog, the whole system can be rotated by any angle. This allows any user defined north arrow direction.

- 90 : World +Y axis as north direction (default).
- 180 : World +X axis as north direction.
- 270 : World -Y axis as north direction.
- 0 : World -X axis as north direction.



The *Project Data* button provides the following dialog which allows setting position text (POS) that can be displayed on the isometric drawing file. Review the file ..\CADWorx\ISOGEN\Isogen_Utils\POS_Help.pdf to locate the POS number associated with each item. These values are stored with the configuration file and are used when an isometric is exported. For I-Configure projects, this is the only option to setting these values. For Project Manager projects, these values can be set from the Project Manager -> Project Default dialogs or with this dialog.

After entering a value, click the *Use* check box to store the value in the configuration file. To delete a value from the configuration file, un-check the *Use* setting. The values that have *Use* checked will be shown first the next time the dialog is started.



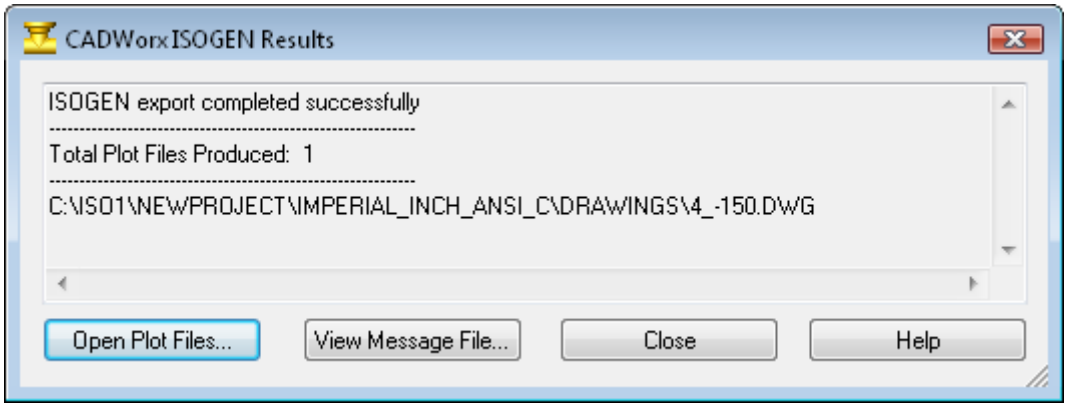
The next prompt informs the user of the number of components processed.

Processed 164 component(s) to file...

As mention above, the components must have been generated with a valid ISOGEN authorized hardware lock. If not, the following message might appear.

Could NOT process 17 component(s) due to permissions failure...
Only components drawn with the ISOGEN interface are allowed...
No components were written to file...

Finally, a dialog appears indicating the results of the components selected.



The top line in the box indicates that export was successful. If there had been a problem, an error such as “IDFGEN reported disconnected pipeline [Personal ISOGEN code 2022]” might be displayed in this box indicating a disconnected system.

The *Total Plot File Produced* will show the number of isometrics produced.

The *Open Plot Files* button opens all the generated ISOs in the current session of AutoCAD (AutoCAD SDI setting has to be set to 0 for the button to be visible)

The *View Message File* opens Notepad and provides vital information that could be used when an error was produced. The text file that contains this information is the *.MES file.

Please read the ISOGEN documentation for details on this file and other features that are not covered here.

ISOGEN Batch

Plant menu: Accessory ISOGEN ISOGEN Batch

Command line: ISOGENBATCH

This command allows the user to generate isometrics using ISOGEN for one or more line numbers. The command uses the default settings from the ISOGEN Project Manager.

Command: **ISOGENBATCH**
Using default settings from ISOGEN Project Manager...
Directory: C:\Iso\
Project: NewProject
Style: Imperial_Inch_Ansi_C

The command then lists all the line numbers available in the model. The user can select one or more from the available line numbers. See Appendix F: Common List Dialog (see page 647) for more information about this dialog.

Once the line numbers are selected, CADWorx Plant sends each line number to ISOGEN for processing. ISOGEN might break up the selected line numbers into multiple sheets.

```
********** Start processing (1"-IG-915-A1) **********
Sent 38 component(s) to ISOGEN for processing...
1 DXF file(s) created:
E:\ISO\NEWPROJECT\IMPERIAL_INCH_ANSI_C\DRAWINGS\1_-IG-915-
A1.DXF
********** End processing (1"-IG-915-A1) **********

********** Start processing (1"-IG-918-A1) **********
Sent 9 component(s) to ISOGEN for processing...
1 DXF file(s) created:
E:\ISO\NEWPROJECT\IMPERIAL_INCH_ANSI_C\DRAWINGS\1_-IG-918-
A1.DXF
********** End processing (1"-IG-918-A1) **********

****************************
********** Summary **********
****************************

*** Line(s) completed successfully ***
1"-IG-915-A1
1"-IG-918-A1
*** Total DXF files:  2

*** Line(s) with errors ***
None

ISOGENBATCH log file written to:
C:\Drawings\Model_ISOGEN.log
```

As each line is processed, any errors encountered are displayed on the screen. A log file is created as well.

Note: ISOGENBATCH will also export line number continuation information to ISOGEN.

ISOGEN Supplemental

Plant menu: Accessory ISOGEN ISOGEN Supplemental

Command line: ISOGENSUPL

There are several types of block available for export to ISOGEN:

ISOGEN_Arrow

ISOGEN_Wall

ISOGEN_Floor

ISOGEN_Instrument

ISOGEN_BreakIn

Arrow, wall and floor blocks have to be inserted along the center line of a pipe component.

Instrument block must have the instrument line connect to the end of an olet component, TAP component, or a wed gap component.

Break in (Tie in) point must be connected to the end point of a component.

To add other block to this menu, review the Graphics (see page 398) topic. The block name must contain "ISOGEN_Arrow", "ISOGEN_Instrument", "ISOGEN_Wall", "ISOGEN_Floor", or "ISOGEN_BreakIn" for export to ISOGEN. For example: To add another instrument type, name the block "ISOGEN_Instrument_4" and add it to the menu.

Note: If placing a flow arrow on a sloped line, align the UCS to the slope before inserting the arrow.

PCF In

Plant menu: ▶ **Accessory** ▶ **ISOGEN** ▶ **PCF In**

Command line: PCFIN

This function draws components into the current drawing that are defined the PCF file. The Piping Component File (PCF) is usually produced from other plant design systems. Utilize this function for importing PCF files from any system that can produce this format. This command requires that proper CADWorx specification and data files be created. If the PCF file used a specification that does not match the CADWorx specification exactly, the model created by PCFIN will have many issues.

The function will present a selection dialog and then request a default specification. There will be an intervention level which is explained in the System In (see page 194) function.

Command: **PCFIN**

PCF Out

Plant menu: ▶ **Accessory** ▶ **ISOGEN** ▶ **PCF Out**

Command line: PCFOUT

This function produces a piping component file (PCF) from components selected in the drawing.

This function presents the standard file selection dialog and prompts for a file name. After providing a file name, the function then prompts the user to select components from the drawing. It allows selection by normal AutoCAD means (window, crossing, etc.) or the user can select components by specifying a line number.

Command: **PCFOUT**
Enter an option [**Line number/Select components**] <Select components>:

Line Number

This option allows the user to select a component with an existing line number or lists all line numbers in the model.

Enter an option [**List/Select component**] <Select component>:

Once the line number(s) are selected by either one of the above methods, the program accesses all the components with matching line numbers.

List

See Appendix F: Common List Dialog (see page 647) for more information about this option.

Select component

This option allows the user to select a component with an existing line number. Select the desired component by crossing, window, or any other preferred method.

Select a CADWorx entity: *select any CADWorx entity*
Accessing entities with line number of "CS1-6"-305"

Once the entity is selected, it's line number will be displayed in the command line as shown above.

Select component

This option allows the user to select the desired components by crossing, window, or any other preferred method. The user can select components that are from an XREF if the UseObjectsFromXrefs (see page 67) is set to 1.

After the prompts are complete, the user will receive a confirmation on how many components were exported. This is a good indicator of whether the right number of components were exported.

Processed 38 component(s)...

ISOGEN Information

Material List Definition

The Material List Definition (MLD) file allows the user to customize the ISOGEN material list. The table below shows how CADWorx data is exported to ISOGEN material list for use with the MLD file.

CADWorx Data (as listed in CEDIT (see page 303) Dialog)	ISOGEN MLD
Alpha size	Not exported ('N.S.' in the MLD is based on the size stored within the component data.)
Short annotation	'COMPONENT-ATTRIBUTE2'
Long annotation	'DESCRIPTION'
Line Number	Not available as an MLD item.
Tag	'COMPONENT-ATTRIBUTE3'
Code	'ITEM-CODE' (If DataBaseCodesISOGEN (see page 56) = 1 in current configuration file)
Code	'COMPONENT-ATTRIBUTE4' (If DataBaseCodesISOGEN (see page 56) = 0 in current configuration file)
Weight	'WEIGHT'
Sort sequence	Not exported
Length	'QTY'
Specification	'COMPONENT-ATTRIBUTE1'

SKEY Information

CADWorx Plant writes components to a Piping Component File with a SKEY (*.PCF) for ISOGEN to use in creating an isometric (See table below).

PRGM #	CADWorx Name	ISOGEN Identifier	SKEY
1	PIPE	PIPE	(Blank)
2	ELBOW 90 LR BUTTWELD	ELBOW	ELBW

3	ELBOW 90 SR BUTTWELD	ELBOW	ELBW
4	ELBOW 90 REDUCING BUTTWELD	ELBOW-REDUCING	ERBW
5	ELBOW 45 BUTTWELD	ELBOW	ELBW
6	BEND 180 LR RETURN BUTTWELD	ELBOW	EUBW
7	BEND 180 SR RETURN BUTTWELD	ELBOW	EUBW
8	TEE STRAIGHT BUTTWELD	TEE	TEBW
9	TEE REDUCING BUTTWELD	TEE	TEBW
10	REDUCER CONCENTRIC BUTTWELD	REDUCER-CONCENTRIC	RCBW
11	REDUCER ECCENTRIC BUTTWELD	REDUCER-ECCENTRIC	REBW
12	CAP BUTTWELD	CAP	KABW
13	STRAIGHT CROSS BUTTWELD	CROSS	CRBW
14	REDUCING CROSS BUTTWELD	CROSS	CRBW
15	LATERAL BUTTWELD	TEE	TSBW
16	STUB END BUTTWELD	LAPJOINT-STUBEND	FLSE
17	LONG WELD NECK	FLANGE / NOZZLE	FLWN / NZFS
18	FLANGE WELD NECK	FLANGE	FLWN
19	FLANGE SLIP ON	FLANGE	FLSJ
20	FLANGE BLIND	FLANGE-BLIND	FLBL
21	FLANGE LAP JOINT	FLANGE	FLLB
22	FLANGE SOCKET WELD	FLANGE	FLSW
23	FLANGE THREADED	FLANGE	FLSC
24	BOLT	BOLT	(Blank)
25	GASKET	GASKET	(Blank)
26	NIPPLE	COUPLING	NRSC
27	CONCENTRIC SWAGE	REDUCER-CONCENTRIC	CSPL
28	ECCENTRIC SWAGE	REDUCER-ECCENTRIC	ESPL
29	90 LR ELBOW SOCKET WELD	ELBOW	ELSW
30	90 STREET ELBOW SOCKET WELD	ELBOW	ELSW
31	90 REDUCING ELBOW SOCKET WELD	ELBOW-REDUCING	ERSW
32	45 ELBOW SOCKET WELD	ELBOW	ELSW
33	STRAIGHT TEE SOCKET WELD	TEE	TESW
34	REDUCING TEE SOCKET WELD	TEE	TESW
35	LATERAL SOCKET WELD	TEE	TSSW
36	STRAIGHT CROSS SOCKET WELD	CROSS	CRSW
37	REDUCING BUSHING SOCKET WELD	REDUCER-ECCENTRIC	RBSC
38	CAP SOCKET WELD	CAP	KASW

39	COUPLING SOCKET WELD	COUPLING	COSW
40	HALF COUPLING SOCKET WELD	OLET	HCSW
41	UNION SOCKET WELD	UNION	UNSW
42	BLEED RING	MISC-COMPONENT	RP
43	90 LR ELBOW THREADED	ELBOW	ELSC
44	90 STREET ELBOW THREADED	ELBOW	ELSC
45	90 REDUCING ELBOW THREADED	ELBOW-REDUCING	ERSC
46	45 ELBOW THREADED	ELBOW	ELSC
47	STRAIGHT TEE THREADED	TEE	TESC
48	REDUCING TEE THREADED	TEE	TESC
49	LATERAL THREADED	TEE	TSSC
50	STRAIGHT CROSS THREADED	CROSS	CRSC
51	REDUCING BUSHING THREADED	REDUCER-CONCENTRIC	RBSC
52	CAP THREADED	CAP	KASC
53	COUPLING THREADED	COUPLING	COSC
54	HALF COUPLING THREADED	OLET	HCSC
55	UNION THREADED	UNION	UNSC
56	PLUG THREADED	MISC-COMPONENT	PL
57	SOCKET WELD ELL-O-LET	ELBOLET	CESW
58	THREADED ELL-O-LET	ELBOLET	CESC
59	WELD ELL-O-LET	ELBOLET	CEBW
60	SOCK-O-LET	OLET	SKSW
61	THREAD-O-LET	OLET	THSC
62	WELD-O-LET	OLET	WTBW
63	BALL VALVE FLANGED or BUTTWELD	VALVE	VBFL or VBBW
64	BALL VALVE SOCKET WELD	VALVE	VBSW
65	BALL VALVE THREADED	VALVE	VBSC
66	BUTTERFLY VALVE FLANGED or BUTTWELD	VALVE	VYFL or VYBW
67	CHECK VALVE FLANGED or BUTTWELD	VALVE	CKFL or CKBW
68	CHECK VALVE SOCKET WELD	VALVE	CKSW
69	CHECK VALVE THREADED	VALVE	CKSC
70	GATE VALVE FLANGED or BUTTWELD	VALVE	VTFL or VTBW
71	GATE VALVE SOCKET WELD	VALVE	VTSW
72	GATE VALVE THREADED	VALVE	VTSC

73	GLOBE VALVE FLANGED or BUTTWELD	VALVE	VGFL or VGBW
74	GLOBE VALVE SOCKET WELD	VALVE	VGSW
75	GLOBE VALVE THREADED	VALVE	VGSC
76	PLUG VALVE FLANGED or BUTTWELD	VALVE	VPFL or VPBW
77	PLUG VALVE SOCKET WELD	VALVE	VPSW
78	PLUG VALVE THREADED	VALVE	VPSC
79	RELIEF VALVE FLANGED or BUTTWELD	VALVE-ANGLE	ARFL or ARBW
80	RELIEF VALVE SOCKET WELD	VALVE-ANGLE	ARSW
81	RELIEF VALVE THREADED	VALVE-ANGLE	ARSC
82	NEEDLE VALVE SOCKET WELD	VALVE	VNSW
83	NEEDLE VALVE THREADED	VALVE	VNSC
84	3-WAY VALVE FLANGED or BUTTWELD	VALVE-3WAY	V3FL or V3BW
85	3-WAY VALVE SOCKET WELD	VALVE-3WAY	V3SW
86	3-WAY VALVE THREADED	VALVE-3WAY	V3SC
87	4-WAY FLANGED or BUTTWELD	VALVE-4WAY	V4FL or V4BW
88	4-WAY SOCKET WELD	VALVE-4WAY	V4SW
89	4-WAY VALVE THREADED	VALVE-4WAY	V4SC
90	CONTROL VALVE FLANGED or BUTTWELD	INSTRUMENT	CVFL
91	CONTROL VALVE SOCKET WELD	INSTRUMENT	CVSW
92	CONTROL VALVE THREADED	INSTRUMENT	CVSC
93	STRAINER FLANGED or BUTTWELD	FILTER	FIFL
94	STRAINER SOCKET WELD	FILTER	FISW
95	STRAINER THREADED	FILTER	FISC
96	ANGLE VALVE FLANGED or BUTTWELD	VALVE-ANGLE	AVFL or AVBW
97	ANGLE VALVE SOCKET WELD	VALVE-ANGLE	AVSW
98	ANGLE VALVE THREADED	VALVE-ANGLE	AVSC
99	GAGE VALVE SOCKET WELD	VALVE	VKSW
100	GAGE VALVE THREADED	VALVE	VKSC
101	SPECTACLE BLIND	MISC-COMPONENT	SB
102	EXPANSION BELLOWS	MISC-COMPONENT	EXFL

103 to 107	Symmetrical user shapes: NCFL Non-symmetrical users shapes with two points: NCFL Non-symmetrical users shapes three or more points: XVFL	MISC-COMPONENT (NCFL) or MULTI-PORT-COMPONENT (XVFL)	NCFL or XVFL
108	ELBOW 22.5 BUTTWELD	ELBOW	ELBW
109	ELBOW 11.25 BUTTWELD	ELBOW	ELBW
110	WYE BW	TEE	TSBW
111	REDUCER CONCENTRIC WITH SOCKET WELD HUB	REDUCER-CONCENTRIC	RCSW
112	REDUCER CONCENTRIC WITH THREADED HUB	REDUCER-CONCENTRIC	RCSC
113	REDUCER ECCENTRIC WITH SOCKET WELD HUB	REDUCER-ECCENTRIC	RESW
114	REDUCER ECCENTRIC WITH THREADED HUB	REDUCER-ECCENTRIC	RESC
115	FLANGE REDUCING SO	FLANGE-REDUCING-CONCENTRIC	FLRC
116	FLANGE REDUCING THREADED	FLANGE-REDUCING-CONCENTRIC	FLRC
117	FLANGE REDUCING WN	FLANGE-REDUCING-CONCENTRIC	FLRC
118	COUPLING SOCKET WELD	COUPLING	COSW
119	COUPLING THREADED	COUPLING	COSC
120	LATERAL-O-LET SOCKET WELD	OLET	LASW
121	LATERAL-O-LET THREADED	OLET	LASC
122	LATERAL-O-LET WELD	OLET	LABW
123	NIP-O-LET THREADED	OLET	NISC
124	NIP-O-LET PLAIN	OLET	NIPL
125	PIPE FLANGED	PIPE-FIXED	FPFL
126	ELBOW 90 LR FLANGED	ELBOW	ELFL
127	ELBOW 45 FLANGED	ELBOW	ELFL
128	TEE STRAIGHT FLANGED	TEE	TEFL
129	REDUCER CONCENTRIC FLANGED	REDUCER-CONCENTRIC	RCFL
130	REDUCER ECCENTRIC FLANGED	REDUCER-ECCENTRIC	REFL
131	WYE FLANGED	TEE	TSFL
132	DIAPHRAGM VALVE SOCKET WELD	VALVE	VDSW
133	DIAPHRAGM VALVE THREADED	VALVE	VDSC
134	DIAPHRAGM VALVE FLANGED or BUTTWELD	VALVE	VDFL or VDBW

135	ROTARY VALVE SOCKET WELD	VALVE	VRSW
136	ROTARY VALVE THREADED	VALVE	VRSC
137	ROTARY VALVE FLANGED or BUTTWELD	VALVE	VRFL or VRBW
138	DUAL CHECK VALVE SOCKET WELD	VALVE	CKSW
139	DUAL CHECK VALVE THREADED	VALVE	CKSC
140	DUAL CHECK VALVE FLANGED or BUTTWELD	VALVE	CKFL or CKBW
141	XBODY GATE VALVE SOCKET WELD	VALVE	VTSW
142	XBODY GATE VALVE THREADED	VALVE	VTSC
143	XBODY GATE VALVE BW	VALVE	VTFL
144	ELBOW 90 WITH SPECIAL END TYPES	ELBOW	EL**
145	ELBOW 90 REDUCING WITH SPECIAL END TYPES	ELBOW-REDUCING	ER**
146	ELBOW 45 WITH SPECIAL END TYPES	ELBOW	EL**
147	TEE STRAIGHT WITH SPECIAL END TYPES	TEE	TE**
148	TEE REDUCING WITH SPECIAL END TYPES	TEE	TE**
149	CROSS WITH SPECIAL END TYPES	CROSS	CR**
150	CROSS REDUCING WITH SPECIAL END TYPES	CROSS	CR**
151	LATERAL WITH SPECIAL END TYPES	TEE	TE**
152	REDUCER CONCENTRIC WITH SPECIAL END TYPES	REDUCER-CONCENTRIC	RC**
153	REDUCER ECCENTRIC WITH SPECIAL END TYPES	REDUCER-ECCENTRIC	RE**
154	STUB WITH SPECIAL END TYPES	CONNECTOR	MPBW
155	WYE WITH SPECIAL END TYPES	TEE	TS**
156	CAP WITH SPECIAL END TYPES	CAP	KA**
157	NUT WITH SPECIAL END TYPES	NUT (BNUT) or CLAMP (CLMP)	BNUT or CLMP
158	BALL VALVE WITH SPECIAL END TYPES	VALVE	VB**
159	BUTTERFLY VALVE WITH SPECIAL END TYPES	VALVE	VY**
160	CHECK VALVE WITH SPECIAL END TYPES	VALVE	VC**
161	GATE VALVE WITH SPECIAL END TYPES	VALVE	VT**
162	GLOBE VALVE WITH SPECIAL END TYPES	VALVE	VG**
163	PLUG VALVE WITH SPECIAL END TYPES	VALVE	VP**

164	3-WAY VALVE WITH SPECIAL END TYPES	VALVE-3WAY	V3**
165	ANGLE VALVE WITH SPECIAL END TYPES	VALVE-ANGLE	AV**
166	MULTIPORT 1 VALVE WITH SPECIAL END TYPES	VALVE, VALVE-ANGLE, VALVE-3WAY	ZG**, 2Z**, 3Z**
167	MULTIPORT 2 VALVE WITH SPECIAL END TYPES	VALVE-MULTIWAY	MZ**
168	MULTIPORT 3 VALVE WITH SPECIAL END TYPES	VALVE-MULTIWAY	MZ**
169	ELBOW 22.5 WITH SPECIAL END TYPES	BEND	BE**
170	ELBOW 11.25 WITH SPECIAL END TYPES	BEND	BE**
171	LATERAL REDUCING WITH SPECIAL END TYPES	TEE	TE**
172	U-BEND WITH SPECIAL END TYPES	Not exported	Not exported
173	WELD GAP	WELD	WW
174 to 197	(Future Components)		
198	GENERIC INSERT	Not exported	Not exported
199	WELD GAP	WELD	WW
200	BENT PIPE	ELBOW, BEND	ELBW, BUBW, PBBW,
201	MITER PIPE	GAP	(Blank)
202	ROLLED PIPE	MISC-COMPONENT	FXBW
203	RESTRAINTS	SUPPORT	(Blank)
204	PAD or SADDLE	REINFORCEMENT-PAD	(Blank)
205	RIGID ELEMENT	GAP	(Blank)
206	SKIPPED ELEMENT	GAP	(Blank)
207	SPECIAL RIGID	GAP	(Blank)
208	TUBING	PIPE or BEND (PB)	(Blank) or PB
209	STOP SIGN	ISO-SPLIT-POINT	(Blank)
210	TAP CONNECTION	TAP-CONNECTION	(Blank)
211	OFFSET TAP CONNECTION	TEE-SET-ON	TTSO

End Type Information

**CADWorx Plant special end types components are written to the PCF based on the end type used in the data file .

CADWorx End Type	ISOGEN End Type**
0	PL
1	BW
2	FL
3	LN
4	LN
5	MP
6	MP
7	MP
8	MP
9	CL
10	CL
11	CL
12	CL
13	CL
14	SW
15	SC

Database

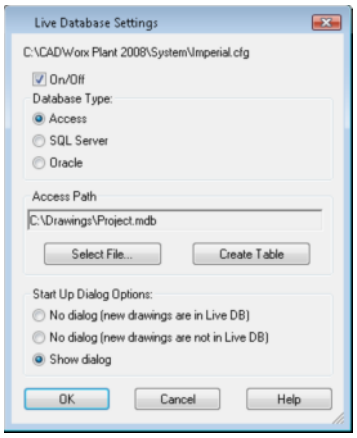
Setup

Plant menu: ▶ **Accessory** ▶ **Database** ▶ **Setup**

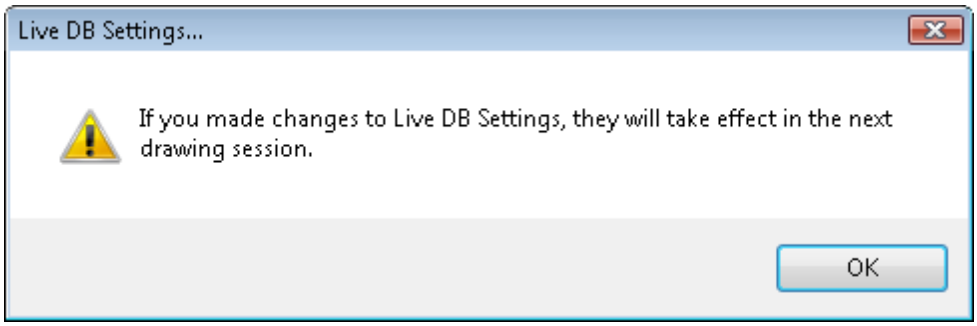
Command line: LIVEDB

CADWorx Plant supports a fully functional live external database. The settings for this database are controlled through this dialog. This system will react according to the configuration file listed in the top of this dialog. When a drawing is started and a configuration file is located within the same directory the drawing will initialize according to that configuration file - live database or not.

Selecting this function, will present the Live Database Settings dialog shown next.



After making any changes to the live database settings, click OK to save the changes. All changes to live database settings take effect in the next drawing session as shown below.



Note: When a component is erased from the drawing, and there is a live database active, it is marked for deletion in the database. When the drawing is closed, the appropriate action will take place within the database. If the drawing is not saved the item is not deleted. If the drawing is saved, then the item is deleted.

On/Off

This toggle turns the live database system on and off.

When the system is on, all components drawn are updated to the database instantaneously. Changes can be made in the drawing or in the database.

When the system is off, all CADWorx Plant component information is stored within the drawing only.

Database Type

These options toggle the type of database to be used as the live database in CADWorx Plant. Depending on the *Database Type* chosen, the path to the database or schema is displayed below.

Please refer to Oracle.txt and SQLServer.txt located in the <INSTALL-DIR>\SYSTEM directory for more information on setting up Oracle or SQL Server respectively.

Live DB - Path

Once the database type is chosen, click on *Create Table* button to create the PIPE table in the database. This is for creating a completely new database table. If the database type is Access, the user can also select an existing database by clicking on the *Select File* button.

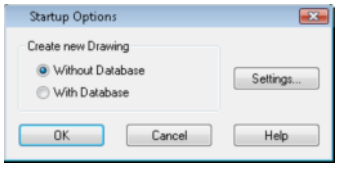
Start Up Dialog Options

There are three options for the start up dialog.

- *No dialog (new drawings are always in the live database)*. This option requires that a new drawing be given a name and saved immediately on creation.
- *No dialog (new drawings are never in the live database)*. This option does not add new drawings to the live database.
- They can be added to the live database using the command.

Note: Drawings that use a live database should be in the same directory with the appropriate configuration file.

- *Show dialog*. This options displays the following dialog every time a new drawing is created. The dialog allows the user to add drawings to the live database or not.



The *Settings* button allows the user to change settings that control whether this dialog appears on start up or not.

Note: Existing drawing can be added to the database by using the DBFGEN (see page 236) command (Pipe LiveDB) or SDBFGEN (Steel LiveDB)

Synchronize

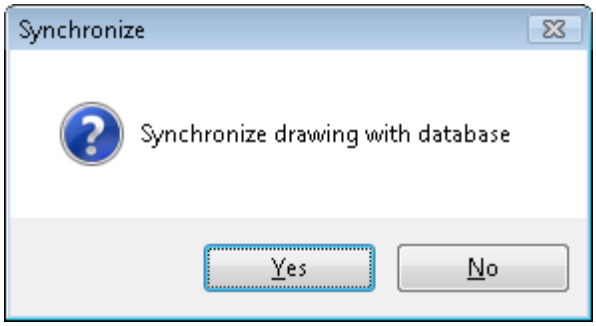
Plant menu: ▶ **Accessory** ▶ **Database** ▶ **Synchronize**

Command line: SYNC

This function checks all components in the drawing and updates them according to the current value within the database tables. This function will check every component within the drawing. This command will not write new records in the database for new components that are in the model that are not in the database.

Command: **SYNC**

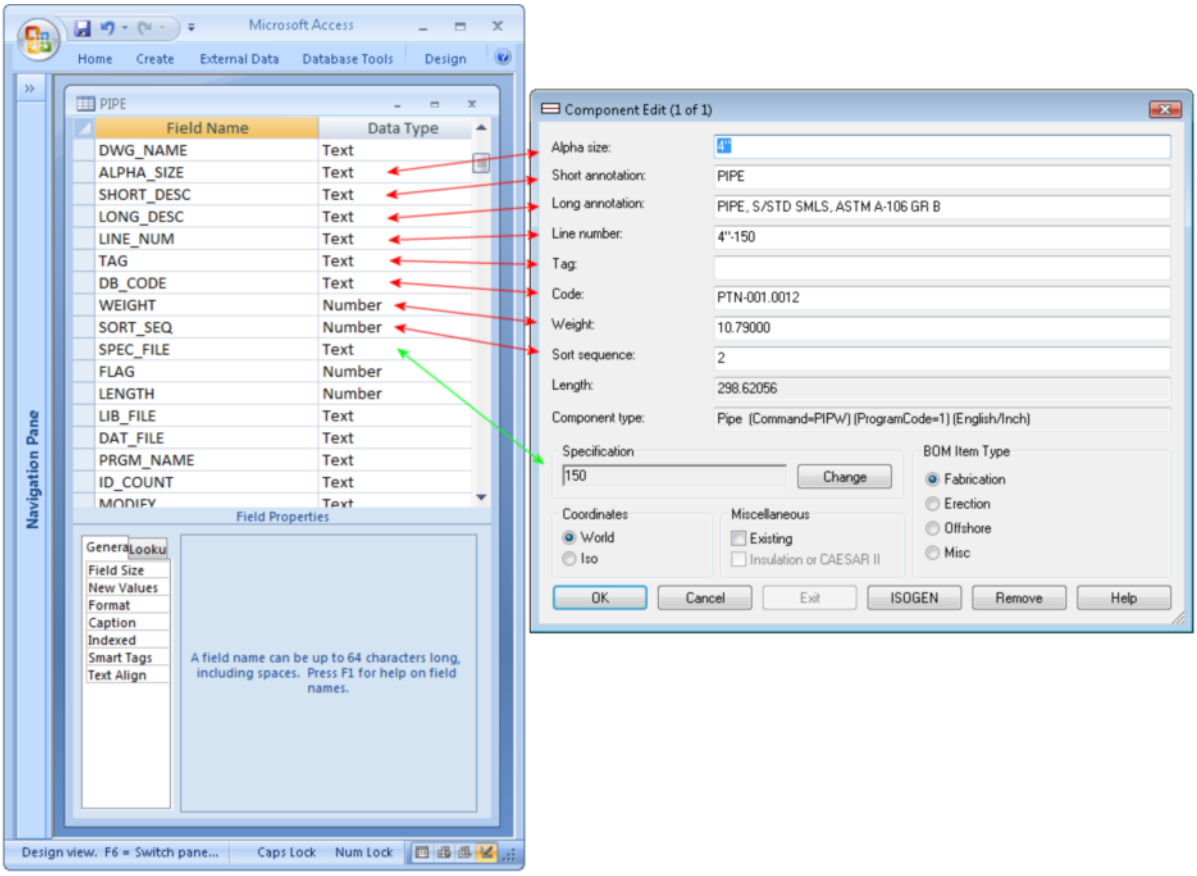
The user will be prompted to confirm the synchronization of the drawing with the database as shown below.



A user would use the SYNC function to update the drawing especially if they know that someone is working in the project database.

When a drawing(s) has a live database attached, the user can enter the database and change fields located within the PIPE table. The columns values which can be modified are the ALPHA_SIZE, SHORT_DESC, LONG_DESC, TAG, LINE_NUM, DB_CODE, and WEIGHT. Changing any other column value will not have any effect on the drawing.

If the SPEC_FILE column value is changed, the values in the other columns listed above will be ignored. The SPEC_FILE column value must be changed to an existing valid specification. The column values listed above will be updated according to the information derived from the specification. When a specification is changed, it must match the base rating of the specification. For example, changing a component from a 150 lb rated specification to a 300 lb rated specification will cause discontinuity within the drawing. This discontinuity will not present itself until a mode convert is performed on the component.



Audit

Plant menu: ▶ **Accessory** ▶ **Database** ▶ **Audit**

Command line: DBAUDIT

This function is provided as a maintenance solution. Whenever the drawing or the database has any problems, this function can be used to assure that there is an entry in the database for each component which has data attached in the drawing.

It also allows the user to delete problems that are evident in the drawing vs. the database. If a problem is found in the drawing, the following prompt allows the user to add the symbol to the database. Here the function found a component that was not present in the drawing. The user has the option to delete this component in the database or to simply ignore it.

```
Command:  DBAUDIT
Checking objects in this drawing with rows in the database...
Checking external database references for this drawing...
```

Here the function found a row in the database that is not in the drawing. The user has the option to delete this row in the database or to simply ignore it. No would simply ignore it.

```
A row with ID_Count of 100004 is not referenced...
Delete row from database? Yes/<No>:  Y
1 row(s) removed from the database...
```

Export

Plant menu: ▶ **Accessory** ▶ **Database** ▶ **Export**

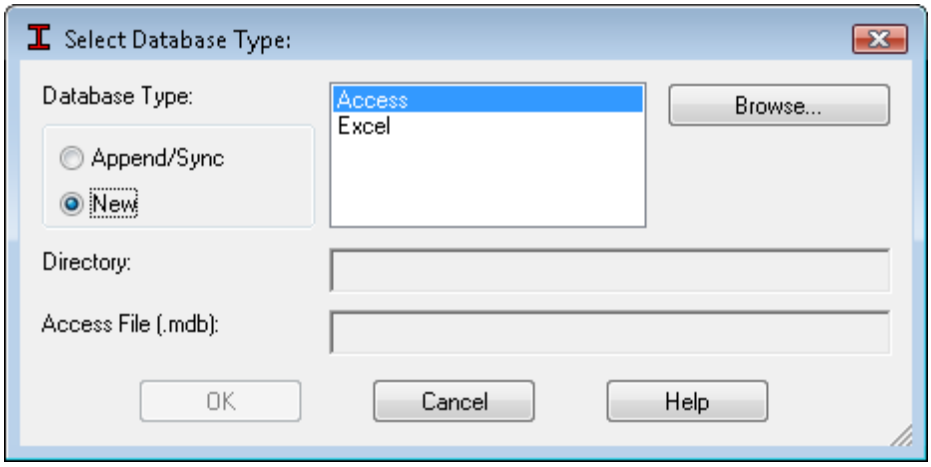
Command line: DBFGEN

This function can export CADWorx component information to a Microsoft Access or Microsoft Excel file when a static database in used. After executing the command, selection of components from the drawing is required. After selection, the following dialog will appear requesting the type of database desired.

Note: When using live database, this command will update the appropriate database setup for the drawing.

With the Append/Sync this function can be used to maintain a perfect database at all times. When the Append/Sync toggle is selected, the exported information will update the database intelligently. Items that have been deleted in the drawing(s) will be deleted out of the database. Items that have been changed in the drawing(s) will be changed in the database. Any new items added to the drawing(s), will be added to the database. Use this functionality once a day or five times an hour. This toggle will always maintain a database according to the drawing(s).

Command: **DBFGEN**



Enter an option **[Line number/Select components]** <Select components>:

Line Number

This option allows the user to select a component with an existing line number or lists all line numbers in the model.

Enter an option [**List/Select component**] <Select component>:

Once the line number(s) are selected by either one of the above methods, the program accesses all the components with matching line numbers.

List

See Appendix F: Common List Dialog (see page 647) for more information about this option.

Select component

This option allows the user to select a component with an existing line number. Select the desired component by crossing, window, or any other preferred method.

Select a CADWorx entity: *select any CADWorx entity*
Accessing entities with line number of "CS1-6"-305"

Once the entity is selected, it's line number will be displayed in the command line as shown above.

Select component

This option allows the user to select the desired components by crossing, window or any other preferred method.

The *Append/Sync* toggle appends or synchronizes the drawing and the database file. It will remove entries from the database that no longer exist in the drawing and it updates any components in the database according to the drawing. It will also perform an Database Audit at the end.

Database type refers to either Microsoft Access, or Microsoft Excel. Select one of these types depending on the database system the user is writing or appending to.

These disabled edit boxes at the bottom of the dialog tell the user the name of database and directory in use. These edit boxes are automatically filled in when the browse button is used. If appending, many users can append to the same database at the same time.

The *New* toggle creates a new database based on the chosen directory and database type.

If live database is being used, the function will prompt to update the live database as shown below.

DBFGEN can only update live database...
Continue to update live database? [**Yes/No**] <Yes>:

Import

Plant menu: Accessory ▶ Database ▶ Import

Command line: DBFIN

This command can import and draw CADWorx components from a Microsoft Access or Microsoft Excel file that stores CADWorx component information. This command is not available when a live database is being used.

Note: CADWorx Plant supports only Microsoft Access or Microsoft Excel when using the DBFIN command.

When the command is run, a standard file selection dialog appears requesting the user to select the database to import. Once the database is selected, the program presents a list of drawings using the common list (see page 647) dialog. The user can select one or more drawings to import. CADWorx Plant then draws the imported components in single line mode. These can be converted to any other modes using the Mode Convert (see page 295) tools available.

P&ID View

Plant menu: ▶ **Accessory** ▶ **Database** ▶ **P&ID View**

Command line: PIDVIEW

This command will start the P&ID View palette. This palette can be used to link data between the Plant and P&ID table in a project database.

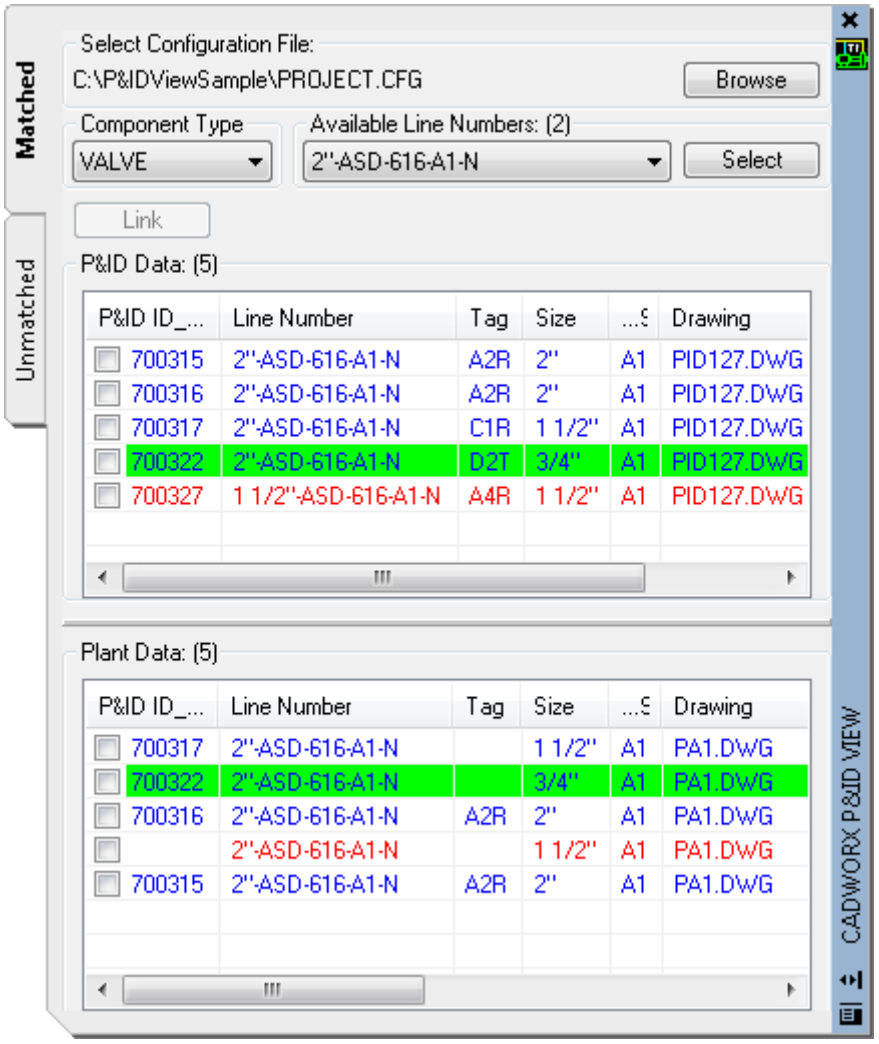
The following has to be setup in order to use this palette:

1. A CADWorx P&ID project has to be created that uses the P&ID live database system.

2. A CADWorx Plant project has to be created that uses the Plant live database system.

3. With Microsoft Access, database tables used by Plant and P&ID have to be written to the same Access file. With SQL Sever and Oracle, tables used by Plant and P&ID have to be written to the same database.

4. Associations between in-line components and the main process line in the P&ID drawings have to created with CADWorx P&ID COMBINEPROCESS command (see CADWorx P&ID help file for information on this command).


Matched
Unmatched
Select Configuration File:
C:\P&IDViewSample\PROJECT.CFG
Browse
Component Type
VALVE
Available Line Numbers: (2)
2"-ASD-616-A1-N
Select
Link
P&ID Data: (5)
P&ID ID_...
Line Number
Tag
Size
...S
Drawing
700315 2"-ASD-616-A1-N A2R 2" A1 PID127.DWG
700316 2"-ASD-616-A1-N A2R 2" A1 PID127.DWG
700317 2"-ASD-616-A1-N C1R 1 1/2" A1 PID127.DWG
700322 2"-ASD-616-A1-N D2T 3/4" A1 PID127.DWG
700327 1 1/2"-ASD-616-A1-N A4R 1 1/2" A1 PID127.DWG
Plant Data: (5)
P&ID ID_...
Line Number
Tag
Size
...S
Drawing
700317 2"-ASD-616-A1-N 1 1/2" A1 PA1.DWG
700322 2"-ASD-616-A1-N 3/4" A1 PA1.DWG
700316 2"-ASD-616-A1-N A2R 2" A1 PA1.DWG
2"-ASD-616-A1-N 1 1/2" A1 PA1.DWG
700315 2"-ASD-616-A1-N A2R 2" A1 PA1.DWG
CADWORX P&ID VIEW

Select Configuration File

- The current configuration file (Project.cfg) in use will be listed.
- *Browse button:* Click this button to select a different Project.cfg file

Available Line Numbers

- This list will have line numbers that are in both the P&ID database and Plant database.
- *Select button:* Click this button to select a component in the current Plant model drawing. The line number of this component will be set to the current line number for P&ID View

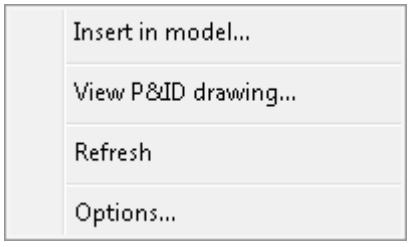
Component Type: This list will have all the component tables from the P&ID database.

Link button: Use this button to link or unlink P&ID and Plant components.

P&ID Data:

- *P&ID ID_Count:* This column will list the P&ID ID_COUNT from the P&ID database.
- *Line Number / Tag / Size / Spec / Drawing:* These columns will list data of the P&ID component.
- Right clicking an item will bring up four menu options: *Insert in model, View P&ID drawing, Refresh, Options*

1. *Insert in model:* This menu time will read the size, spec, and component type from the P&ID database and attempt to insert the component into the Plant model.
2. *View P&ID drawing:* This menu item will open a dialog with a preview of the P&ID drawing (see details below).
3. *Refresh:* This menu item will refresh the P&ID and Plant Data lists.
4. *Options:* This menu item will display a dialog that will allow you to set the colors of the items displayed in the P&ID and Plant data (see details below).



- Left clicking an item will update the P&ID drawing viewer dialog if the viewer dialog is active. Double clicking an item will run the "Insert in model" menu option.

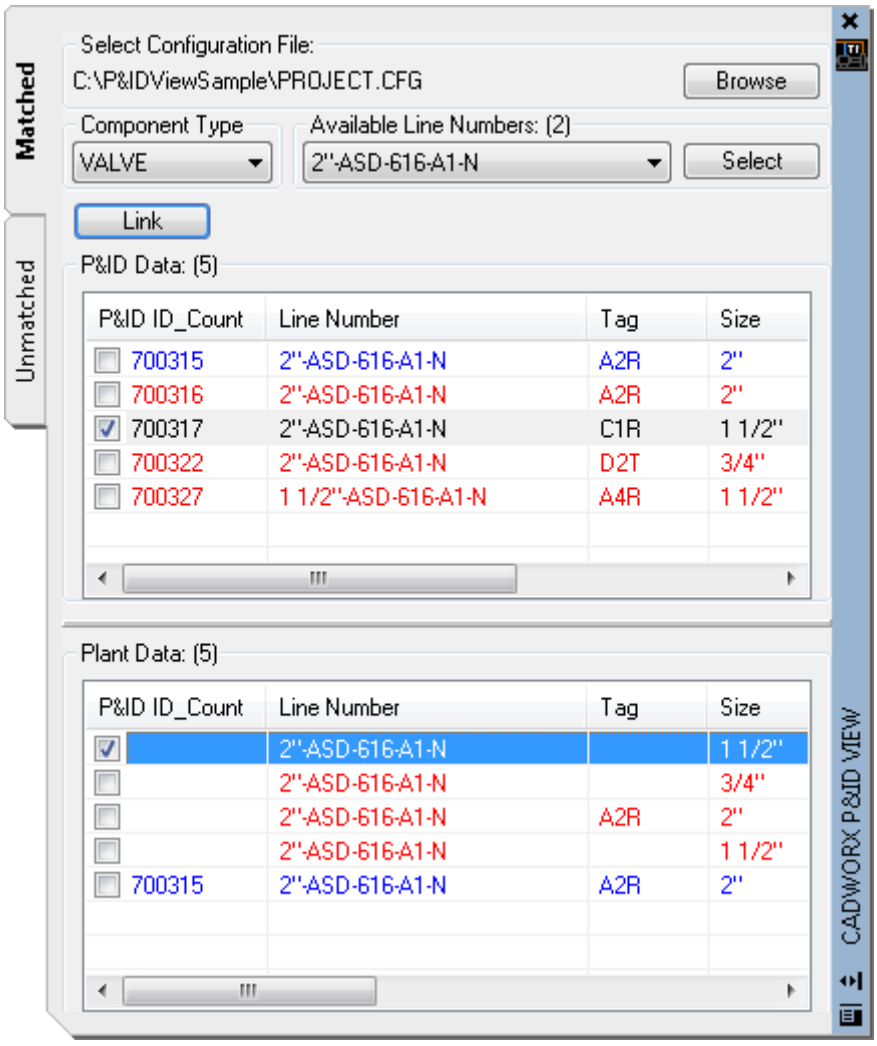
Plant Data:

- *P&ID ID_Count:* If a link between the P&ID component and the Plant component is made, the ID_COUNT of the P&ID component is listed in this column. The Plant database (PIPE table) will save the P&ID ID_COUNT in the RESERVE_10 column.
- *Line Number / Tag / Size / Spec / Drawing / Description:* These columns will list data of the Plant component.
- Right clicking an item will bring up four menu options: *CEdit, Zoom.*

1. *CEdit:* This menu item will display the CEdit dialog.
2. *Zoom:* This menu item will zoom and highlight the component in the model
3. *List data from all drawings:* This option will list Plant Data from all drawings that have the line number
4. *List data from current drawing:* This option will only list Plant Data from the current drawing that is open.
5. *Refresh:* This menu item will refresh the P&ID and Plant Data lists.

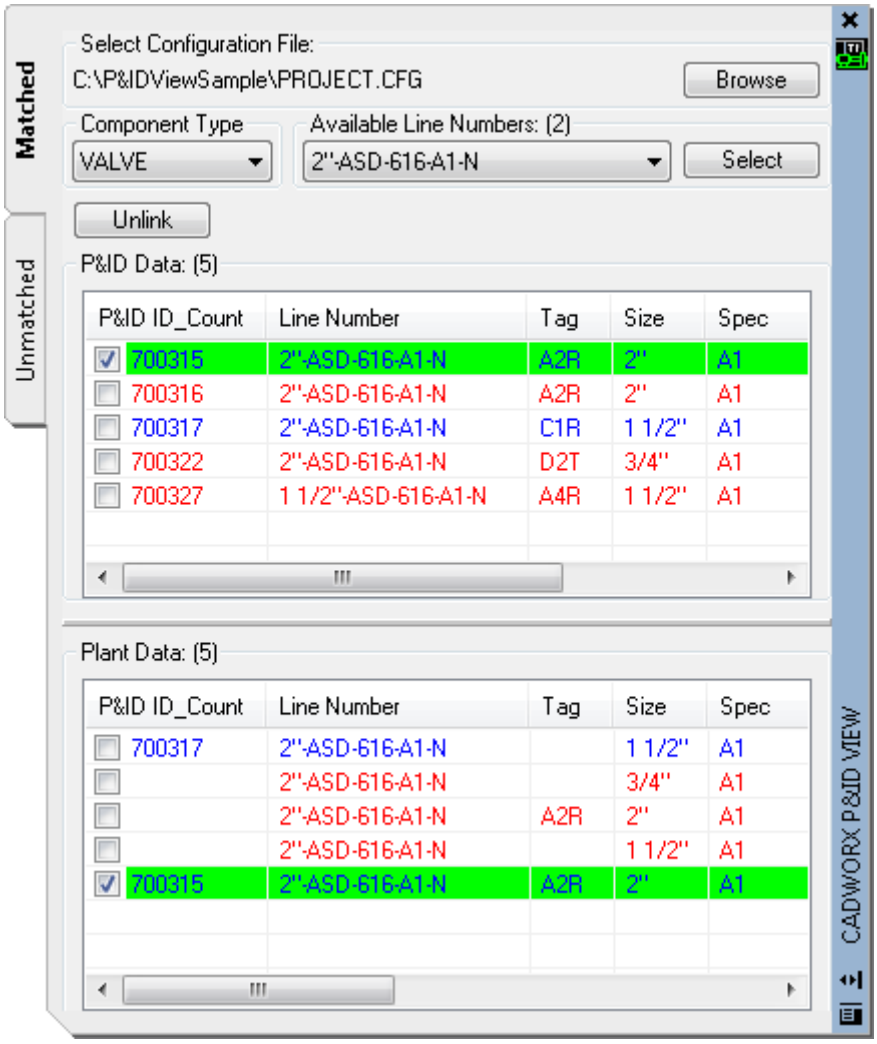
Link Components:

To link components, select one item in the P&ID Data list and one item in the Plant Data list by clicking the check boxes and clicking the Link button. The P&ID ID_Count value in the Plant Data section will update. Vales, Reducers and Nozzles components will be available for liking. Instrument components cannot be linked.



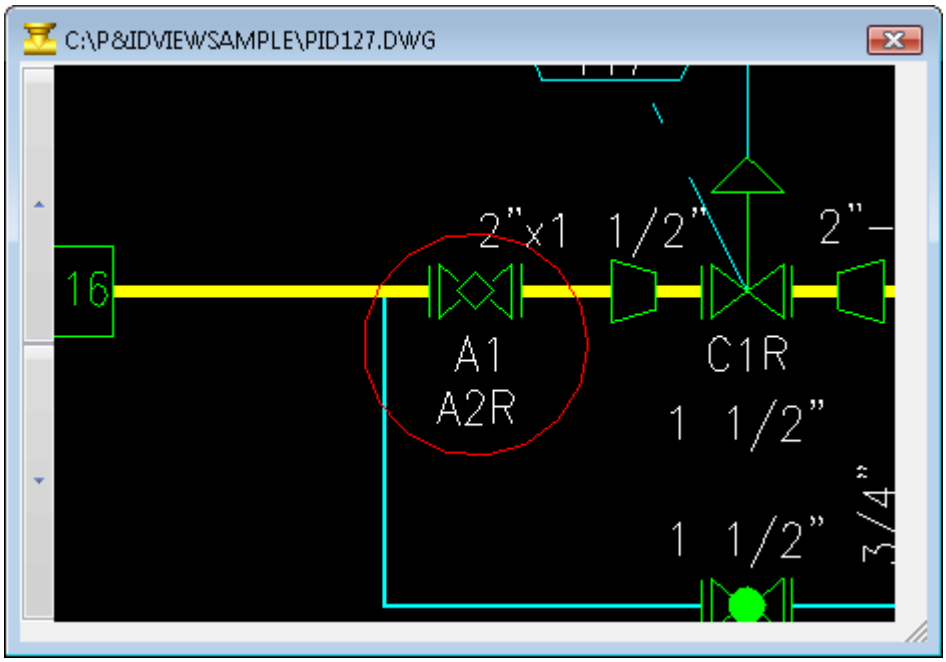
Unlink Components:

To unlink components, select the two components that have been linked. The link button will change to Unlink. Click the Unlink button to remove the link between the two components.



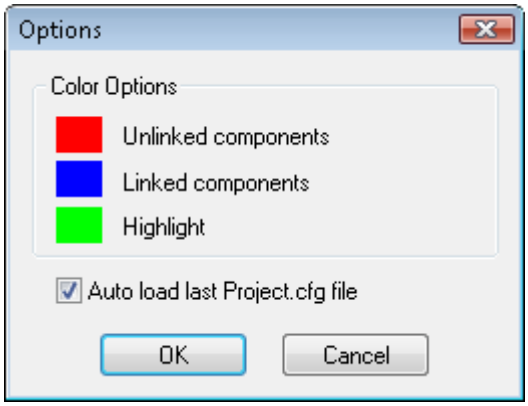
P&ID Viewer:

To zoom inside the viewer, use the two arrow buttons on the left side of the dialog. Zooming inside the viewer can also be done by holding down the left mouse button while rotating the mouse wheel. The viewer dialog is closed when switching between drawings in the same session of AutoCAD. The dialog also has to be started with every new session of AutoCAD.



Color Options:

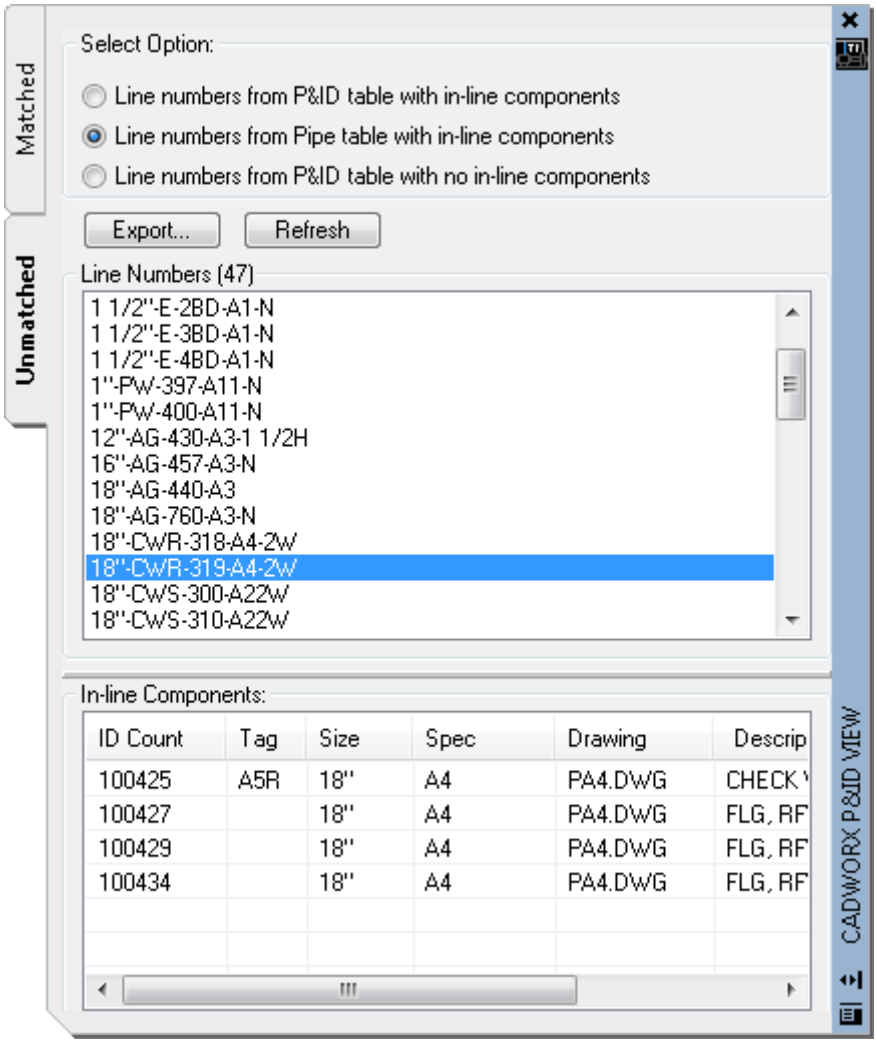
This dialog will allow you to set the colors of the items displayed in the P&ID and Plant Data lists.



Unmatched Tab:

The Unmatched Tab will display line number information from the P&ID and Plant databases.

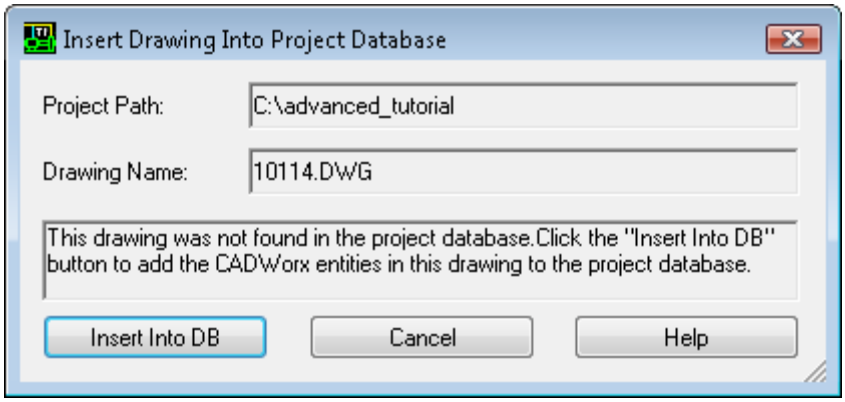
- *Line number from P&ID table with in-line components:* These are line number from the P&ID database that do not have a corresponding line number in the Plant database.
- *Line numbers from Pipe table with in-line components:* These are line number from the Plant database (PIPE table) that do not have a corresponding line number in the P&ID database.
- *Line numbers from P&ID table with no in-line components:* These are line number from the P&ID database that do not have any P&ID components associated with them in the P&ID database.
- Export button: Click this button to export all data from the Unmatched tab to a report. The report will be exported to the same directory as the Project.cfg. The file names of the reports will be P&IDViewUnmatched1.html, P&IDViewUnmatched2.html, and P&IDViewUnmatched3.html
- Refresh button: Click this button to refresh the data presented in the Unmatched Tab.


Matched
Unmatched
Select Option:
Line numbers from P&ID table with in-line components
Line numbers from Pipe table with in-line components
Line numbers from P&ID table with no in-line components
Export...
Refresh
Line Numbers (47)
1 1/2"-E-2BD-A1-N
1 1/2"-E-3BD-A1-N
1 1/2"-E-4BD-A1-N
1"-PW-397-A11-N
1"-PW-400-A11-N
12"-AG-430-A3-1 1/2H
16"-AG-457-A3-N
18"-AG-440-A3
18"-AG-760-A3-N
18"-CWR-318-A4-2W
18"-CWR-319-A4-2W
18"-CWS-300-A22W
18"-CWS-310-A22W
In-line Components:
ID Count
Tag
Size
Spec
Drawing
Descrip
100425
A5R
18"
A4
PA4.DWG
CHECK
100427
18"
A4
PA4.DWG
FLG, RF
100429
18"
A4
PA4.DWG
FLG, RF
100434
18"
A4
PA4.DWG
FLG, RF
CADWORX P&ID VIEW

Insert Drawing

Whenever a CADWorx Plant drawing is opened from a current project directory, CADWorx Plant attempts to insert it into the existing project. The following dialog is provided for the user to acknowledge this action. When a CADWorx Plant drawing is inserted into the database, rows are added to the appropriate tables within the database.

Drawings created without a database will have xdata attached to each component that allows the program to automatically add rows into the appropriate tables.



Exterior File

Pipe In

Plant menu: ▶ **Accessory** ▶ **Exterior File** ▶ **Pipe In**

Command line: PIPEIN

This function draws components into the current drawing. Use it in conjunction with the PIPEOUT command. Files with a *.PMF and *.PMI (AutoISO isometric data files) can be used with this function.

Command: **PIPEIN**

Pipe Out

Plant menu: ▶ **Accessory** ▶ **Exterior File** ▶ **Pipe Out**

Command line: PIPEOUT

This function writes components into an exterior data file. Used it in conjunction with PIPEIN command. The file with have a *.PMF extension.

Command: **PIPEOUT**
Enter an option [**Database/Line number/Select components**] <Select components>:

Database

This option allows the user to create isometrics or export components from an external database that can contain line numbers that span across different drawing files. The user is first asked to select the external database using a standard file open dialog.

Note: If a live database (see page 231) is used, the user will not be prompted to select the external database and the live database will be used for creating the isometrics or external files.

Without Line Numbers

If no line numbers or stop signs are detected in the database, the following prompt is displayed.

Database does not have any line number or stop signs...

If the database contained line numbers but no stop signs, the user will be prompted as to whether the stop signs should be used or not.

Use Stop Signs [**Yes/No**] <Yes>:

Yes

If YES is selected, continuity is checked and the ISOs are broken as a stop sign is encountered.

No

Ignores all stop signs and process all components checking for continuity.

With Line Numbers

If line numbers are found in the database, the following prompt is displayed:

Enter an option [**Line number/All components**] **<All components>**:

Line number

Selecting this option displays the common line list dialog (see page 647). One or more line numbers can be selected.

Generate files based on [**Continuity/Line number**] <Continuity>:

Continuity

This option checks the continuity of the line numbers selected. The function will combine line numbers that are connected to each other.

Line number

This option ignores continuity between the line numbers selected and will create separate files for each line number selected. Continuity within the same line number is checked. The file names will include the line number as part of it.

All components

All components will be processed without regard to line numbers.

If stop signs are detected, the user will be prompted as to whether the stop signs should be used or not.

Use Stop Signs [**Yes/No**] <Yes>:

Yes

If YES is selected, continuity is checked and ISOs are broken as a stop sign is encountered.

No

Ignores all stop signs and process all components checking for continuity.

When the function is completed, the number of files generated is shown.

Generated 12 AutoISO files:
E:\Drawings\COADE_001.pmi
to
E:\Drawings\COADE_012.pmi

Line Number

This option allows the user to select a component with an existing line number or lists all line numbers in the model.

Enter an option [**List/Select component**] <Select component>:

Once the line number(s) are selected by either one of the above methods, the program accesses all the components with matching line numbers.

List

See Appendix F: Common List Dialog (see page 647) for more information about this option.

Select component

This option allows the user to select a component with an existing line number. Select the desired component by crossing, window, or any other preferred method.

Select a CADWorx entity: *select any CADWorx entity*
Accessing entities with line number of "CS1-6"-305"

Once the entity is selected, it's line number will be displayed in the command line as shown above.

Select component

This option allows the user to select the desired components by crossing, window, or any other preferred method. The user can select components that are from an XREF if the UseObjectsFromXrefs (see page 67) is set to 1.

C.G. Generator

Plant menu: ▶ **Accessory** ▶ **C.G. Generator**

Command line: CG

This routine provides center of gravity calculations for components drawn in X,Y planes (2D plans) or in X,Y,Z planes (3D Model). It will not provide center of gravity locations for equipment and other miscellaneous items that do not have a centerline xdata. The user can use a generic attach (from Plant>Toolbars>Misc) which will provide the desired center of gravity location and weight.

Selection can be done by crossing, window, or selecting individual component centerlines.

Command: **CG**

Enter an option **[Line number/Select components]** <Select components>:

Line Number

This option allows the user to select a component with an existing line number or lists all line numbers in the model.

Enter an option [**List/Select component**] <Select component>:

Once the line number(s) are selected by either one of the above methods, the program accesses all the components with matching line numbers.

List

See Appendix F: Common List Dialog (see page 647) for more information about this option.

Select component

This option allows the user to select a component with an existing line number. Select the desired component by crossing, window, or any other preferred method.

Select a CADWorx entity: *select any CADWorx entity*
Accessing entities with line number of "CS1-6"-305"

Once the entity is selected, it's line number will be displayed in the command line as shown above.

Select component

This option allows the user to select the desired components by crossing, window or any other preferred method.
Select objects:
99 found (not necessarily the number of components)
44 were filtered out.
Select objects: *select more or enter to end*

The next option writes the information to an external file. The user can hit enter for no file or enter a file name.

Enter name of file or enter for none: C:\CG\32-002.CGC

View file [Yes/No] <Yes>: *enter to view items*

If the user answered yes to the preceding prompt, they will be able to view the contents of the file name above. If the user just hit enter at the prompt asking for a file name, no file written, they will still be able to view the contents of CADWorx Plant's default file. A list of items will scroll by of each component found. It is best to write to a file, then view it with a text editor (NOTEPAD).

```
Total count...55 items
Listing...finished
Items....
Weight Sum  =        1759.22 lb
Sum (X * W) =        44829.7300 ft-lb
Sum (Y * W) =        25511.6400 ft-lb
Sum (Z * W) =        4750.8520 ft-lb

The Center of Gravity X,Y,Z is 25.4827 ft 14.5017 ft 2.7005 ft
Insert drawing marker [Yes/No] <Yes>:
```

Note: To combine center of gravity reports from steel and piping components, take the weight summations and add them together. Take each separate moment (ft-lb) X, Y, Z and add them together separately. To find the new X centroid location, divide the total X moment by the total weight. To find the new Y and Z centroid location, divide them by the total weight.

Specifications

Editor

Plant menu: ▶ **Accessory** ▶ **Specifications** ▶ **Editor**

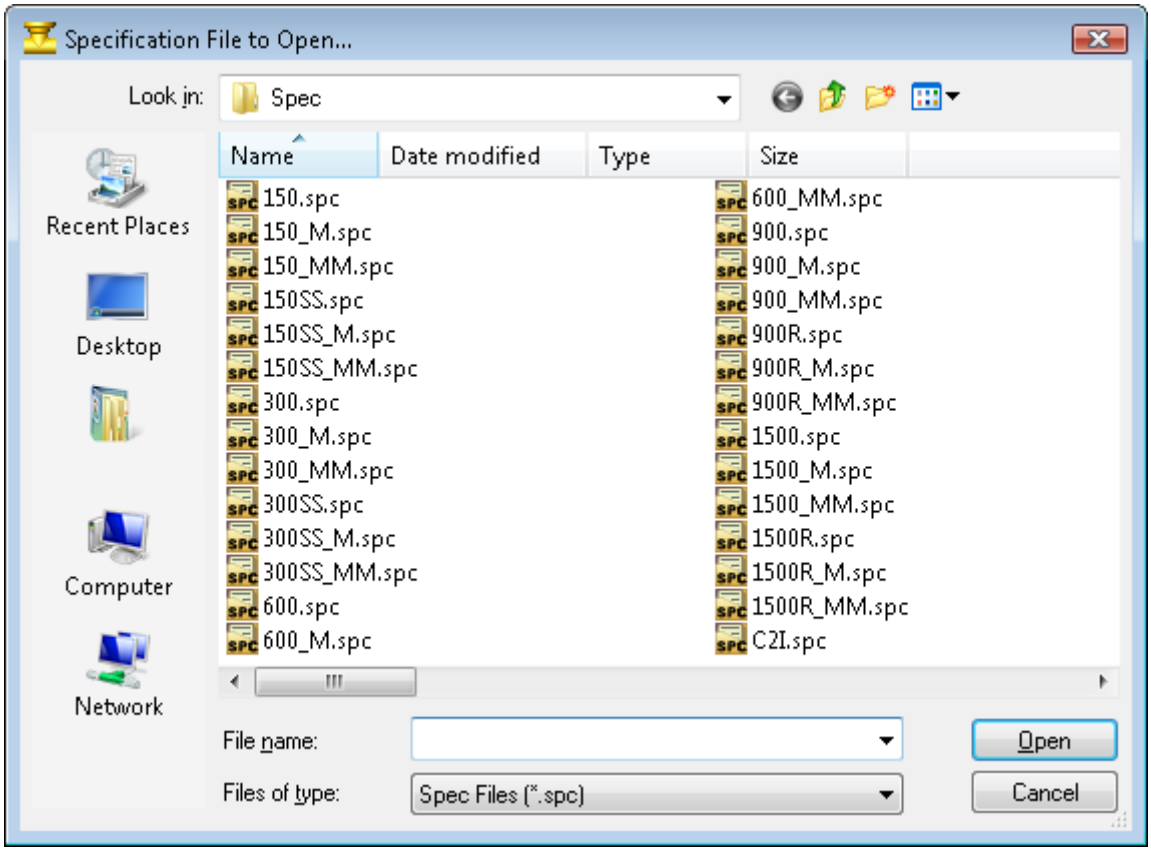
Command line: SPECEDIT

The specification editor allows the modification of the specifications supplied with CADWorx Plant; creates any user desired specifications, and saves them. It also allows the user to edit the specification while in the drawing environment. It can also be used as a standalone program and can be copied to any machine. It does not require a hardware lock (ESL). SPECEDIT will prompt the user to select a specification to modify.

Note: The specification file is specially formatted text file. If modifying the file with a text editor (Notepad.exe), care should be taken to not change the structure to the file. If the specifications file is not formatted correctly, components that use the spec will not draw.

Selecting this function, presents the file selection dialog shown below. The user can select the specification they would like to view/modify and click *Open*.

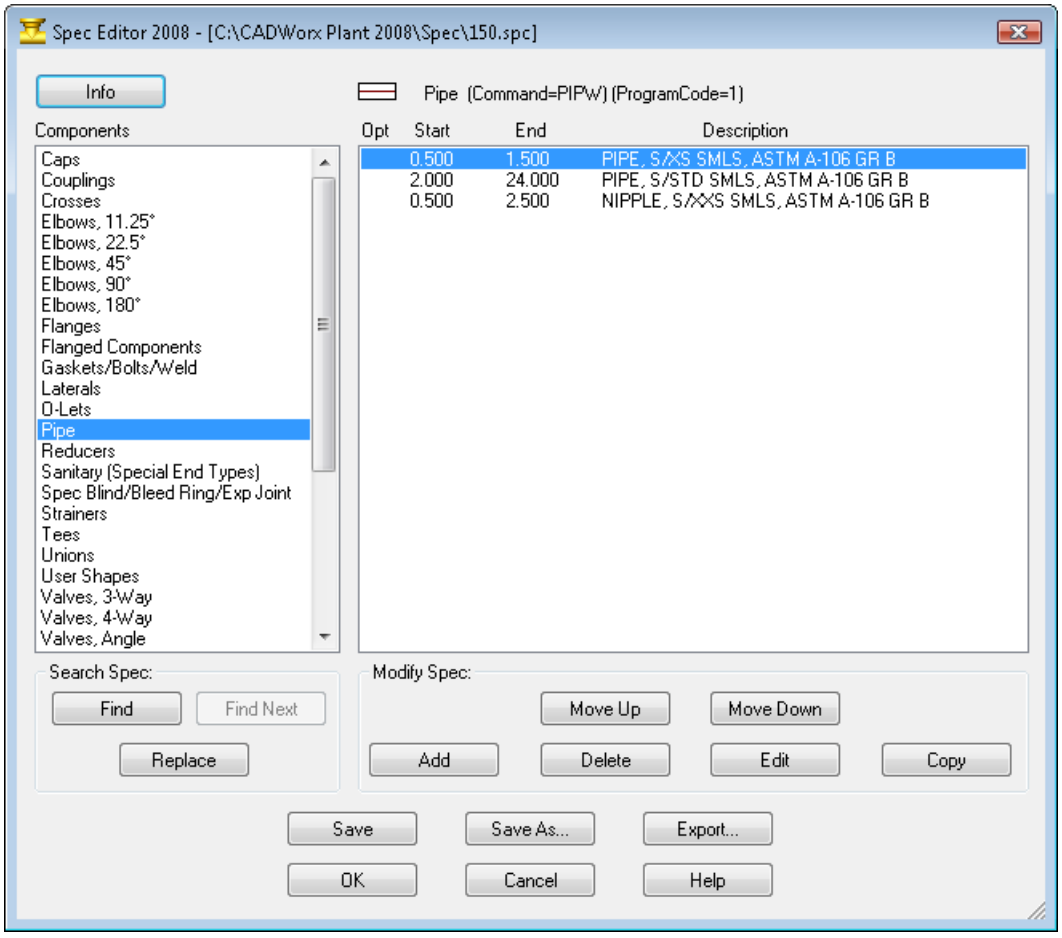
Command: **SPECEDIT**



After selecting the desired specification, the main editing dialog box shown below will be presented. This dialog box allows the user to:

1. Edit the specification information.
2. Edit the individual items.
3. Add new components to the specification.
4. Delete components from the specification.
5. Perform text search and replacements.
6. Save the specification.
7. Save the specification under another name.

Use the scroll bar on the left side of the dialog box to scroll down to the remaining items.

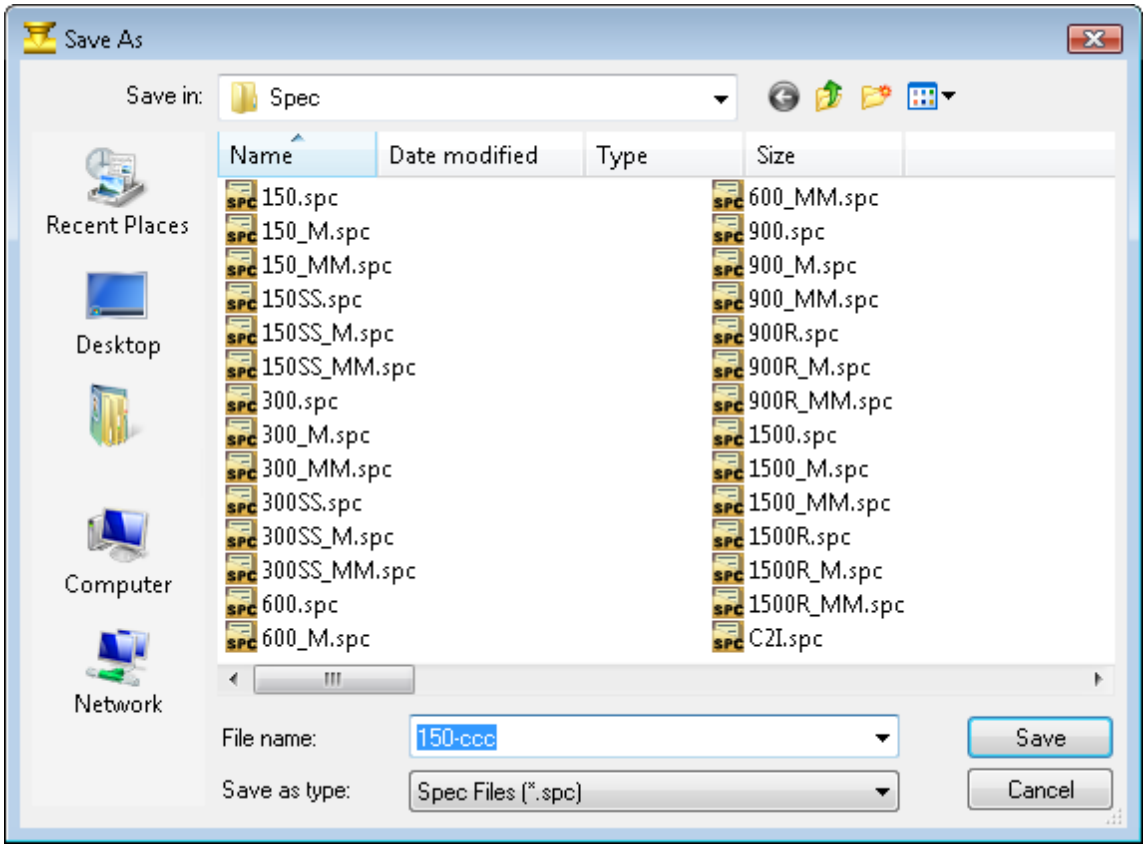


Save

This button saves the current specification with the current modifications.

Save As

This option provides a dialog box requesting the user to enter a file name. The user can select one of the existing files or enter a new name. This is one way a new specification can be made.



Export

This option allows the user to export the currently open specification to a Text or Excel file.



Note: Specifications exported cannot be imported back into the program.

Cancel

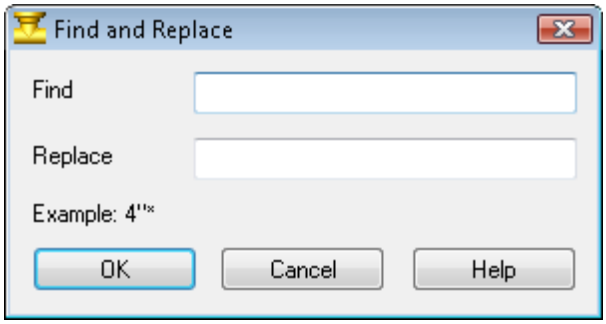
This option cancels any editing, search and replacement or anything else that has be performed.

Find (and Find Next)

This allows searching for a text string such as a grade or material. Utilize the *Find Next* button to find the next occurrence in the specification of the desired search string.

Replace

This option allows the user to search all items in the short and long descriptions for text that needs to be replaced. The following search and replacement dialog box would be presented for the user to fill in the blanks. Please note that the search is case sensitive.



A message indicating the exact number of replacements made (if any) will be displayed.

Delete

Highlight the component and then press this button to delete it. A dialog confirming deletion will appear.

Copy

The *Copy* button will copy a highlighted component entry in the right list box to a new entry in this same list. This button can be used for copying similar entries that might need only slight modifications.

Info

The *Weight* factor is used for adding various margins into component weights. These can include items such as paint and weld material. This number could be increased to 1.03 to indicate paint has been applied to the component. There are various other uses as well.

The *Measure-Bore* is used to tell the program which measurement the specification adheres to. The available options are Imperial-Inch, Metric-Inch, and Metric-Metric. Measure-Bore refers to the length being used in the drawing (inches or millimeter) and the nominal pipe size (either inches or millimeter).

The *Library* sub directory directs the specification to the proper library that the specification uses. This location can be anywhere on the network or on a wide area network.

The *Nominal rating*, *Temperature*, *Pressure*, *Flange face*, and *Materials* are all used as reference when populating information in the symbols in CADWorx P&ID. In CADWorx PLANT, the piping material will be used when transferring information to CAESAR II. Review the Configuration (see page 50) file for more information on these features.

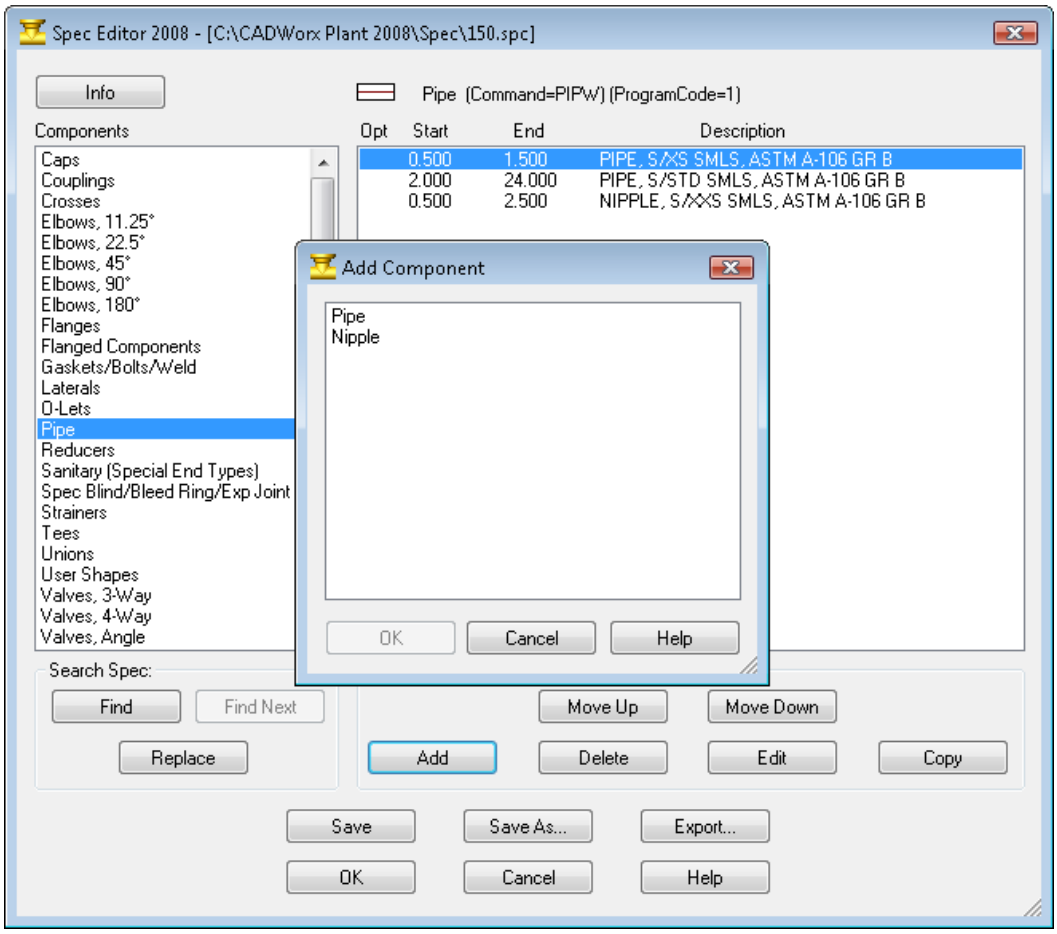
The *Database codes* is used with the DataBaseCodes start up variable. This edit box allows the entry of any user defined database code data file. Please refer to the Edit section (see page 261) regarding the use of database codes.

The *Color* is used as the default color for the specification. The *Select color* button shows the standard AutoCAD color selection dialog.

Note: The Color button is only available when running the specification editor inside of AutoCAD.

Add

When a component category is chosen from the list on the left side of the dialog, individual items belonging to that category are displayed in the list on the right side of the dialog. Items can be added with the *Add* button or by double clicking on the item in the left list. The following dialog lists all the components that are available within the category selected.



Selecting any component in this list box opens the Edit Component dialog discussed next.

Move Up / Move Down

The Move Up and Move Down buttons allows the entries in the spec for a particular type of component to be moved.

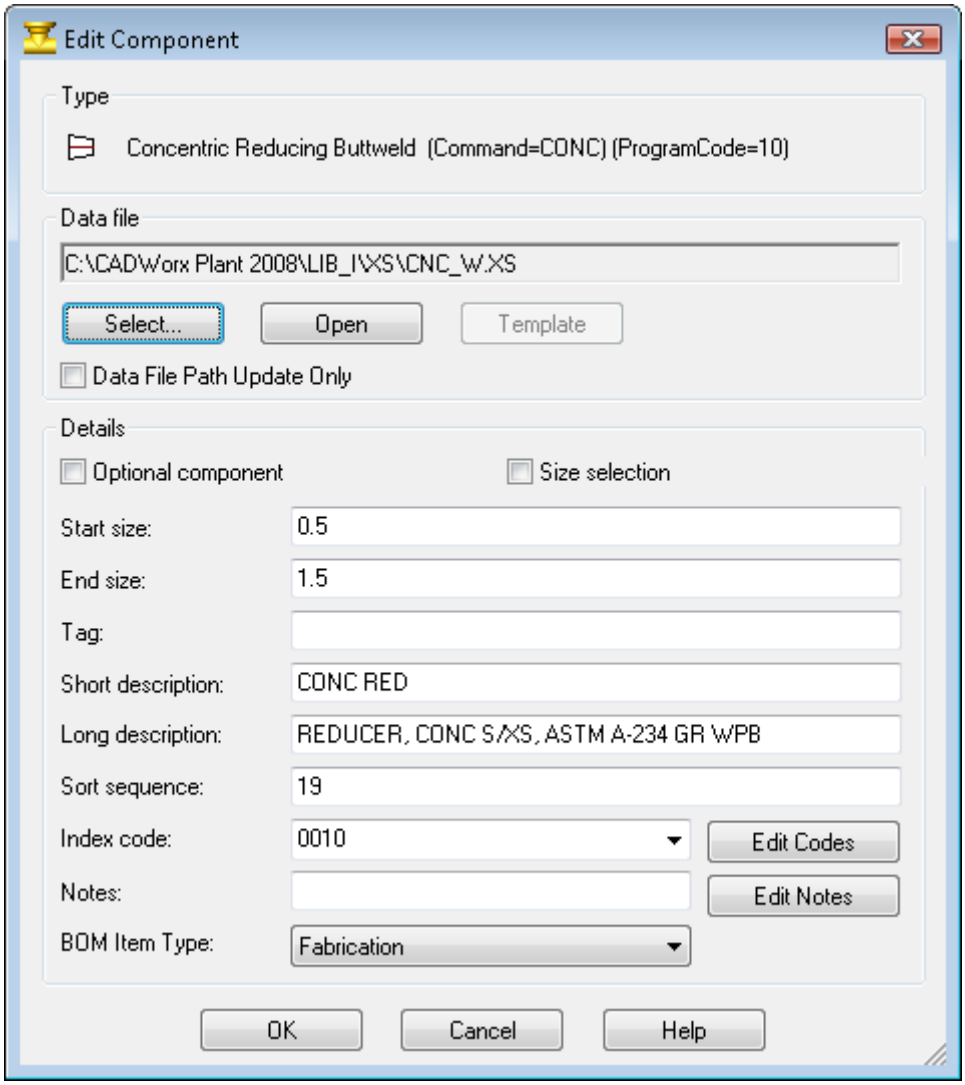
This is especially useful when working with larger number of optional components and using the Specification option toggle (see page 274). The order that the components are listed in the specification file is the order that the components will appear in the optional component dialog.

Edit

This dialog allows modification to any of the components within the specification and appears after adding any new component. After selecting the desired component in the right side list box, utilize the *Edit* button to modify any existing component located within the specification. The *Template* button will be disabled when editing.

If adding a new component with the *Add* button, the dialog appears and will be sparsely filled in. The *Open* button will be disabled when adding.

Note: A component has to be associated with a valid CADWorx data file only when working with CADWorx Plant. It does not require a data file when working with CADWorx P&ID. The data files entries supplied with the specifications in CADWorx P&ID are from CADWorx Plant. Both programs use the same specifications.



Type:

The area at the top of the dialog, *Type*, indicates what type of component is to be edited. Make sure to select the same type of data file when changing data files. If a pipe data file is used to draw an elbow it will cause the program to draw an incorrect shape.

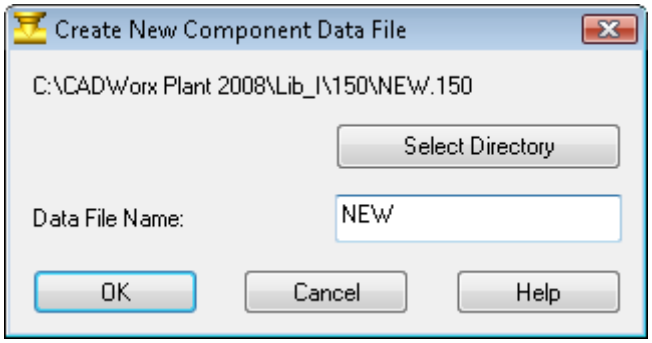
Data file:

The *Select* button provides a file selection dialog for browsing for the data file needed for the component entry. After selecting this button, directories will be presented such as 150, 300, XXS, etc. The user will need to click on the desired sub-directory and then pick a data file. Examples of data file are PIP_W, BAL_F, FLG_F_W, etc. Make sure that the file selected will work in the component's position. If there is a question, generate a template file and compare the one selected with it. After selecting the file, the name will appear above this button.

Note: When a file is highlighted, the title of the file selection dialog will have the files size range in it along with a short description. Another useful feature is right clicking on the data file name and using the open option to view and edit the file. The installation process will associate all CADWorx Plant data files with the NOTEPAD editor. The user can also edit the file in this manner.

The user can pick the *Open* button to review the data file that is present. The data file will open with the associated editor.

With the *Template* button, a data file can be built from scratch. CADWorx PLANT can provide a template data file for which the user can start adding the sizes that they might require. Clicking on the *Template* button displays the dialog below.



A directory has to be selected to store the data file and the name of the data file should be typed in the edit box. The program will automatically add the file extension depending on the name of the data directory. Click *OK* and the data file is opened with Notepad containing sample data required by CADWorx PLANT to draw the component.

The *Data File Update Path Only* toggle will use the file and not it description within the file. When this toggle is not activated, it will populate the long and short description. It will also automatically place the start and ending sizes based on the contents of the file selected. It will be disabled when adding a new component - it will automatically use the descriptions and sizes.

Top Works:

The *Select* button provides a file selection dialog for browsing for the data file needed for valve top works. All top works data file have to be located in the directory set by the TopWorksDirectory (see page 67) configuration variable Sample data files for Actuator, Gear, Lever, Wheel and Custom (TYPE=DWG) are located in the ..\CADWorxPlant\TopWorks\ directory. The date files contain parametric data to draw the top works for the valve. For each type of top works, a sample drawing file is located in the ..\CADWorxPlant\TopWorks\ directory that shows how the data file parameters are used to draw the top works shape. For custom top works (TYPE=DWG), a drawing file will have to be created for each size of top works.

If a top works data file is set for a valve, the user will be asked to orient the top works when the valve is inserted into the model. The top works (spindle) direction selected during the valve insertion will also be set into the ISOGEN Data. The top works will only be drawn when working in 3D drawing mode.

The user can pick the *Open* button to review the data file that is present. The data file will open with the associated editor.

Note: The command TOPWORKSADD can be used to add top works to a valve already drawn in the model The command TOPWORKSCHANGE can be used to change the orientation or delete top works already associated with a valve.

Details:

The *Optional component* toggle marks the component as a optional component in the specification. For more information on optional components, please refer to the Specification option toggle (see page 274) discussed later.

The *Size Selection* toggle will only appear for reducing components. The size ranges listed in the spec for reducing component is the reduction size. For a 10"x2" reducing component, the description in the spec that is listed for the 2" will be used when the component is drawn. This toggle will change reducing components to be selected by main size when drawn in the model. With the toggle checked, a 10"x2" component will take the description listed for a 10" component in the spec.

The *Start size* edit box is the beginning size of a component. For example; if the user wanted this specification to provide 1/2" to 1-1/2" sizes at schedule extra strong pipe and larger sizes to be schedule standard pipe. The user needs to enter 0.5 for 1/2" pipe here. This is the starting size. The ending size will be required in the next edit box. All entries here should be listed as the nominal size, not the actual size.

The *End size* edit box is the ending size of a component. In the above example, the components' size started at 1/2". Here, it can be end at 1-1/2". This provides the component available in sizes 1/2" through 1-1/2". If the user sets a size of 24", and attempts to place this component in the drawing, they would see the Option Toggle dialog.

This *Tag* edit box provides an optional tag description for valves and such.

The *Short description* edit box is the abbreviated description for the component. This description will be used by the Component Annotation (see page 499) routine under the short annotation option within CADWorx.

The *Long description* edit box describes the component. This description can be used in the Bill of Materials schedule and the Component Annotation routine under the long annotation option within CADWorx

The *Sort sequence* edit box locates the item in the bill of material schedule. If the user entered 999, this component would be placed last in the schedule. If it was set at 0, it would be placed first. Setting the same sort numbers to different component in the specification places the components in the schedule in an alphabetical order instead.

ISOGEN Symbol Information:

The default ISOGEN Component Identifier and SKEY used by CADWorx can be overwritten and set from the specification file.

Note: CADWorx P&ID does not process keywords.

Keywords

Keywords can be used in specification descriptions that allow information to be automatically inserted during placement of a component. Keywords can be used anywhere within the Tag, Short Description, and Long Description fields. Keywords can also be used in the Index Code data file.

The keywords are:
{MAIN} {MAINACT} {RED} {REDACT} {MAINTHK} {REDTHK} {LENGTH} {SPEC} {BORE} {PIPETHK} {PIPETHKRED}

Four additional keywords are also available if main and reduction sizes are needed in decimal format. These four additional keywords are:
{MAINDEC} {MAINACTDEC} {REDDEC} {REDACTDEC}

The keywords {BORE}, {PIPETHK} and {PIPETHKRED} are calculated from the pipe data file in the specification instead of the component's data file. For example, a 4"x1" reducer, {PIPETHK} would get the values from the 4" pipe data file as listed in the specification. {PIPETHKRED} would get values from the 1" pipe data file. A 4"x1" reducer with {MAINTHK} or {REDTHK} would get the keyword values from the reducer data file as listed in the specification. Olet data files do not have any thickness values listed. Olet {MAINTHK} or {REDTHK} values are always read from the appropriate pipe data file.

The {BORE} keyword is calculated using the following formula: {BORE} = [(Pipe Outside Diameter of Main Size) - (2*Pipe Wall Thickness of Main Size)]. { BORE} keyword is a calculation of the inside diameter of the main pipe size.

Example keywords usage:

- PIPE, {MAIN} {MAINTHK}" API-5L GR.B DSAW, PSL2
- FLANGE, {MAIN} ANSI 150 RFWN {BORE}" BORE ASTM A105, ASME B16.5
- NIPPLE, PIPE {MAIN} x {LENGTH} XS ASTM A106 GR.B
- WELDOLET, {MAIN} {MAINTHK}" x {RED} {REDTHK}" ASTM A105, ASME B31.8
- VALVE, PLUG {MAIN} ANSI 150 WE API 6D TWT {PIPETHK}" API-5L GR.B PIPE REG PAT LO
- GASKET, INSULATING KIT {MAIN} ANSI 600, RF, B={BORE}, PIKOTEK VCS WITH ZINC PLATED WASHERS, GRE INSULATING WASHERS AND SLEEVES
- BOLT, STUD {MAIN} x {LENGTH} ALLOY STEEL ASTM A193 CL 2A GR B7 W/2 HEX NUTS ASTM A194 CL 2B GR 2H (FOR {MAINDEC}" 150# FLG.)

Note: When running SPECCHG (see page 271) using a spec with keywords, the user must mode convert (see page 295) components with keywords. Mode convert will replace the keyword in the description with the appropriate values.

User Defined Keywords (Keyword.dat file):

User defined keywords can be created using the Keyword.dat file located in the specification directory. The top the Keyword.dat file explains how to use this file.

Keyword Formatting:

The numeric values are formatted based on SystemMeasure (see page 66) in use. The tables below has an explanation of the different formatting for each of the keywords. The AlphaSizeControl (see page 50) is read from the current configuration file.

Keyword	SystemMeasure - English/Inch
MAIN, RED	AlphaSizeControl
MAINACT, REDACT	AlphaSizeControl
MAINDEC, REDDEC	Always 3 decimal places
MAINACTDEC, REDACTDEC	Always 3 decimal places
MAINTHK, REDTHK	Always 3 decimal places
BORE, PIPETHK, PIPETHKRED	Always 3 decimal places
LENGTH	Based on DIMLUNIT, DIMDEC, DIMLFAC
SPEC	Specification filename without extension

Keyword	SystemMeasure - English/Inch
MAIN, RED	AlphaSizeControl
MAINACT, REDACT	LUPREC and DIMZIN control number of decimals
MAINDEC, REDDEC	Always 3 decimal places
MAINACTDEC, REDACTDEC	Always 1 decimal places
MAINTHK, REDTHK	Always 1 decimal places
BORE, PIPETHK, PIPETHKRED	Always 1 decimal places
LENGTH	Based on DIMLUNIT, DIMDEC, DIMLFAC
SPEC	Specification filename without extension

Keyword	SystemMeasure - Metric/Metric
MAIN, RED	AlphaSizeControl
MAINACT, REDACT	AlphaSizeControl
MAINDEC, REDDEC	Always 1 decimal places

MAINACTDEC, REDACTDEC	Always 1 decimal places
MAINTHK, REDTHK	Always 1 decimal places
BORE, PIPETHK, PIPETHKRED	Always 1 decimal places
LENGTH	Based on DIMLUNIT, DIMDEC, DIMLFAC
SPEC	Specification filename without extension

Note: CADWorx P&ID does not process keywords.

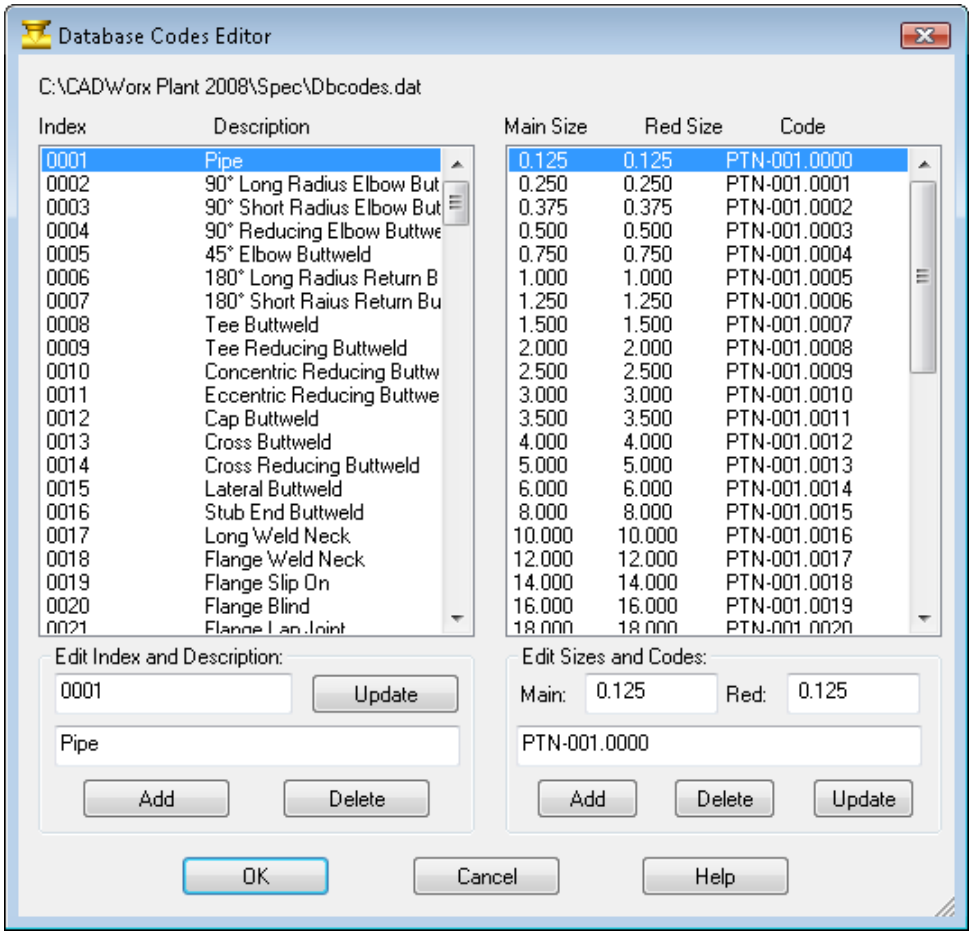
Index Codes

This code is the index (or pointer) for the generation of database codes placed within components. When a component is placed in the drawing, the system looks at this code and tries to find a corresponding index in the file named by the DataBaseCode variable in the configuration file. It can also use the DataBaseCodes file as specified in the specification.

For example – within the specification, this code is set to “0001” for categories of pipe from 2.0 - 24.0 – the DataBaseCode startup variable is set to DBCODES.DAT and within this file there exists an index of “0001” – under this index there are sizes 2.0 through 24.0 and the 4.0 has a code of “76.35.38.237”.

When a 4” pipe is drawn, it will have attached the code 76.35.38.237. If the index was not found within the DBCODES.DAT, then “0001” would be placed in the component. For more information refer to the DataBaseCodes (see page 55) variable in the Configuration file.

To set an index for a specification component, simply highlight an index in the left list box and select OK at the bottom of the dialog. This will place the index code in the edit box on the edit dialog. Whenever a component is drawn on the drawing, it looks at this index, searches for the proper size and finds the corresponding code to place in the component.



Note: Separate DBCODES.DAT files are not necessary when creating index, size and code combinations. Different specifications can all access the same index in the same file. A carbon steel 150-pound flange might have the same order number or database code across several specifications.

The left list box shows all the available indexes in DBCODES.DAT (or a name as specified by DataBaseCode (see page 55) variable in setup). To set this index, and corresponding size and code, to a component, highlight it here and select the OK at the bottom of this dialog.

The Edit Index and Description section allows the user to change an index for a range of sizes. Simply highlight the index, change it's value and hit the update button. Changing focus will also update the list box above. The edit box below this allows the user to enter a description for the index that was entered above. The Add button allows the user to add any index desired. Just click the Add button and a generic entry for the index, pipe size and code will be added. Use the edit boxes above to edit and change this entry. The Delete button index located within the code file can be removed with this button.

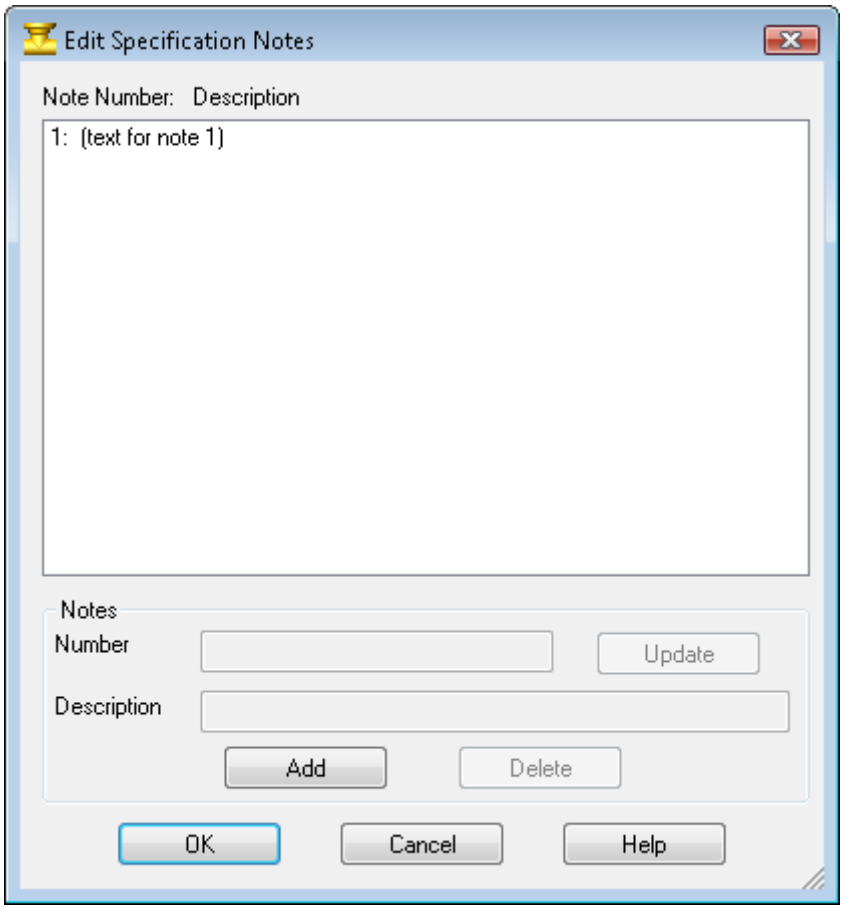
The right list box shows all the sizes and codes associated with an index. Simply highlight the size and then edit it as desired.

The Edit Sizes and Codes section can be changed to any size desired. Make sure that the sizes entered here are the same available size located within the SETSIZE.TBL, SETSIZEA.TBL, and SETSIZEM.TBL.

The Main and Red edit box allows the modification of any code that is associated to any size. The Add button allows the user to add any size – code combination desired. Just click the Add button and a generic entry for the size – code combination. The Delete button simply deletes any entry in the size – code combination located in the list box above. After an entry has been modified in the edit boxes, just hit the Update button to update the list box above. Changing focus will also accomplish the same thing.

Notes

This edit facility allows the user to add notes to any component that is in the specification. Notes can be added and used in CADWorx P&ID utilizing the COMPNOTE column name when using a database. Multiple notes can be applied to a single component and when that component is associated with a symbol in CADWorx P&ID, the notes will be applied to that column separated by a semicolon.



The *Note Number* Description list box contains all the notes present within the current specification.

The *Notes* section allows the user to modify any note number highlighted. This entry can be alphanumeric. After changing the number, simply hit the update button to change it in the list above.

The *Update* button will update the number and the description after a modification has been made. The Description edit box allows the user to modify any description located within the notes. After a modification has been made, use the update button to change the list box above.

Clicking on the *Add* button will add a generic entry into the specification. This entry can be modified as desired.

The *Delete* button will delete any note located within the list.

Change

Plant menu: ▶ **Accessory** ▶ **Specifications** ▶ **Change**

Command line: SPECCHG

This function provides the capability to change the specification used in components already placed in a drawing. For example, if a AA1 specification was used for components placed in the drawing, this function could change all the components to a different specification like BA1SS.

Caution should be used when changing a 150lb based specification to a 300lb based specification. This function will change the associated data file and the next mode conversion will show a problem of over lapping components due to lengths that are different in the two specifications. A 300lb RFWN flange is longer than a 150lb RFWN flange.

Note: When running SPECCHG using a spec with keywords (see page 261), the user must mode convert (see page 295) components with keywords. Mode convert will replace the keyword in the description with the appropriate values.

Command: **SPECCHG**

Enter an option **[Filter spec/Line number/Select components]** <Select components>:
Select objects: Other corner: 340 found
Select objects: *enter to finish selection*

Filter spec

This option allows the user to filter the drawing by selecting one or more specifications used in the drawing.

For example, if the user wished to chang all components in 150 and 300 specifications to 600 specification, this option will allow them to do so.

[**List/Select component**] <Select component>:

List

See Appendix F: Common List Dialog (see page 647) for more information about this option.

Select component

This option allows the user to select a component with an existing spec. Select the desired component by crossing, window or any other preferred method.

Select a CADWorx entity: *select any CADWorx entity*
Accessing entities with spec of "150"

Once the entity is selected, it's spec will be displayed in the command line as shown above.

Line Number

This option allows the user to select a component with an existing line number or lists all line numbers in the model.

Enter an option [**List/Select component**] <Select component>:

Once the line number(s) are selected by either one of the above methods, the program accesses all the components with matching line numbers.

List

See Appendix F: Common List Dialog (see page 647) for more information about this option.

Select component

This option allows the user to select a component with an existing line number. Select the desired component by crossing, window, or any other preferred method.

Select a CADWorx entity: *select any CADWorx entity*
Accessing entities with line number of "CS1-6"-305"

Once the entity is selected, it's line number will be displayed in the command line as shown above.

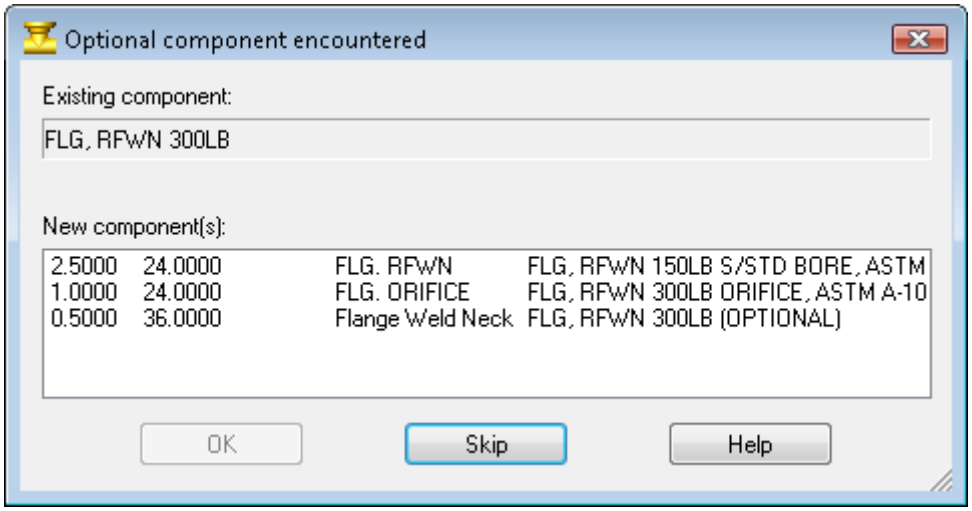
Select component

This option allows the user to select the desired components by crossing, window or any other preferred method.

Once the components are selected, CADWorx Plant changes their specification and informs the user.

Changed 58 components to 150 spec

If an optional component is selected for specification change, the following dialog will be displayed prompting the user to pick the type of component from the new specification or skip the component.



The dialog lists all the available components including optional components of the same type in the new specification. If there are no similar type components available in the new specification, the list will contain the message "None Available" and user will be able to skip the component.

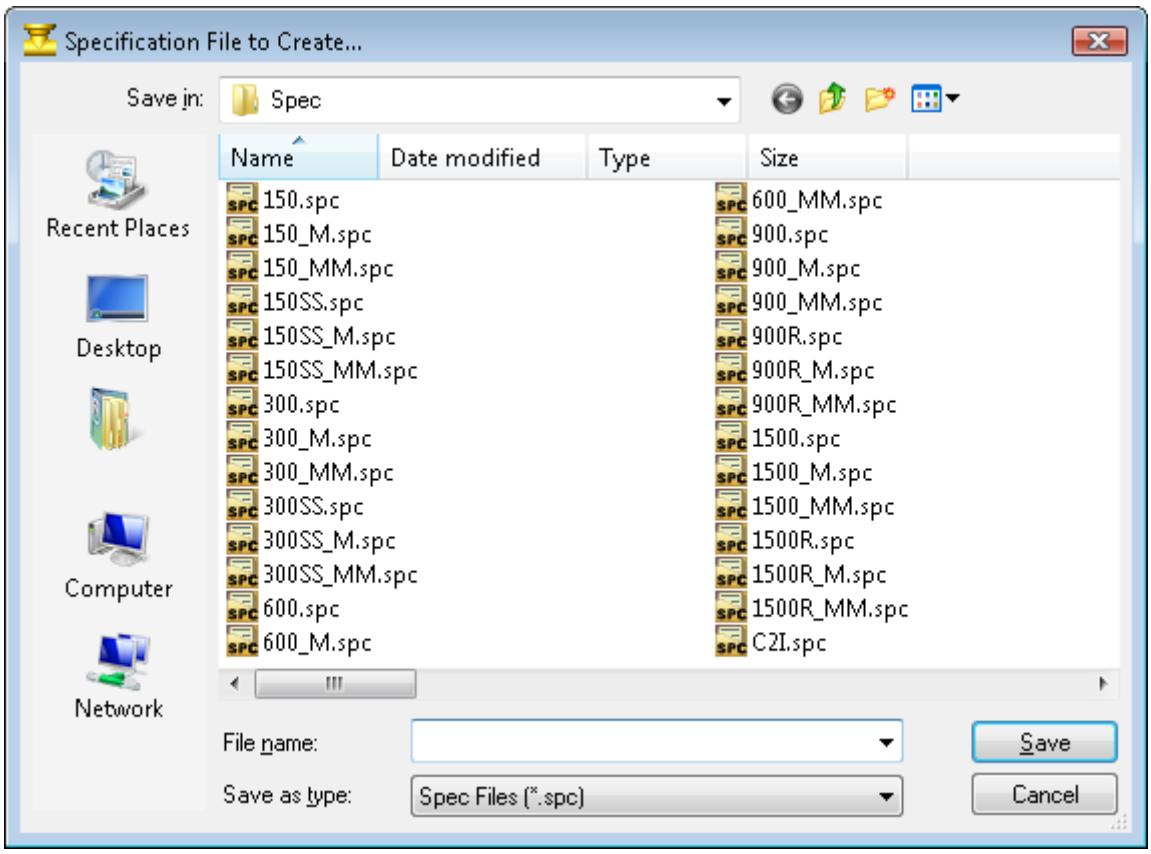
Blank

Plant menu: ➤ **Accessory** ➤ **Specifications** ➤ **Blank**

Command line: SPECBLANK

This function will display the following dialog and allow the user to create a new blank specification. After supplying a name and clicking on the *Save* button, the Specification Editor as discussed earlier will be provided to allow modifications to the specification. The specification will be completely empty and the user will be required to use the *Add* button in the editor to add all the required entries. This function is very useful when only a few components are required in a specification such as a lined or possibly a PVC specification.

Command: **SPECBLANK**



Option Items

Plant menu: ▶ **Accessory** ▶ **Specifications** ▶ **Optional Items**

Command line: SPECOPTION

Settings toolbar:

This function has two levels.

- Level 1:

  When a component type is selected, all components in the specification of that component type will appear in this dialog box.

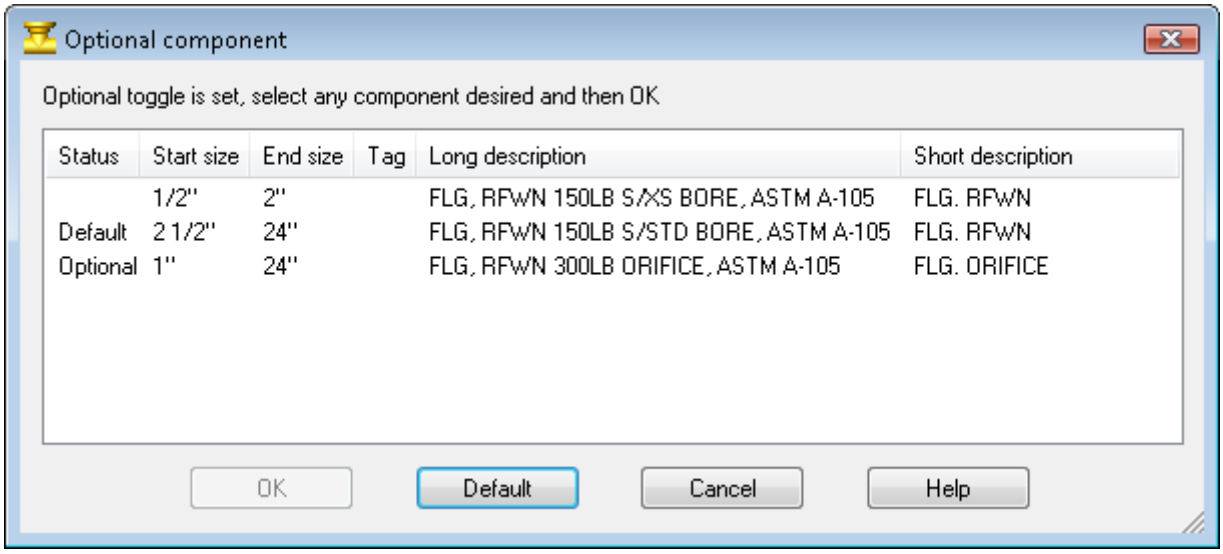
  Command: **SPECOPTION**
  All components will be dialogued...

- Level 2:

  When a component type is selected, the default component will be selected for placement on the drawing. No dialog options will appear unless the components' size is out of the listed range in the specification or the component is not allowed by the specification.

  Command: **SPECOPTION**
  Specification dialogue turned off...

This function provides the ability to place components into the drawing that are located in the specification as optional components. A good example would be a 300# orifice flange in the 150# specification.



When all components are dialoged, note that the specification's default component is labeled. This component can be chosen by selecting the default component and then picking the OK button (or by simply picking the Default button). All components can be dialoged and the user can easily pick the default components the same way as if the system was off.

When the SpecSizeOverride (see page 63) variable is off(0), the system will not allow any size other than what is allowed by the specification. When this variable is off, and the size selected is not within the range listed within the specification, the OK and Default button will be disabled and will not allow selection of any component.

This dialog also appears when a component is not in the specification whether the option toggle is on or off.

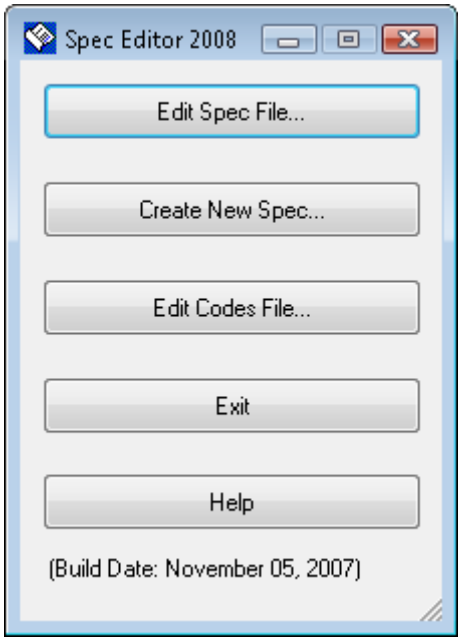
If SpecSizeOverride is on, the dialog will appear if the component size is not in the range thus allowing the user to select the desired component and range. Below is a table explaining what happens according to how the system is set.

Component in specification	**Size in specificatio n**	**Specification option toggle**	**SpecSizeOverride variable**	**Dialog behavior**
Yes	Yes	No	Off = 0	Normal operation. No errors.
Yes	No	No	Off = 0	Dialog with error message.
Yes	No	No	On = 1	Dialog with error message, but program allows override.
Yes	Yes	Yes	Off = 0	Dialog shows all components in optional components, but does n allow override.
Yes	No	Yes	Off = 0	Dialog shows all components bu button is disabled for componen in size range.
Yes	No	Yes	On = 1	Dialog shows all components ar allows override.
No	n/a	No	Off = 0	Dialog with error message.

Stand Alone Spec Editor

This dialog allows the user to execute the specification editor outside the AutoCAD environment. Just double click on the program (SpecEdit.exe) in the explorer environment or start it from the CADWorx Plant program group. This editor can be distributed freely and can be used by any administrator or manager who needs to create, modify, or care for the program's specifications.

This executable can be located in the specification directory along with the specifications. The program will search the local directory and open any specifications that it finds. If there are not any specifications found in the local directory, the program retrieves the SpecificationDirectory (see page 63) variable location from the configuration file and opens in that directory.

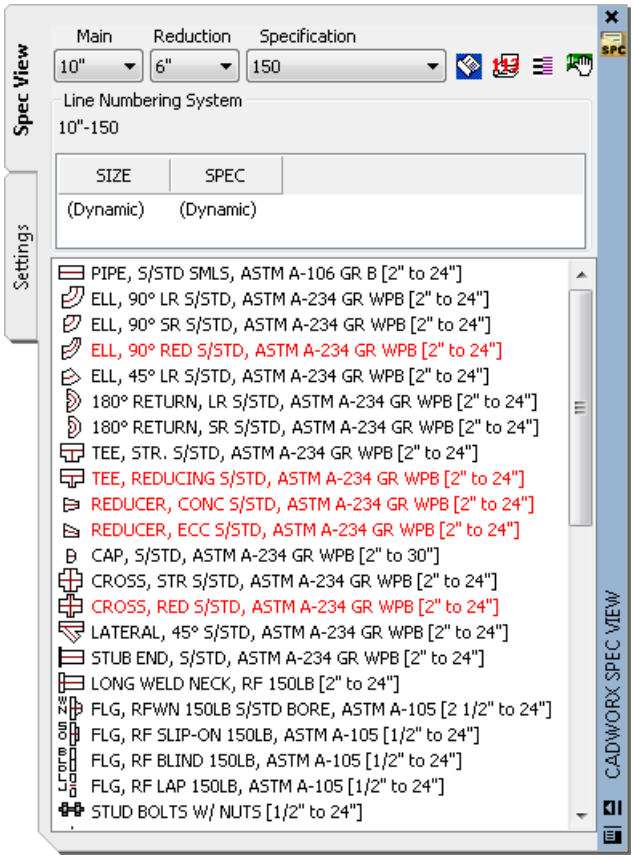


After a specification is opened, it uses the Specification Editor in CADWorx Plant.

SpecView Palette

CADWorx Plant components are typically drawn using toolbars, or by typing the command name at the AutoCAD command prompt. The SpecView palette provides an alternate method of inserting CADWorx components. SpecView will filter the available components based on the size and specification settings. In addition, SpecView also provides a quick way to change the line number settings.

SpecView Tab:



Three drop down lists are provided at the top of the palette to set the *Main* size*, Reduction* size*,* and *Specification*. The specification drop down list is based on the specification directory (see page 63) setting in the current configuration file.

Five toolbar buttons are also provided for quick access to certain features.

- *First button* will start the Specification Editor (see page 254).
- *Second button* will bring up the Line Number Setup dialog (see page 283).
- *Third button* will bring up a dialog to create the drop down values for the line number system. This dialog can be used to add values to the drop down list that appears with the line numbering system. These values will be stored in a file called DropDownList.txt. This DropDownList.txt file will be located in the same directory as the current configuration file.
- *Fourth button* will run the CADWorx command COMP2ALL. (see page 300)
- *Fifth button* will bring up the help file.

To draw Pipe components, use the following options:

- *Left mouse click:* Draws buttweld pipe.
- *Right mouse click:* Displays a menu with options to draw buttweld, socket weld, or threaded pipe.

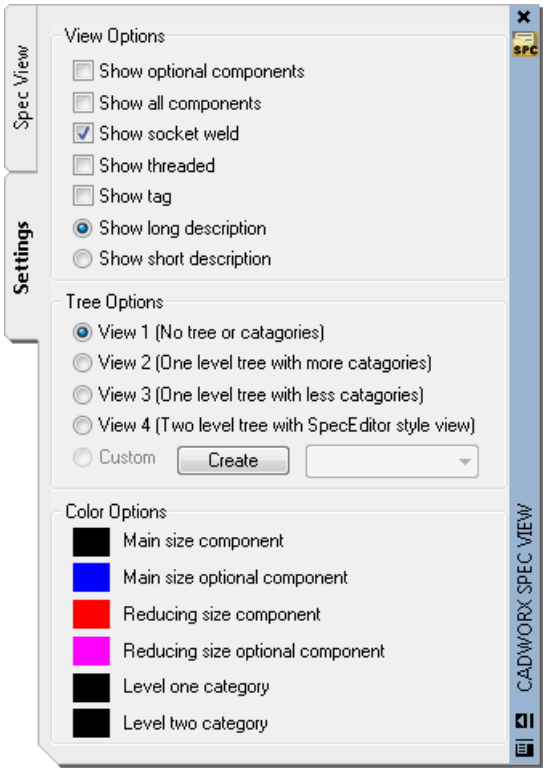


Top draw non-pipe components, use the following options:

- *Left mouse click:* Draws the component in plan view based on the current UCS.
- *Right mouse click:* Displays a menu that will allow other insert options available for the component. Not all components will have all four options as shown in the image below.



Settings Tab:



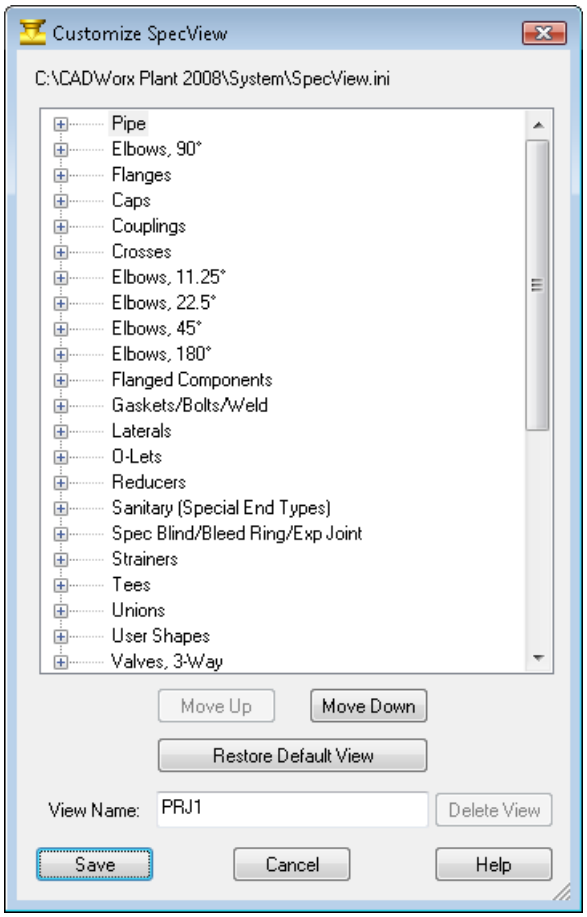
The *Settings* tab of the pallet allows you to customize the appearance of the SpecView.

View Options:

- *Show optional components* check box: Displays optional components in SpecView. With this setting on, always use the SpecView dialog to draw components. If you issue the component command from outside of SpecView (Command Line Entry, space bar to run last command, or toolbar), CADWorx will use the default component based on the current size setting. This may not be the same component that was selected in SpecView.
- *Show all components* check box: Displays all components listed in the specification file regardless of the current size set. With this setting on, always use the SpecView dialog to draw components. If you issue the component command from outside of SpecView (Command Line Entry, space bar to run last command, or toolbar), CADWorx will use the default component based on the current size setting. This may not be the same component that was selected in SpecView.
- *Show socket weld* check box: Displays socket weld components in SpecView.
- *Show threaded* check box: Displays threaded components in SpecView.
- *Show tag* check box: Displays tag in SpecView.
- *Show long description* radio button: Displays long descriptions in SpecView.
- *Show short description* radio button: Displays short descriptions in SpecView.

Tree Options:

- *View 1 (No tree or categories):* This view will list components in numerical order based on the Sort Sequence set for the component in the specification file.
- V*iew 2 (One level tree with more categories):* This view will create a category for each type of component. For example, buttweld and socket weld tees will be listed in separate categories.
- *View 3 (One level tree with less categories):* This view will create less categories than *View 2*. For example, all tees will be listed together in one category.
- *View 4 (Two level tree with Spec Editor style view):* This view is a two level tree that is arranged in the same sequence as the Spec Editor application.
- *Custom:* This setting will allow you to create a customized view. If a view has not been created, click the *Create* button. This will display the Customize SpecView dialog which will allow you to create a new view. All customized views will start with *View 3* as the default. To select an existing SpecView configuration, use the drop down list.



Move Up/Move Down: These buttons will allow you to move a category, or an individual item inside a category.

Restore Default View. This button will restore the tree to the default (View 3).

View Name: This box will have the name of the current SpecView.

Delete View: This button will allow you to delete a SpecView.

Save: This button will close the dialog and save any changes made to the SpecView configuration.

Color Options will control the color displayed at various levels of the tree.

Modify Library

Plant menu: ▶**Accessory** ▶**Modify Library**

Command line: MODIFYLIB

Modify Library (from the pulldown under Accessory/Modify Library) modifies data files while in the drawing environment. If the user selects a size that is not in the component's data file they will receive the following error message.

Size <size> is not available in current libraries...
Edit last library file - <INSTALL-DIR>\LIB_I\S40\PIP.S40 <Yes>:

At this point the user can enter **Y** (for yes) to edit or view this file. After viewing the file and discovering that the size is not available, they can use the Library Modification routine.

Command: **MODIFYLIB**
Modify previous component library file? **[Yes/No]** <Yes>: *hit enter or type* **N** *for no*

Answering N (for no) will display the standard file selection dialog. A data file will be required from this dialog. The first available listing will be the different directories (150, 300, etc.) that are supplied with CADWorx Plant. Selecting one of these directories will show various data files (flg_w.150, gat_f.300, etc.) within this directory for the user to select from.

[**Append/Edit**] a file <Edit>:

Append

This option appends the existing data file. The new data will be placed at the end of the file. Even if the size was 1/2", it would be placed at the end following a 24". The following prompts will change depending on which file is being appended. The prompts are the same as the descriptions in the column header of the data files. See sample of data files below.

Modifying...PIPE, SCH 40 SMLS

NOM? *enter real, nominal diameter*
OD? *enter real, outside diameter*
THRD_ENG? *enter real, thread engagement*
SOC_ENG? *enter real, socket engagement*
WGT? *enter real, weight*
THK? *enter real, thickness*

Are above entries OK? [Yes/No] <No>: *hit enter or type N for no*

This prompt allows the user to change the entries. Answering N (for no) will re-cycle the routine and present the data file prompts again. Answering Y (for yes) will end the routine and return the user to the command line.

Edit

This option takes the user straight into their editor. The user can then freely modify the file as they wish. The user can save the file with the same name or give it another.

Line Numbers

This system performs various line number functions as listed below. Line numbers can be created and stored in different configuration files for various user and customer requirements in a variety of formats. There can be any number of categories within a line number. When starting a new drawing the line number system will not be active. The user should use the NUMBERSETUP command to have components inserted to have a line number. Once the line number is set in the current drawing, it will remain set indefinitely.

Shown below is the line number toolbar. Located on this toolbar are functions to set the line number properties, assign existing components a line number, place annotation on the drawing, and set the line number from an existing source. There are also buttons to set, increment, and decrement the count.



Assign

Plant menu: ▶ **Accessory** ▶ **Line Numbers** ▶ **Assign**

Command line: NUMBERASSIGN

Line Numbers toolbar: 

This routine allows the user to assign new or different line numbers to existing components. In the static mode, the line number will be replaced exactly as it is pre-viewed in the setup dialog. This means that a 3/4” line can actually be assigned a 8”-150-2001-N line number. When in dynamic mode, the line number is automatically adjusted and the 3/4” line would receive a line number of 3/4”-150-2001-N. This is based upon the present size and specification listed within the components data.

When selecting a component with a line number already attached, a message will appear indicating such. By typing “Y”, (for yes) the component will be assigned the new line number based upon a static or dynamic nature as explained above. Selection can be made by window, crossing or by individually selecting component. During the process, a rubber-banded cursor appears indicating which component is being assigned the new line number.

Command: **NUMBERASSIGN**
Current line number is 4"-150-HWS-3001-N
Select objects: *select all the components requiring a new number*
Select objects: *enter to finish*

As mentioned above, if the component already has a line number, a prompt appears confirming replacement.

Component has existing line number of 4"-150-HWS-3001-N
Replace with line number of 2"-150-HWS-3001-N
Enter an option **[No/Yes/yes to All]** <Yes>:

No

This option will not replace the line number within the selected component.

Yes

This option will replace the line number within the currently selected component.

yes to All

This option skips all future prompts and replaces the line number within all the currently selected components.

Annotate

Plant menu: ▶ **Accessory** ▶ **Line Numbers** ▶ **Annotate**

Command line: NUMBERANNOTATE

Line Numbers toolbar: 

This routine labels components with existing line numbers. The text size is based upon the AutoCAD environment variable TEXTSIZE and is placed on the default text layer. Individual components can be annotated by selection. During the labeling process, a rubber-banded cursor appears indicating which component is being labeled.

Command: **NUMBERANNOTATE**
Select component to annotate line number: *select component*
Start point or **[Leader/Justify]**: *pick point*
Rotation angle <0>: *pick direction or enter real*

Set

Plant menu: ▶ **Accessory** ▶ **Line Numbers** ▶ **Set**

Command line: NUMBERSET

Line Numbers toolbar: 

This function allows the line number to be set from three different sources. The line number can be set from a P&ID project, the current drawing or by selecting a component.

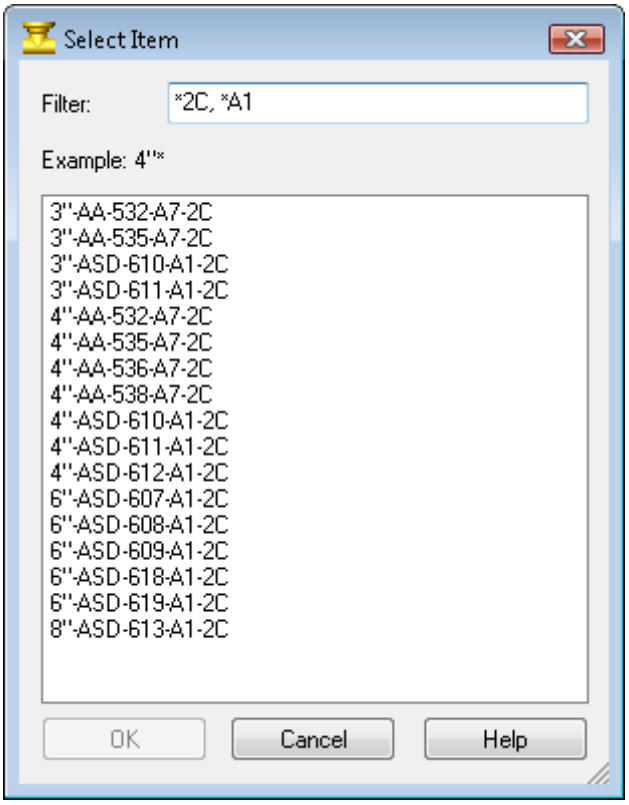
The first source is requires a valid CADWorx P&ID project using an Access database. If the configuration file being used is the same as an existing P&ID project, this function automatically supplies all the line numbers within that project from the process lines table. The line number supplied will be of the same format as the project.

The second source is from the current drawing. The third source allows the user to select a component within the drawing to set the line number. The last two sources require that the LineNumberSystem (see page 60) be set to 2. This is the same as having the System ON and the Static (Size/Spec) toggle in the setup dialog turned on.

Command: **NUMBERSET**
Enter an option **[Process drawings/current Drawing/Component]** <Component>:

Process drawings

This option allows line numbers to be set from a P&ID project. If the configuration file used in the CADWorx Plant environment is the same configuration file used in the P&ID project, then this function will automatically find the project database and automatically present the line numbers from that P&ID project as shown in the following dialog. If the configuration file is not the same, a standard file dialog will be presented requesting the location of the database to be used.



The Filter at the top of this dialog allows the user to type in a part of the line number to show only those line numbers (i.e. 4" would show all the 4" line numbers within the table).

This function only works with a Microsoft Access database. The following prompt will appear if the project chosen is not an Access database.

This function only supports Access databases at this time...

current Drawing

This option allows the user to select a line number from the current drawing environment. It will search the entire drawing and present all the line numbers in a dialog. For this option to work, the LineNumberSystem (see page 60) must be set to 2, or a prompt will indicate a problem.

Can be used only when LineNumberSystem = 2 (Static)

Component

This option allows the user to select a component within the current drawing to set the current line number. If the LineNumberSystem (see page 60) is not set to 2, a prompt indicating this will appear as in the option above.

Count

Plant menu: ▶ **Accessory** ▶ **Line Numbers** ▶ **Count**

Command line: NUMBERCOUNT

Line Numbers toolbar: 113

This function sets the present line number count value. The count must be chosen from the available fields discussed above.

Command: **NUMBERCOUNT**
Enter new line count value: enter real (ex: 0001)

Line number count set to 0001
Current line number is 4"-150-AEC-0001-N

This function requires that the LineNumberSystem (see page 60) be set to 1. If not, an error will appear. This is the same as having the System ON toggle turned on and Static (Size/Spec) toggle turned off in the NUMBERSETUP (see page 289) dialog.

Can be used only when LineNumberSystem = 1 (Dynamic)

Increment

Plant menu: ▶ **Accessory** ▶ **Line Numbers** ▶ **Increment**

Command line: NUMBER+

Line Numbers toolbar: +

This function increments the line number count value. If the user set 0001 for the line count value above, then this would increment it to 0002.

Command: **NUMBER+**

Line number count incremented to 0002
Current line number is 4"-150-AEC-0002-N

Decrement

Plant menu: ▶ **Accessory** ▶ **Line Numbers** ▶ **Decrement**

Command line: NUMBER-

Line Numbers toolbar:

This function decrements the line number count value. If the user set 0002 for the line count value above, this would decrement it to 0001.

Command: **NUMBER-**

Line number count incremented to 0001
Current line number is 4"-150-AEC-0001-N

Setup

Plant menu: Accessory Line Numbers Setup

Command line: NUMBERSETUP

Line Numbers toolbar: 

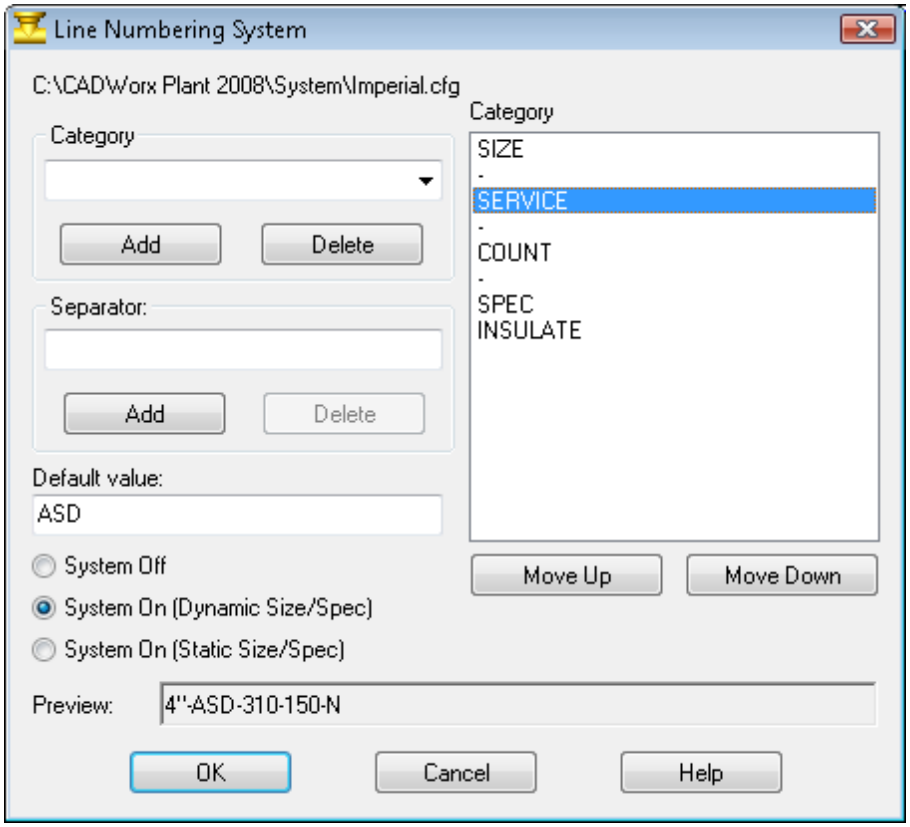
The first function on this toolbar is the NUMBERSETUP function. This function allows the user to modify any line number configuration. There are three levels at which the line numbering system can operate. The system can be turned on or off. While on, it can operate in a static or dynamic mode.

In the dynamic mode, CADWorx Plant adjusts the line number according to the size and/or specification. Whenever the size or specification is changed, the line number will change accordingly. In the static mode, the user must manually change the size and specification. Static is useful when all components and sizes of a particular piping run are required to have the exact same line number (i.e. the 3/4" vent on the 8" process line needs a line number of 8"-150-2001-N). Static line numbering is also useful when selecting runs of piping for other functions with CADWorx Plant such as the automatic isometric generator. Also, see the LineNumberSystem (see page 60) variable in the configuration file for more information.

This system also works with a CADWorx P&ID setup configuration file. The user can change the configuration file location in SETUP to use the same project configuration as any P&ID project. The Add button under the Category group in this dialog allows the selection of any of the additional fields located in the process lines table of a typical P&ID project database. When using a P&ID project, changes made to a line number configuration within CADWorx Plant will reflect within P&ID project.

If the system is not using a CADWorx P&ID setup configuration file, the additional fields are obtained from the DATABASE.TBL file located in the <INSTALL-DIR>\SYSTEM directory. The user will have to manually edit this file for any modifications required of the existing fields within the default line number system.

Command: **NUMBERSETUP**



File

At the top of the dialog is the name of the configuration file that is currently being used. As mentioned above, it may also use a P&ID project configuration file.

Category

The user can view all process line items located within the DATABASE.TBL file in the <INSTALL-DIR>\SYSTEM directory by selecting the *Category* drop down list box. These items can be added to any line number system. Once an item has been selected, use the Add button to place it in the list box on the right. The user can also highlight the item in the right list box and then click the Remove button to delete it from the line number system. The user can also type in a value and click the add button.

Separator

The *Separator* group also has an Add and Remove button which works very similar to the buttons above. Any alphanumeric separator can be added to the system (i.e. "-", "~", "#", space, etc.).

Default value

The *Default* value box allows the user to add any default value to any category of the line number system (a value for each category in the right list box). The size and specification will be disabled if the system is set to the dynamic mode.

System On (Dynamic Size/Spec)

The *System On (Dynamic Size/Spec)* toggle turns the system on or off. Each component that is placed in the drawing receives a line number when this toggle is on. The line number will not be placed in the component when this system is off.

System On (Static Size/Spec)

The *System On (Static Size/Spec)* toggle notifies the line number system to use what has been typed into the Default value box including the size and specification. This allows the user to type anything for the size and specification.

Preview

The *Preview* box shows what the line number looks like based on the categories selected. The line number added to components will look similar to the preview value.

Move Up/Move Down

The *Move Up* and *Move Down* button below the list box allows the user to select any category within list box and relocate them within the line number. Use the preview to determine the final constructed line number system.

Auto Route

Auto Route

Auto Route

Plant menu: ▶ Accessory ▶ Auto Route ▶ Buttweld LR, Buttweld SR, Threaded and Socketweld

Command line: ROUTEBWLR (ROUTEBWSR, ROUTETHRD, **&** ROUTESW)

The auto route commands in CADWorx Plant provide a quick and effective way of drawing pipe. The user can draw long radius, short radius, threaded, or socket weld pipe. All commands have the same options.

Command: **ROUTEBWLR (ROUTEBWSR**, **ROUTETHRD**, & **ROUTESW)**
Enter an option **[Elbows only/Pick points/Select polyline]** <Select polyline>:

Elbows only

This option installs only elbows on a particular pipe run. The user will be asked to select a starting location, first elbow point, next elbow point, etc., last elbow point, then finally, the ending location. An elbow will not be placed at the starting and ending locations. This routine only allows picks and not polyline selection.

Hit enter to end....
Pick first point: *pick first point, not an elbow location*
To point: *pick next point, for the first elbow*
To point: *...*
To point: *pick next point, for the last elbow*
To point: *pick last point, not an elbow location*
To point: *enter to end*

Pick points

This option installs pipe and elbows on a particular pipe run. As above, in the elbow only option, the user will be asked to pick a starting location, first elbow point, next elbow point, etc., last elbow point, then the ending location. Pipe and elbows will be drawn along these construction picks. This option only allows picks and not polyline selection.

Hit enter to end....
Pick first point: *pick first point, not an elbow location*
To point: *pick next point, for the first elbow*
To point: *...*
To point: *pick next point, for the last elbow*
To point: *pick last point, not an elbow location*
To point: *enter to end*

Select polyline

This option allows the user to select a polyline or 3Dpoly to build pipe and elbows. Use the router function below for 3D polyline construction.

Select polyline or 3dpoly: *pick polyline*

This function will also provide rolling offsets.

Router

Plant menu: ▶ **Accessory** ▶ **Auto Route** ▶ **Router**

Command line: ROUTER

This function provides a 3dpoly line for the previous welded, threaded and socket automatic routing routines. It will allow the user to draw the 3dpoly line with different options. It also provides an undo feature similar to the AutoCAD's 3dpoly command. If the user is supplying known 3D points, they should make sure the UCS is at the world position (type UCS and hit enter).

Command: **ROUTER**
Specify start point or **[Position/Reference]**:

The above prompt is the first point to be selected. Unlike a normal 3dpoly line, the elevation must be controlled with the Elev(ation) option or the UCS position. The user cannot pick a point on top of an object 10' high with the UCS at the world position. The 3dpoly would start at the 0' elevation. Also, the Slope and Position options will not take affect until the next point is picked. In other words, use the Slope option, then pick the next point of the 3dpoly for a change to the new position. For more details, see below.

Specify next point or **[Slope/Position/Elevation/Undo/Reference]**:

Slope

This option will not take effect until the next point is picked. It allows the entry of a known slope or drop per foot.

Enter slope in degrees (+ or -) or [**Meter**/**Foot**/**Drop**/**Rise**/**Continuous**]:

Meter

This option allows the user to enter a rise (or drop) per meter.

Enter rise in mm (+ or -) per meter: *enter real (ex: -3)*

Foot

This option allows the user to enter a rise (or drop) per foot.

Enter rise in inches (+ or -) per foot: *enter real (ex: -.125)*

Drop

This option allows the user to enter a drop to the next point.

Enter drop to next point: *enter real (ex: 1'-8)*

Rise

This option allows the user to enter a rise to the next point.

Enter rise to next point: *enter real (ex: 2'-8)*

Continuous

This option allows the user to turn constant sloping on or off.

Apply slope for each segment [ON/Off] <Off>: *return for off or type ON*

Enter slope in degrees (+ or -)

This allows the user to enter a slope to the next point in degrees.

Enter slope in degrees (- or +) to next point: *enter degrees, positive or negative*

Position

This option will not take effect until the next point is picked. It is used to start the 3dpoly line at elevated positions. Picking a point at 0,0,0 and having the pipe size set to 4" with the BOP option set below, will start the 3dpoly line at 0,0,2.25.

Enter an option [**Nominal**/**Size**/**BOP**/**TOP**/**Centerline**] <Centerline>:

Nominal

This option allows changes in the present size for options below. There are two very similar options in this command. One is *Nominal* and the other is *Size.* When *Nominal* is selected the options below will use the nominal diameter (i.e. 2.0). When *Size* is used, the actual diameter is used. Both function will use the nominal size for the selection from the file.

Nominal size: *enter a real (nominal size)*

Size

See *Nominal* above.

Nominal size: *enter a real (nominal size)*

BOP

This will force the 3dpoly line to be drawn one half the present set size upward.

TOP

This will force the 3dpoly line to be drawn one half the present set size downward.

Centerline

This is the default when starting this routine. The user can change to the bottom of pipe (BOP) option, then change back with this option.

Reference

This option allows the user to start a line at a given distance from a reference point. After providing this reference point, provide a new point at a location in reference to that point. In this second prompt, give the location of the point with an “@” location.

Pick reference point: *pick point*
Use last point Z elevation <0.0000>:

This prompt requests the use of the previous last point Z elevation (point prior to the reference point). This will be used in the Z coordinate of the next prompt. If a “No” is returned here, the next prompt will use the Z of the reference point or the Z which is entered with an X and Y location (i.e.: @12,6,36 - 36 is the desired Z).

Enter relative/polar coordinates (with @): @12,-1’6-1/8 (or @1’8.25<45)

Elevation

It allows the user to change elevation in conjunction with the present UCS coordinate system. This option will also function with negative numbers. The world option allows entry of exact elevations in conjunction with the world coordinate system.

Enter elevation change or [**World**/**Pick**] <World>: W

World

Use this option to enter elevation in the world coordinate system.

Enter world elevation: *enter a real (ex: 12'1-1/4)*

Pick

Use this option to pick an elevation to establish the working elevation.

Pick elevation: *pick a point - use OSnap if neccessary*

Enter elevation change

This option changes the elevation based on the number enter here.

Undo

This option will undo the last 3dpoly line segment; prompting the user to enter the next point.

Specify next point

Use this option to pick the next point along the 3dploy line. *Enter* when finished.

Mode Convert

This routine converts components from mode to mode. This is accomplished by selecting the desired conversion from the pulldown *Plant>Accessory>Mode Convert*. For example, converting

2D Double Line	3D Solids
3D Solids	2D Double Line
2D Single Line	2D Double Line
2D Double Line, Flat	2D Double Line, North

Etc.

There are over 50 possibilities, including converting to different planes, and components needing to be rebuilt in their current mode and plane. This routine will erase the existing component and draw a new one in its' place. It will use the specification layer name to indicate which layer to delete and replace. The specification layer name will have a "_" (underline character) in front of it (ex: "_150"). This tells the Mode Convert to delete it and redraw it in its' new position or mode. Objects on other layers will not be changed by this routine as long as the layer does not have "_" in front of its' name.

Converting from one plane to another can be achieved by just changing planes and Mode Converting the components.

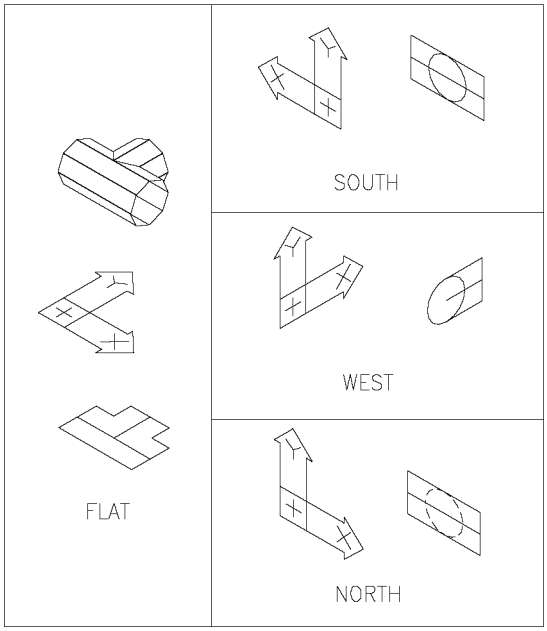
Selection can be accomplished by crossing, window or selecting individual components (including the centerlines). The following prompts will appear.

Command: **CONVERTDOUBLE** (or **CONVERTSINGLE**, **CONVERTSOLID**, and **CONVERTISO**)

Pick piping components (pipe & centerlines) to convert:
Select objects: *select by crossing*
60 found *(not necessarily the number of components)*

Select objects: *select more or enter to end*

The following tee was drawn once, copied to the other four locations, then Mode Converted into the different planes.



3D Solids

Plant menu: ▶ **Accessory** ▶ **Mode Convert** ▶ **3D Solids**

Command line: CONVERTSOLID

Settings toolbar: 

Converts selected components to 3D Solid mode.

Command: **CONVERTSOLID**
3D solids conversion...
Select objects: *Select by crossing or enter to end*

2D Double Line

Plant menu: ▶ **Accessory** ▶ **Mode Convert** ▶ **2D Double Line**

Command line: CONVERTDOUBLE

Settings toolbar: 

Converts selected components to 2D double line mode.

Command: **CONVERTDOUBLE**
2D double line conversion...
Select objects: *Select by crossing or enter to end*

2D Single Line

Plant menu: ▶ **Accessory** ▶ **Mode Convert** ▶ **2D Single Line**

Command line: CONVERTSINGLE

Settings toolbar:

Converts selected components to 2D single line mode.

Command: **CONVERTSINGLE**
2D single line conversion...
Select objects: *Select by crossing or enter to end*

Isometric

Plant menu: ▶ **Accessory** ▶ **Mode Convert** ▶ **Isometric**

Command line: CONVERTISO

Settings toolbar:

Converts selected components to Isometric mode.

Command: **CONVERTISO**
Isometric conversion...
Select objects: *Select by crossing or enter to end*

Existing

Plant menu: ▶ **Accessory** ▶ **Mode Convert** ▶ **Existing**

Command line: EXIST

This routine will turn xdata information off or remove xdata totally from the components. It also allows centerline deletion. After xdata has been turned off or removed, the graphics will be moved to the "EXIST" layer.

The following prompts for complete removal of xdata. A "yes" response would remove the xdata completely. "No", the default will leave the xdata but turn it off in Bill of Materials generation.

Command: **EXIST**
Remove all information from components <No>: *enter or type* **Y**

This option only appears when complete xdata removal is requested with a "Yes".

Remove centerlines from components <No>: *enter or type* **Y**

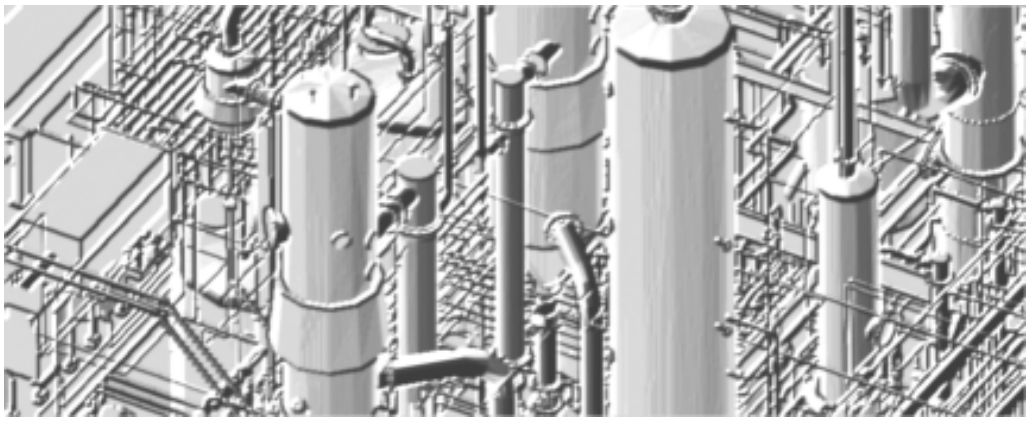
This option will move only components or everything selected to the "Exist" layer.

Convert [**All objects/Components only**] <Components only>:
Select objects: *select components, objects, etc.*

This function will remove gaskets and bolts from the drawing environment. If the user wishes not to remove the gaskets and bolts, they can use the GCEDIT (see page 321) command and use the existing check box available in that command.

CHAPTER 5

Utility



In This Chapter

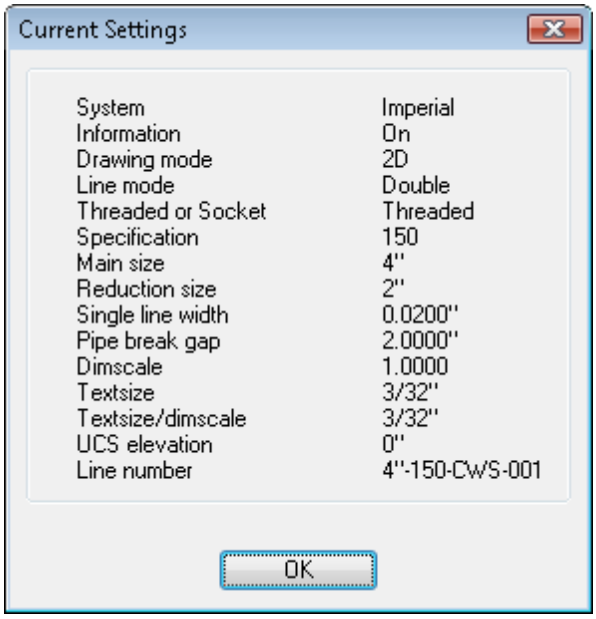
Current Settings 300
Set Component 300
Component Edit 303
View 322
Elevation 324
Centerline 325
Continuity 326
Clash 327
Break 332
ISO Flag 333
Insulation 334
Insulation View Palette 335
Dim Adjustment 348
Line Isolate 350
Line View Palette 351
Layer Control 355
Drawing Control 358
MVSetup 369
Zoom Lock 370
Zoom Factors 370
Pipe Support Modeler Palette 371

Current Settings

Command line: CURRENT

Settings toolbar:

This dialog displays the current main and reducing sizes. It also displays the currently set specification. If the items are not set it will indicate as such.



Set Component

There are four commands available that allow the user to set the size, spec, line number and all of the above by selecting a component.

Size

Plant menu: ▶ **Utility** ▶ **Set Component** ▶ **Size**

Command line: COMP2SIZE

Settings toolbar: 

This command allows the user to set or change the main and/or reduction size by simply selecting a component.

Command: **COMP2SIZE**
Select component to set size: *pick component*

The new main size is then displayed.

Main size has been set to 4"...

If the reduction size was also set, the new reduction size is also displayed.

Reduction size has been set to 2"...

If main and reduction sizes are both set, CADWorx Plant will allow the user to reverse the main and reduction sizes as shown below.

Reverse main and reduction [**Yes/No**] <No>:

Spec

Plant menu: ▶ **Utility** ▶ **Set Component** ▶ **Spec**

Command line: COMP2SPEC

Settings toolbar: 

This command allows the user to set the current specification by selecting a component of the desired spec.

Command: **COMP2SPEC**
Select component to set specification: *pick component*

CADWorx Plant will prepare the new specification and display the new spec.

Prepared <INSTALL-DIR>\SPEC\150.spc specification...

Size Spec

Plant menu: ▶ **Utility** ▶ **Set Component** ▶ **Size-Spec**

Command line: COMP2SIZESPEC

Settings toolbar:

This command allows the user to set the main, reduction sizes and the current specification by selecting a component.

Command: **COMP2SIZESPEC**
Select component to set size and spec: *pick component*

CADWorx Plant will prepare the new specification and display the new spec.

Prepared <INSTALL-DIR>\SPEC\150.spc specification...

The new main size is then displayed.

Main size has been set to 4"...

If the reduction size was also set, the new reduction size is also displayed.

Reduction size has been set to 2"...

If main and reduction sizes are both set, CADWorx Plant will allow the user to reverse the main and reduction sizes as shown below.

Reverse main and reduction [**Yes/No**] <No>:

Line

Plant menu: ▶ **Utility** ▶ **Set Component** ▶ **Line**

Command line: COMP2LINE

Settings toolbar:

This command allows the user to set the current line number by selecting a component of the desired line number. Any components created after running this command will use the new line number.

Note: This command will only work if LineNumberSystem (see page 60) = 2 (Static).

Command: **COMP2LINE**
Select component to set line number: *pick component*

CADWorx Plant will display what the line number was set to.

Line number set to 4"-150-2200CWC...

All

Plant menu: Utility ▶ Set Component ▶ All

Command line: COMP2ALL

Settings toolbar: 

This command allows the user to set the main, reduction sizes, current specification, and line number by selecting a component.

Note: Line number will be set in this command only if LineNumberSystem (see page 60) = 2 (Static).

Command: **COMP2SIZESPEC**
Select component to set size and spec: *pick component*

CADWorx Plant will prepare the new specification and display the new spec.

Prepared <INSTALL-DIR>\SPEC\150.spc specification...

The new main size is then displayed.

Main size has been set to 4"...

If the reduction size was also set, the new reduction size is also displayed.

Reduction size has been set to 2"...

If main and reduction sizes are both set, CADWorx Plant will allow the user to reverse the main and reduction sizes as shown below.

Reverse main and reduction [**Yes/No**] <No>:

CADWorx Plant will display what the line number was set to.

Line number set to 4"-150-2200CWC...

Component Edit

There are three different commands editing component(s).

Component Edit

Plant menu: ▶ **Utility** ▶ **Component Edit** ▶ **Local Edit**

Command line: CEDIT

Settings toolbar:

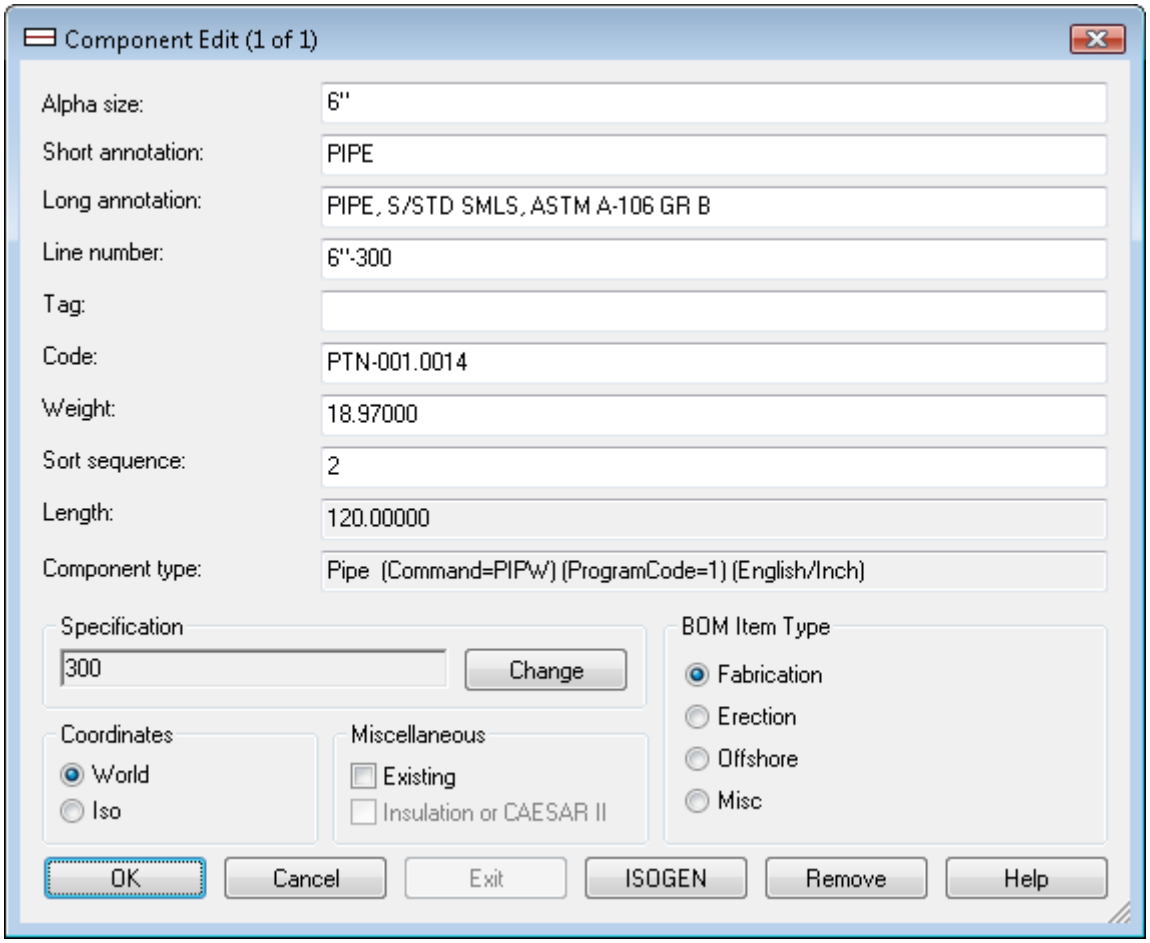
This routine accesses data stored within components. It allows descriptive modifications, data removal, and different flag settings.

Note: CEDIT can also be invoked by double-clicking on one or more CADWorx Plant entities.

This command first requests the user to select components for editing.

Command: **CEDIT**
Select components for editing...
Select objects: *select one or many*

After selection is made, the following dialog will appear.



Specification

This function is similar to the Specification Change (see page 271) function. The button located here will only change the specification for the currently selected component. It is also provided here to allow the user to view the specification used to draw the component.

Coordinates

This setting controls how pipe lengths are calculated. With "World" setting, calculation of pipe length is based on the length of the pipe component center line. This setting affects the pipe components only (pipe, nipple, flanged pipe, tubing) and not other components such as elbows or flanges. If a pipe component is stretched, trimmed, or broken, the position of the component is automatically updated along with the pipe length

With the "Iso" setting, the length that is set in this dialog will be used. This length will appear in Bill of Material generation. This is useful in AutoISO (see page 183) Isometric creation (refer to the ISO Flag (see page 333) option for more details).

This Iso toggle will also activate the Length edit box for making changes to bolts lengths. After making length changes, set the toggle back to World.

Miscellaneous

The *Existing* check box option is enabled, the component will not be available in the Bill of Material generation. This will also move the graphics of the component to the “Exist” layer. For ISOGEN, the component will be shown as DOTTED and DIMENSIONED in the ISO drawings.

The *Insulation/CAESAR II* check box indicates this component either has CAESAR II information and/or insulation attached. Use the insulation routine to remove or add insulation. CAESAR II information is pipe stress analysis information which might have been added.

Note: This check box will show status of insulation added with INSULATION command. It will NOT show the status of insulation added with the InsulationView palette.

BOM Item Type

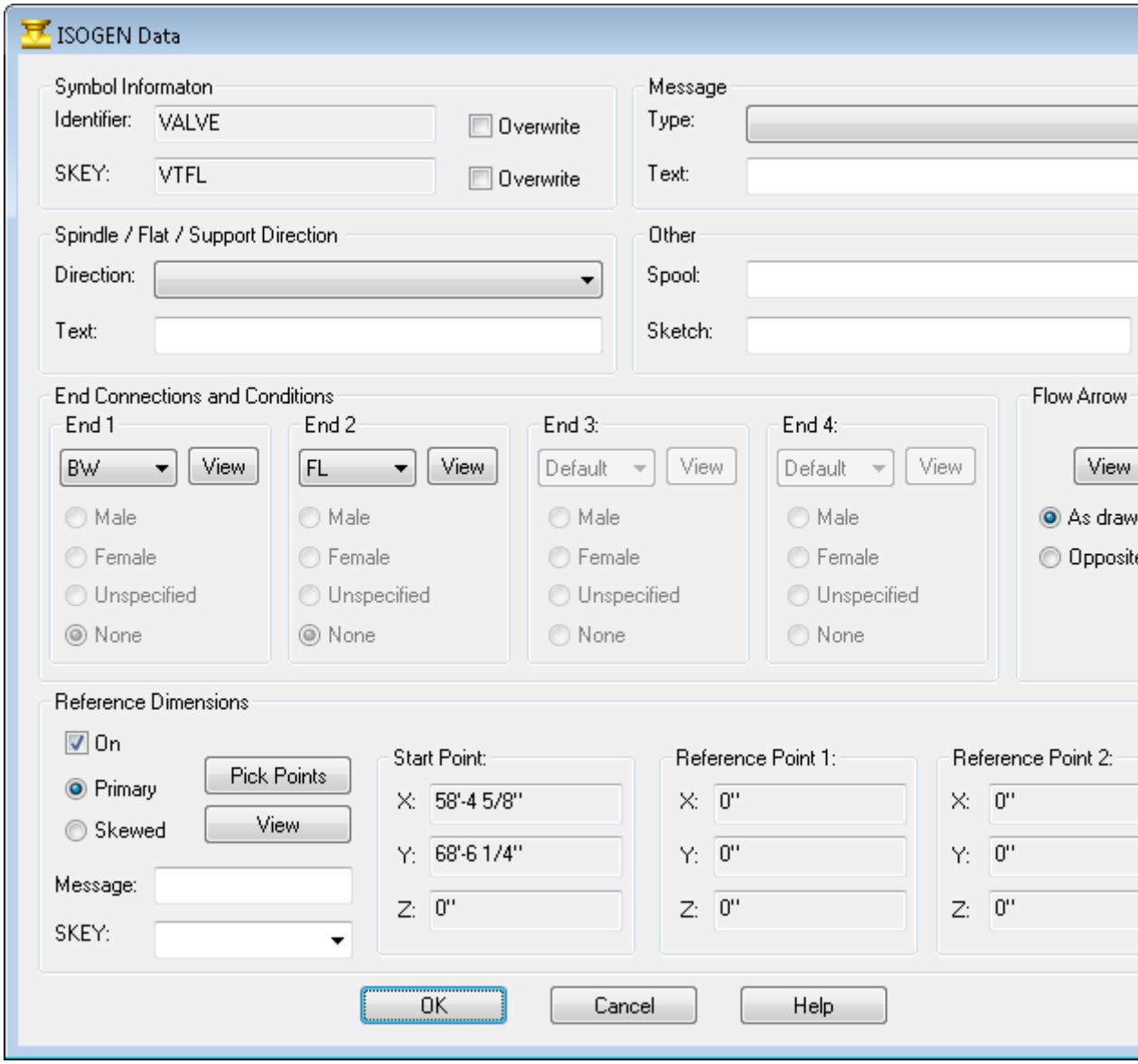
The *BOM Item Type* toggles allows the user to specify the item type for each component. The item type then appears on the Bill of Materials if the FLAG column is selected to appear in the bill of material. The FLAG column can be added to appear in the bill of material using the BOMSETUP (see page 386) command. The user will also have to sort (see page 388) by the FLAG column. This will group similar types in the Bill of Material for AutoISO and ISOGEN.

The BOM Item Type can be set in the specification file.

The following information only applies to ISOGEN and weld gap components: The "Misc" BOM Item Type for weld gaps is represented as a "Field Fit Weld" for ISOGEN output. Use ISOGEN Option Switch 22 to increase cut piece add on allowances for field fit welds.

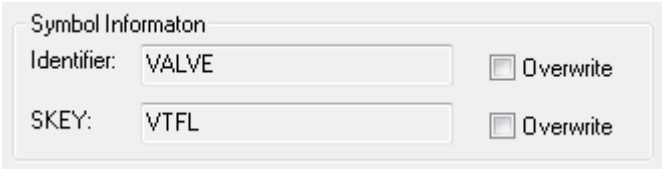
ISOGEN

Clicking the ISOGEN button will invoke the following dialog:



Note: The ISOGEN button will only be available if CADWorx Plant Professional is active on the current hardware lock in use

Symbol Information

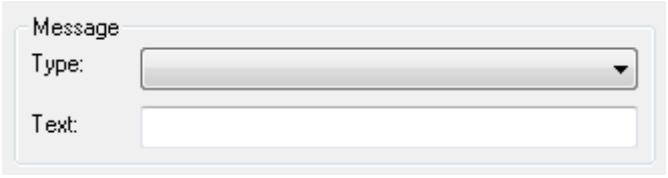


This section of the dialog can be used to set a different SKEY for the component.

The default SKEYs assigned to a component are listed in this table: ISOGEN SKEY Information

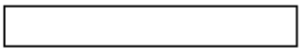
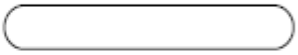
The following file contains sample images of SKEYs: ..\CADWorx Plant\Isogen\Isogen_Utils\Skey.pdf

Message

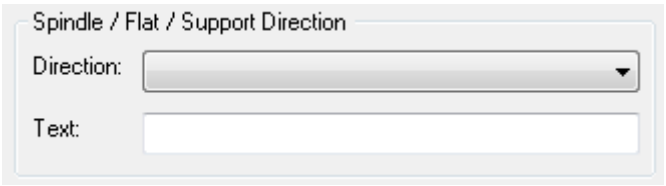


This section of the dialog can be used to display a message within the Isometric.

The message text can be enclosed in several different shapes: UNBOXED, SQUARE, POINTED, ROUND, CIRCLE, TRIANGLE, DIAMOND, DOUBLE CIRCLE, or ELLIPSE.

Attribute	Isometric Presentation
MESSAGE	Unboxed
MESSAGE-SQUARE	
MESSAGE-POINTED	
MESSAGE-ROUND	
MESSAGE-CIRCLE	
MESSAGE-TRIANGLE (Two characters maximum)	
MESSAGE-DIAMOND (Three characters maximum)	
MESSAGE-DOUBLE-CIRCLE	
MESSAGE-ELLIPSE	

Direction



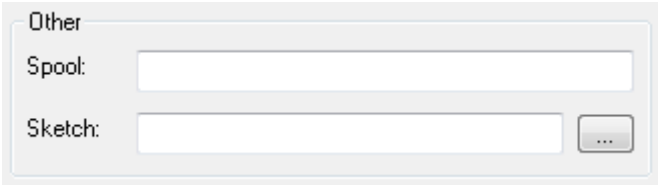
This section of the dialog can be used to set valve spindle direction, eccentric reducer flat direction, or support direction.

The isometric graphic will only be displayed in one of the following directions: NORTH, SOUTH, EAST, WEST, UP, or DOWN.

The Text edit box can be used to display a text message indicating a more exact direction.

Note: ISOGEN Option Switch 80 (Dimension to Valve Centers) will only work if the valve has a spindle direction set.

Other



This section of the dialog can be used to set a spool identifier, detail sketch, dimension status, and mark number.

The Spool edit box can be used to set the spool identification. A value entered in this edit box will overwrite the default spool value created by ISOGEN.

The Sketch edit box can be used to display a detail sketch within the isometric.

For more information on detail sketches, review the following help files:

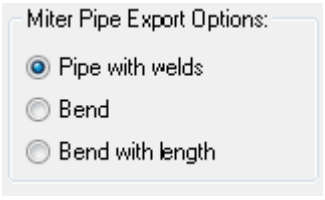
..\CADWorx Plant\Isogen\Isogen_Utils\Information Notes.pdf

..\CADWorx Plant\Isogen\Isogen_Utils\Detail Sketches.pdf

The Dim Status can be used to control the dimensioning of individual components.

The Mark edit box can be used to set the mark number for a component.

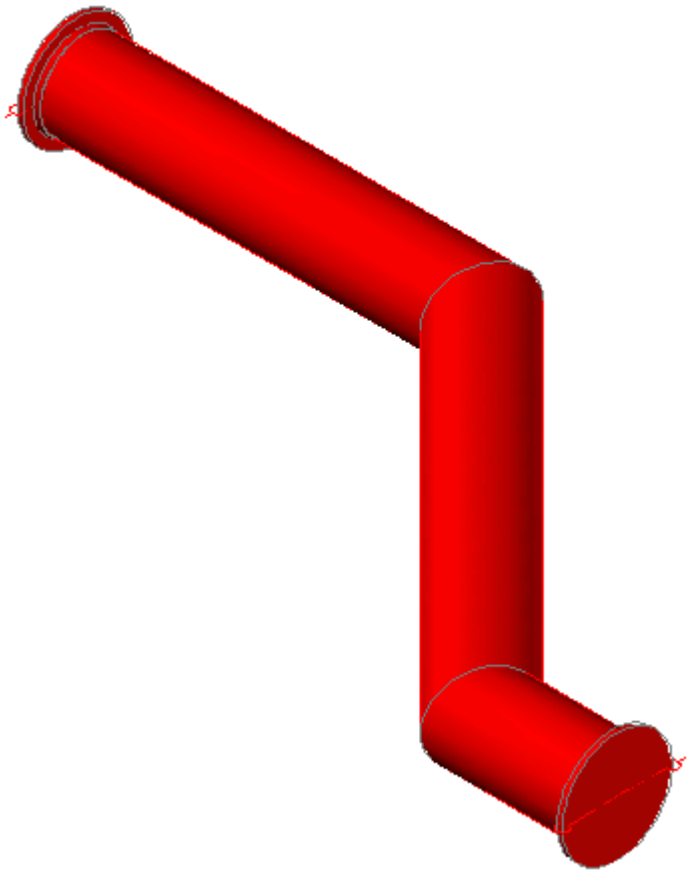
ISOGEN Edit - Miter Pipe Options



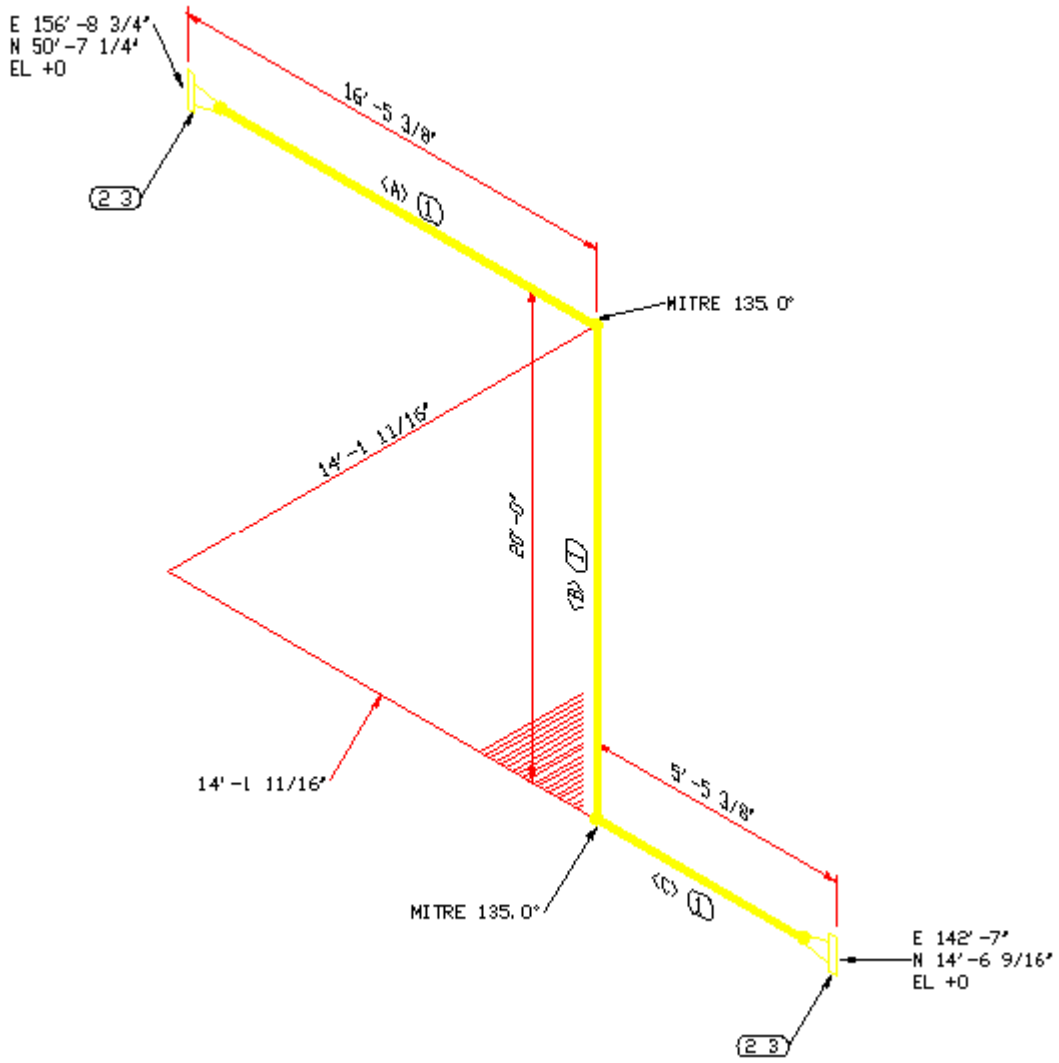
This section of the dialog is used to control how miter pipe is exported to ISOGEN. Miter pipe can be many types of shapes. For export to ISOGEN, only three options are available: Pipe with welds, Bend, or Bend with length. If the shapes make a bend (elbow) of any angle, use the Bend or Bend with length option. If the shape is a straight pipe run mitered at an angle, use the Pipe with welds options.

Pipe with welds option will export each straight run as pipe pieces with a miter weld connecting each segment.

Sample model set to Pipe with welds option:

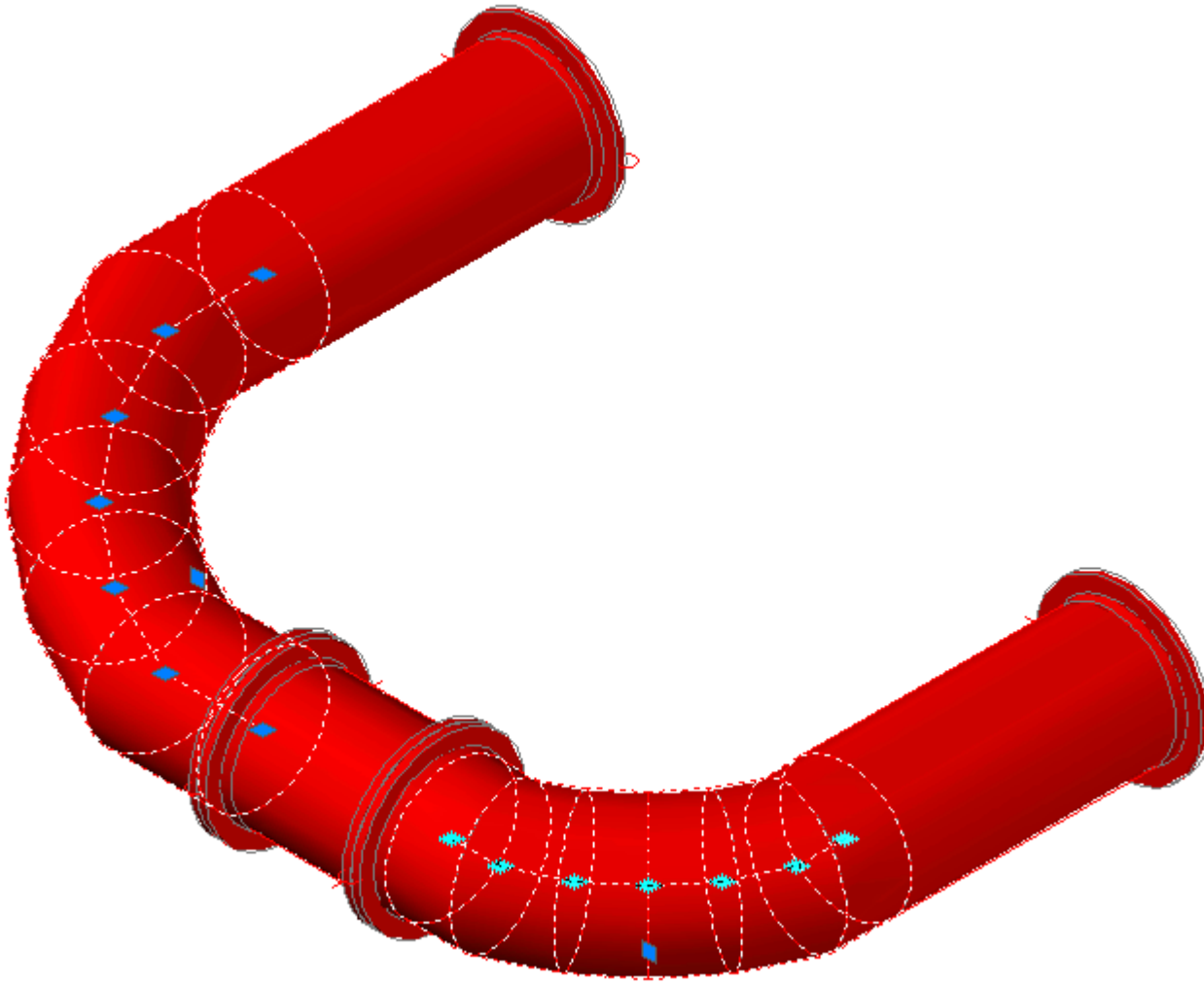


Sample ISO resulting from Pipe with welds option:

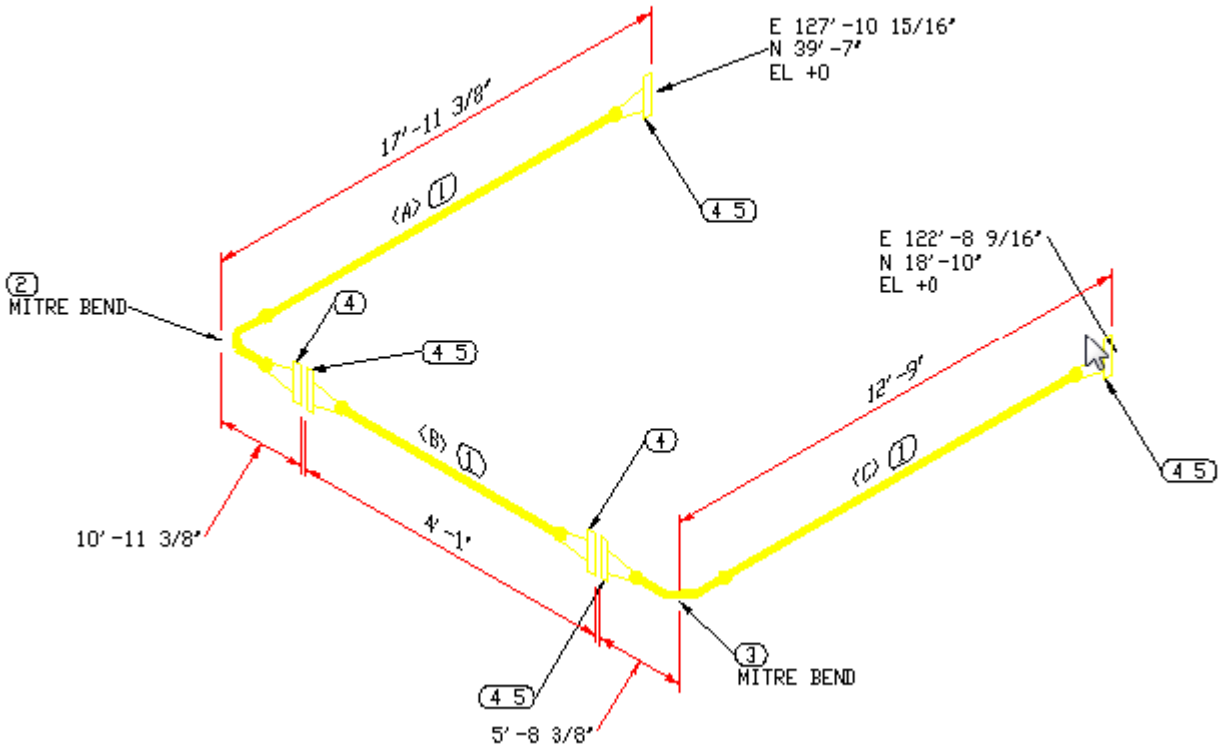


Bend or Bend with length option will export all the segments of the miter pipe component as a single miter bend. The Bend with length option will append the length of the component to the long description. This will allow two bends with different lengths to have separate marks in the material list.

Sample model set to Bend with length option:



Sample ISO and material list resulting from Bend with length option:



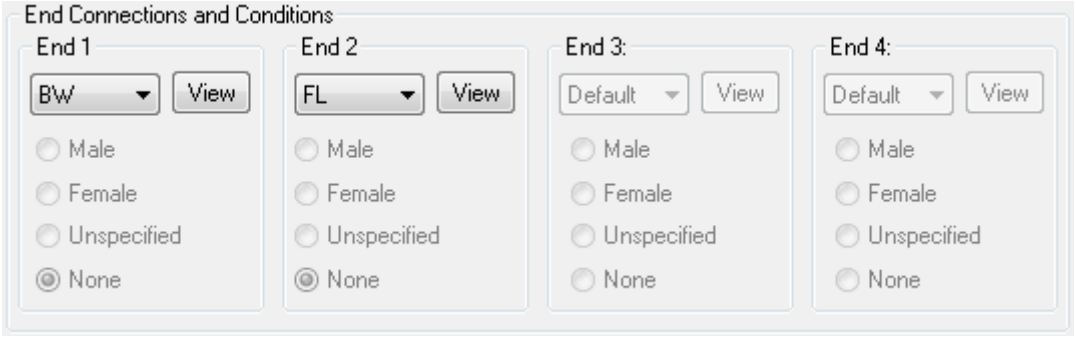
BILL OF MATERIAL

MARK	SIZE	DESCRIPTION	QTY
1	42	PIPE, STD, WELDED OR SEAMLESS	17' 3"
2	42	MITERED PIPE, STD, WELDED OR SEAMLESS (16' -8 5/8" LG)	1
3	42	MITERED PIPE, STD, WELDED OR SEAMLESS (8' -4 5/16" LG)	1
4	42	FLG, RFWN 150LB S/STD BORE, ASTM A-105	6
5	42	GASKET, 1/8" THK, 150LB (NO WGT)	4
		TOTAL FABRICATION WEIGHT	1

CUT PIPE LENGTH

PIECE NUM	LENGTH (FT-INCH)	SIZE (INCH)
<A>	7' 0"	42
<B>	3' 2 1/4"	42
<C>	7' 0 5/8"	42

End Connections and Conditions



This section of the dialog can be used to set the end connections and end conditions of a component. The default end connection of a component is set by the last two characters of the SKEY.

Connection Types	Description	Default Condition
BS	Ball and Socket	Male
BW	Butt Weld	Unspecified
CL	Clamped	Male
CP	Compression	Female
FA	Flared	Male
FL	Flanged	Unspecified
GF	Gland	Female
GL	Glued	Female
LC	Liner (Clamped)	Male
LR	Liner (Reducing)	Male
LN	Liner (Nut)	Male
MP	Male Part Hygienic	Male
PF	Push Fit	Female
PL	Plain End	Male
SC	Screwed	Female
SW	Socket Weld	Female

Only the following end types will allow a Male or Female end condition setting: SC, SW, CP, GL, and PF.

If the end condition does not fall into either a Male or Female category, a setting of Unspecified can be used.

Flow Arrow



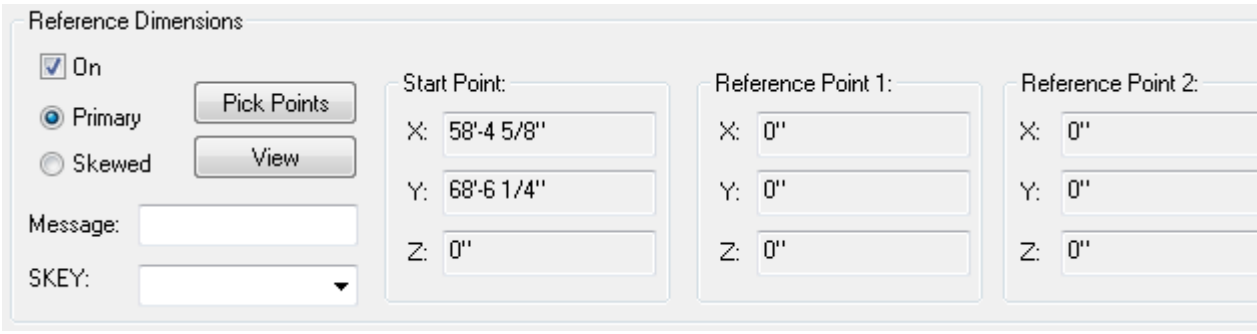
This section of the dialog is used to control how flow arrows are displayed for valves.

Use the radio buttons As drawn and Opposite to control the direction of flow arrows displayed within the isometric.

The View button will display a preview within the model drawing of how the flow arrow will be displayed within the isometric drawing.

Note: ISOGEN Option Switch 17 controls the display of flow arrows.

Reference Dimensions



This section of dialog can be used to display reference dimensions within the isometric.

On: Use this check box to activate a reference dimension.

Primary / Skewed: Use these to toggle to set the dimension as either primary or skewed.

Message: Use this edit box to display a message along the reference dimension.

SKEY: Use this edit box to set the reference dimension SKEY. Either select a value from the drop down or enter a value (see table below for more details)

Pick Points: This button will ask the user to select two points at the command prompt. These two points will be used to position the isometric graphic used for the reference dimension.

View: This button will draw a temporary graphical line connecting the Start Point, Reference Point 1, and Reference Point 2.

Start Point: These edit boxes will display a point on the component that is automatically determined by CADWorx.

Reference Point 1 and *Reference Point 2*: These edit boxes will display the two selected points that will be used by the reference dimension.

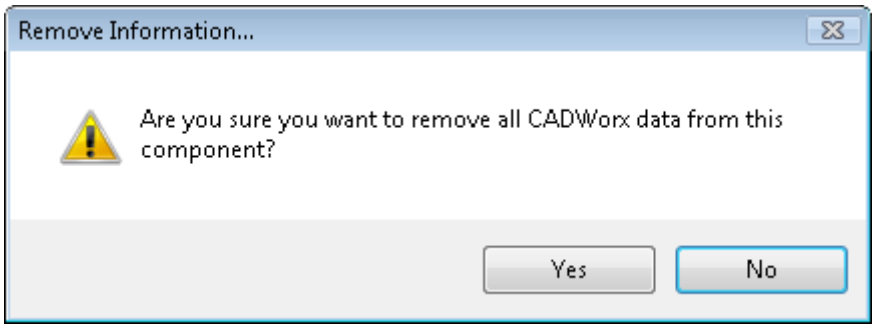
SKEY	Item Category	Default Dotted Line No.	Default Plotted Line Type
BLD	Building	2	
EQU	Equipment Item	2	
FLR	Floor Level	3	
GRD	Grid Line	3	
HST	Horizontal Steelwork element	1	
PIP	Pipeline	4	
VST	Vertical Steelwork element	1	
WAL	Wall	3	
XXX	Miscellaneous category (User defined)	'Normal'	

A single digit (1, 2, 3, or 4) number can be added to the end of the SKEY to change the default line type.

For Example: An SKEY of EQU4 would use a line type 4 for this reference dimension.

Remove

This button allows removal of component xdata. Whenever *Remove* is picked, the following dialog will appear confirming deletion of xdata.



If this is not the desired result, pick *No* from the dialog.

Note: Several functions in CADWorx Plant rely on the presence of xdata in the components. Removing this data can render these functions useless.

XREF Edit

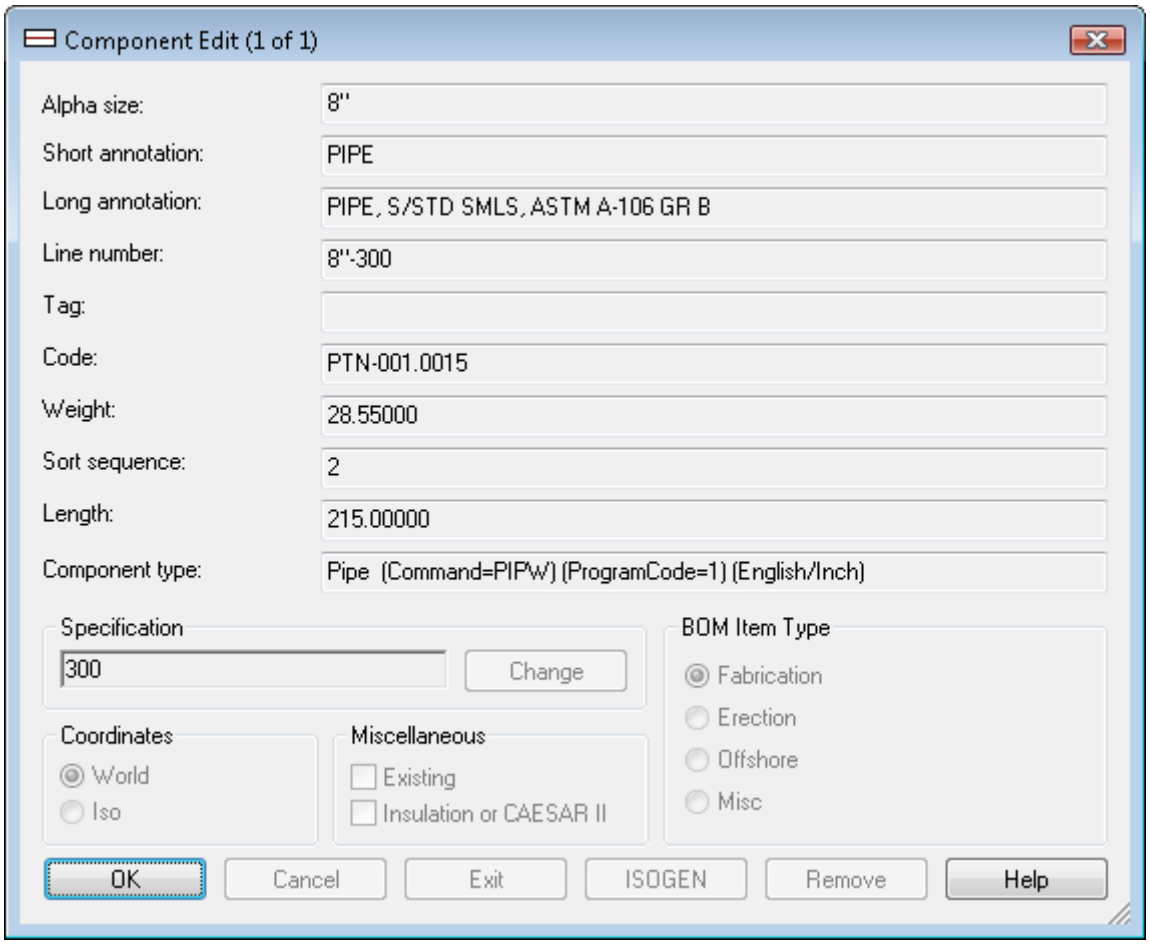
Plant menu: ▶ **Utility** ▶ **Component Edit** ▶ **Xref Edit**

Command line: XCEDIT

Settings toolbar:

XREF (eXternal References) CEDIT (see page 304) is a function that works in a similar fashion to the CADWorx Plant CEDIT functionality. This function uses the same dialog as CEDIT but it will not allow the user to modify any of the fields within it. All the modifiable fields will be disabled. If a modification to a component is required from within an Xref, use the AutoCAD command REFEDIT to open that part of the Xref, then use the normal CEDIT function to make any modification which might be necessary. After the reuired modification is completed, close the Xref with the REFCLOSE command. The main purpose of this function is to allow the user to view sizes or information within the component.

Command: **XCEDIT**



Global Edit

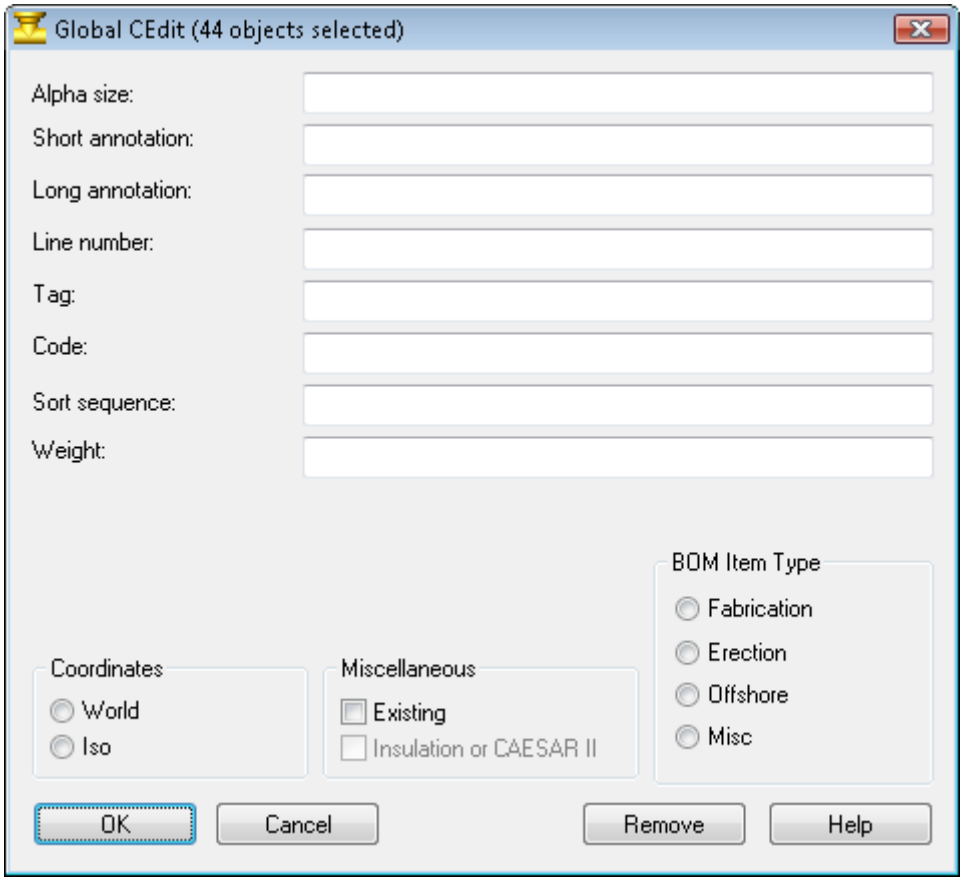
Plant menu: ▶ **Utility** ▶ **Component Edit** ▶ **Global Edit**

Command line: GCEDIT

Settings toolbar:

This command allows multiple components to be modified at one time. When multiple components are selected, this command displays the following dialog. Each field within this dialog will be blank. If any field or edit box is modified within this dialog, then that field will be changed throughout all the components that were selected. The edit boxes or fields that were not modified or changed will not be updated within the components.

Command: **GCEDIT**



When finished, a prompt will appear indicating the total number of components that were updated.

Updated 44 component(s)...

Coordinates, Miscellaneous and *BOM Item Type* are similar to those appearing in the CEdit (see page 304) dialog.

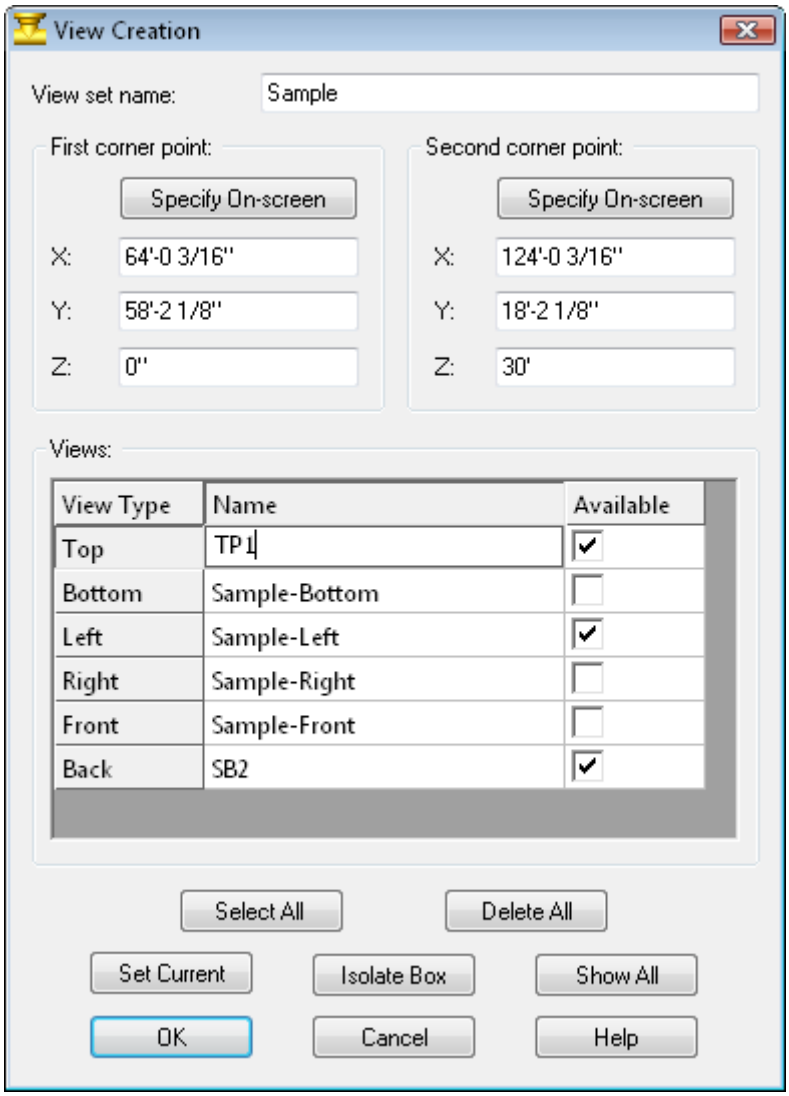
View

View Box

Command line: VIEWBOX

This command allows the user to create clipped views by defining a 3D box. This command is very useful in creating layout drawings in paperspace. The following dialog is displayed when the command is run.

Note: If running CADWorx Equipment, run the command EQUIPVIEWBOX



The user can either type in or specify on-screen the two opposite corners of the box they intend to define. The Z value of the first and second corners cannot be the same value. Once the corners are specified, the user can then add the views they want generated by check the box in the *Available* column. To delete a view already added, uncheck the box in the *Available* column.

Each view added is also assigned a name based on the overall view set name. This name can be changed and once the views are created, they will be displayed in the AutoCAD View dialog. Clicking *OK* creates the sides of the box that represent each view the user requested.

Select All button will check all boxes in the Available column.

Delete All button will delete all AutoCAD Views that were previously created with this view set and close the dialog.

Set Current button will set the AutoCAD window to the view selected.

Isolate Box button will hide all AutoCAD entities outside the View box. Entities that cross the boundaries of the View box will typically not be hidden.

Show All button will show all entities in the drawing.

View Edit

Command line: **VIEWEDIT**

This command allows the user to edit the box created with the VIEWBOX command described earlier. Please refer to the VIEWBOX command on how to add/delete views from the box. Double clicking on an existing view box will automatically run this command.

Note: If running CADWorx Equipment, run the command EQUIPVIEWEDIT

Elevation

Set

Plant menu: ▶ **Utility** ▶ **Elevation** ▶ **Set**

Command line: ELEV

This command is the standard AutoCAD Elev command. It can be used to place piping at a preset elevation. The only problem with this command, and our most often supported problem, is the user forgets to set it back to 0'-0" elevation, and all piping drawn is to some arbitrary elevation that can confuse the user. It is suggested that this command be used with caution. An alternative is to use the Point and Shoot UCS (see page 506) built into CADWorx Plant (VIEWFLAT).

Refer to the AutoCAD documentation for complete detail regarding this command (ELEV).

Change

Plant menu: ▶ **Utility** ▶ **Elevation** ▶ **Change**

Command line: CHANGELEV

This routine relocates all entities (blocks, lines, polylines, etc.) selected to a new elevation in reference with the world coordinate system. Also provided is the option to change a component elevation higher to 1/2 its actual outside diameter.

Command: **CHANGELEV**
Elevation change **[BOP correction/Arbitrary]** <Arbitrary>:

BOP correction

Select objects: *select by any method (add and remove is allowed)*

Arbitrary

Enter world elevation: *enter new elevation (ex. 10'1.5)*
Select objects: *select by any method, enter to end selection set*
Pick base point: *endp of pick any component*

Centerline

Convert

Plant menu: ▶ **Utility** ▶ **Centerline** ▶ **Convert**

Command line: CONVERTCL

Convert centerline allows the user to move all selected centerlines to the “SYSTEM” layer. This allows freezing the “SYSTEM” layer and plotting shows the plans and elevation without centerlines. Optional points and some Single Line objects are drawn on the “SYSTEM” layer also. This layer needs to be frozen before plotting.

Selection can be accomplished by crossing a window, or selecting individual components (including the centerlines). The following prompts will appear.

Command: **CONVERTCL**
Pick piping centerline(s) to convert:
Select objects: *select by crossing*

After the preceding prompt, the option to draw new centerlines on top of the centerlines exist.

Draw new centerline over existing [Yes/No] <Yes>: *enter or type N for no*

Offset

Plant menu: ▶ **Utility** ▶ **Centerline** ▶ **Offset**

Command line: OFFSETCL

This routine allows the user to offset CADWorx Plant pipe and fittings. Offsetting a component with xdata attached is like copying it to the offset location. This prevents this copy effect from occurring. On fittings, such as elbows, this routine will not offset the centerline radius, although, it will provide a straight line offset to one side or the other. This routine also offsets ordinary lines, circles and other objects. The following prompts, which are similar to the AutoCAD Offset command, will appear.

Command: **OFFSETCL**
Specify offset distance or [Through] <Through>: *pick point or enter real number*

Select object to offset: *select object*
Specify point on side to offset: *pick side*

Select object to offset: *enter to end*

Continuity

Plant menu: ➤ **Utility** ➤ **Continuity**

Command line: CONTINUITY

This function finds all gaps and ends of any piping system. It will provide the user a visual reference to each and every end. This function explicitly finds any component's end that does not have an attaching component. It draws a circle on the current layer at each end it finds.

If an elbow was not properly attached to a piece of pipe, this function would draw two circles that would represent the end of the elbow and the end of the pipe.

Command: **CONTINUITY**
Select objects: *select objects*
18 found, 7 groups
11 were filtered out.
Select objects: *enter to end selection*

After selecting all components in the desired piping system, the function requires a diameter for the circle markers. The quickest way to respond to this prompt would be to pick two point on the screen for the desired size of the circles.

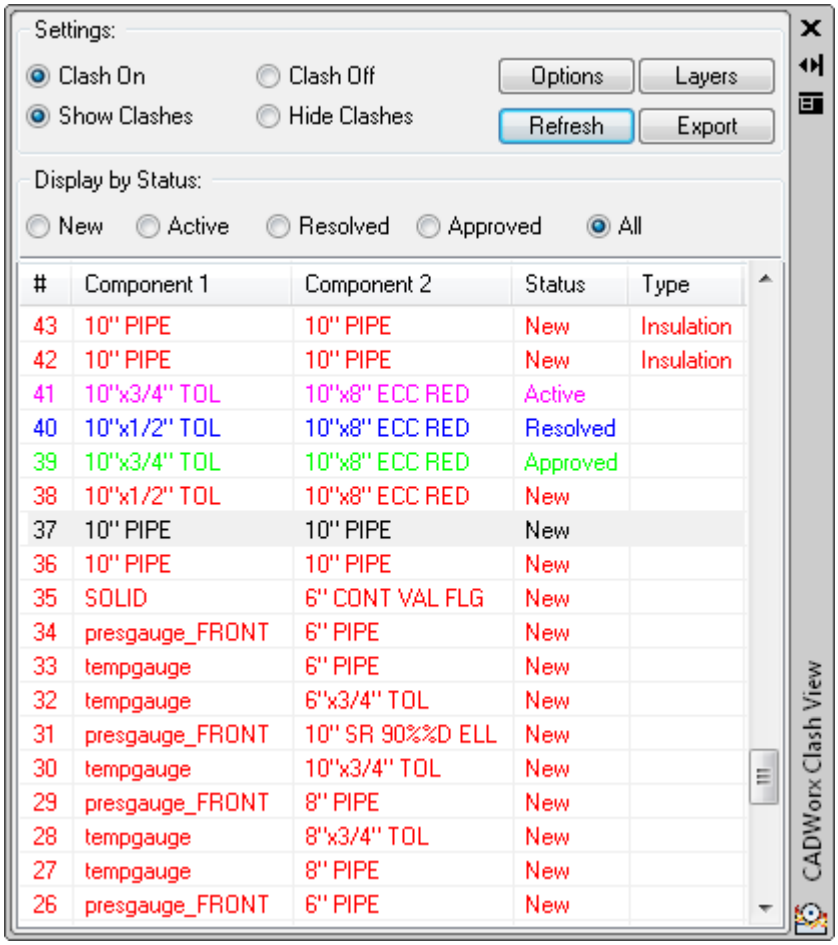
Enter marker diameter: *enter a diameter or pick two points*
Enter tolerance <0.0200>: *enter a real*

Clash

Plant menu: ➤ **Utility** ➤ **Clash**

Command line: CLASH

This command allows the user to run interactive clash detection on AutoCAD 3D solids present the current drawing and any attached XREF drawings. Upon running the command, a palette is displayed as shown below.



#	Component 1	Component 2	Status	Type
43	10" PIPE	10" PIPE	New	Insulation
42	10" PIPE	10" PIPE	New	Insulation
41	10"x3/4" TOL	10"x8" ECC RED	Active	
40	10"x1/2" TOL	10"x8" ECC RED	Resolved	
39	10"x3/4" TOL	10"x8" ECC RED	Approved	
38	10"x1/2" TOL	10"x8" ECC RED	New	
37	10" PIPE	10" PIPE	New	
36	10" PIPE	10" PIPE	New	
35	SOLID	6" CONT VAL FLG	New	
34	presgauge_FRONT	6" PIPE	New	
33	tempgauge	6" PIPE	New	
32	tempgauge	6"x3/4" TOL	New	
31	presgauge_FRONT	10" SR 90%%D ELL	New	
30	tempgauge	10"x3/4" TOL	New	
29	presgauge_FRONT	8" PIPE	New	
28	tempgauge	8"x3/4" TOL	New	
27	tempgauge	8" PIPE	New	
26	presgauge_FRONT	6" PIPE	New	

Clash On / Clash Off: These radio buttons will temporarily turn the clash system on and off.

Show Clashes / Hide Clashes: These radio button will show or hide the clash entity that have been previously created in the model.

Options button: See section on Options dialog

Layers button: See section on Layers dialog

Refresh button: Click this button to refresh the Clash List

Export button: Click this button to export the Clash List to a text file. The file will be exported to the same directory as the model drawing file. The name of the file will be the drawing name with _Clash.txt appended.

Display by Status: These radio buttons will control what type of clashes are displayed in the list.

Clash List:

- A number (#) given to each clash.
- CADWorx data (if available) on the two components is listed.
- Status column will list clash status.
- Type column will list if clash was from CADWorx Insulation View created component.
- Color of the grid items can be set using the Options dialog.
- Left clicking an item in the list will generate the clash entity in the model and zoom to the clash
- Right clicking on the list displays the following options:



Clash Status (New, Active, Resolve, Approved): These options will allow the user to change the status of a clash.

Zoom To: This option will zoom to the clash point

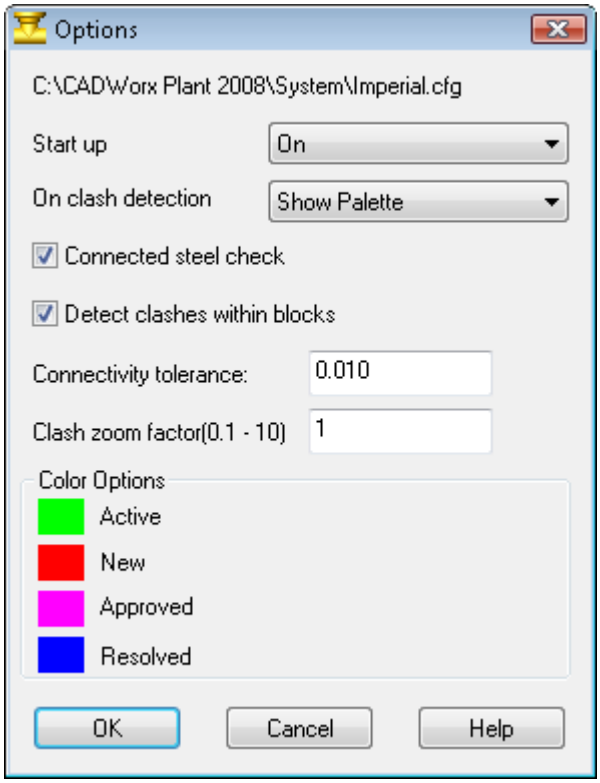
Viewer: This option will display a dialog with two clashing components.

Isolate Selected: This option will isolate the current clash that is selected in the list.

Isolate All: This option will create clash entities for all clashes and isolate them. To remove all clash entities from the model run the command CLASHDELETEALLOBJECTS

Show All: This option will show all components in the drawing.

Options



Start up

This option controls clash detection on drawing start up.

On: Clash detection is always on for every drawing.

Off: Clash detection is off for every drawing.

Display Prompt: The program will prompt the user every time a drawing is opened to run clash detection or not.

On clash detection

This option controls what happens when a clash is detected.

Do Not Report: The program will not indicate to the user if a clash has been detected.

Show Tray Notification: The program will notify the user through the tray bubble notification that a clash has occurred.

Show Palette: The program will display the palette every time a clash is detected.

Connected steel check

This option controls whether the clash detection system checks for connecting steel members. If this option is unchecked, steel members that are touching each other will be reported as a clash.

Detect clashes within blocks

The system will not detect clashes inside blocks. If this option is turned on, the system will detect clashes within any blocks in the current drawing.

Synchronize clashes from XREFs

The system will check for clash data inside an XREFs and update the master drawing data for that clash. Only "New" clashes in the master drawing which are from an XREF drawing will have the status updated to match the status that is saved in the XREF drawing.

Synchronize layers from XREFs

The system will check for clash layer data inside an XREFs and update the master drawing clash layer data to match. If a layer in an XERF is excluded from clash detection, the XREF layer will also be excluded in the master drawing.

Ignore insulation clashes

The system will ignore all clashes generated from components created with the CADWorx Insulation View palette.

Connectivity tolerance

This value is the tolerance for checking component connectivity. Note that if two components are connected, it will not be reported as a clash even if the actual solids intersect.

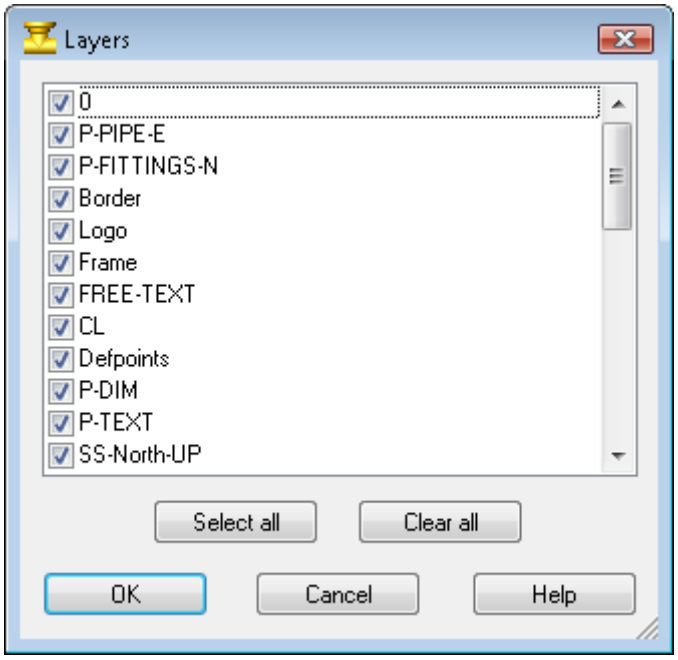
Clash zoom factor

This value control the zoom factor used during clash.

Color Options

These options control the color of the clash entity and the color of the list.

Layers



This dialog allows the user to include/exclude certain layers from clash detection. By default, all layers are considered for clash detection. The user could exclude certain layers. XREF layers will also be displayed in this dialog.

Break

Autobreak

Plant menu: ▶ **Utility** ▶ **Break** ▶ **Manul/Auto**

Command line: AUTOBREAK

This routine is used to break pipe. It will determine if the pipe has been drawn in double line or single line mode, thus providing different prompts for different modes. It is not necessary to set any drawing or line modes prior to using this routine.

Note: 3D pipe cannot be broken with this routine.

If the pipe has been previously broken by other means, the desired results might not be achieved. The user may want to use the Convert Centerline routine instead (from the pulldown, Utility>Centerline>Convert).

Command: **AUTOBREAK**
Enter an option **[Auto/Under pipe/Over pipe]** <Over pipe>:

Auto

This option automatically breaks pipes that overlap each other at different elevations. The pipes must totally overlap or the routine will not work. If the pipes are at the same elevation an error message will appear.

"Both pipe runs are the same elevation, Auto can not be used..."

The routine will determine whether the chosen pipe should be broken with or without pipe break symbols. If the user breaks the pipe below, no pipe break symbols will be provided. If the user breaks the pipe above, each end of the broken pipe will have pipe break symbols. The distance of the break will be the sum of the pipe's outside diameter (the one not broken) and 2 times the break distance setting. For Double Line pipe and Single Line pipe, the following prompts will appear.

Select first point on centerline to break: *pick pipe centerline*
Select second point on centerline to break: *pick another point on pipe centerline*

This command does not work with 3D solids.

Under pipe

This option breaks a pipe without the pipe break symbols and the presence of another pipe. It will also allow the user to determine the distance of the break on the pipe. The user can enter polar values if desired, as long as the location lands on the centerline. The following prompts will appear.

Select first point on centerline to break: *pick pipe centerline*
Select second point on centerline to break: *pick another point on pipe centerline*

Over pipe

This option breaks a pipe with the pipe break symbols and without the presence of another pipe. It will also allow the user to determine the distance of the break on the pipe. The user can enter polar entries if desired, as long as the location lands on the pipe's centerline. The following prompts will appear:

Select first point on centerline to break: *pick pipe centerline*
Select second point on centerline to break: *pick another point on pipe centerline*

End

Plant menu: ➤ Utility ➤ Break ➤ End

Command line: ENDBREAK

This routine will provide a pipe break symbol on the end of a pipe. The symbol will be different for Double Line and Single Line pipe. Its' size and color will depend on the current set size and specification. The following prompts will appear.

Command: **ENDBREAK**
Pick end point of pipe: *pick end of pipe or hit enter for last*
Pick direction: *pick direction*

ISO Flag

Plant menu: ➤ Utility ➤ ISO Flag

Command line: ISOFLAG

This function sets a flag in the component indicating that the present stored length should be used regardless of what type of length modifications are made to the pipe component. This function only affects pipes and nipples. This is useful when adjusting a pipe which is to be used in an isometric. For example, if used prior to stretching a pipe which was 120'-0" long to 5'-0" long, the Bill of Material generator would reflect a 120'-0" long piece of pipe.

This flag is also set by the Manual-Auto Break (see page 332) function. The component's xdata will be flagged when using the CEDIT dialog. The lower left corner in the coordinates box toggle will have Iso flagged. The following prompt will appear.

Command: **ISOFLAG**

Select components to flag with ISO lengths...
Select objects: *select components*
Select objects: *enter to finish*

Insulation

Plant menu: ▶ **Utility** ▶ **Insulation**

Command line: INSULATION

This routine is used for adding insulation to a line. The user will be prompted for the pipe's outside diameter if a CADWorx component is not found at the start point.

Note: More advance insulation features can be access by using the InsulationView palette.

Command: **INSULATION**
[Remove] Select insulation start point:

Remove

This option removes data from the component that indicates insulation.

Select centerline of insulation to remove: *select component*

The final prompt will indicate that the user needs to remove the graphic manually.

Manually erase insulation from component...

Select insulation start point

This option adds data and graphics to an existing component. Only pipe components will show the insulation graphic. All other components, the insulation information is stored within the component.

Pick other point: *pick another point along centerline*
[Insulation only/Tracing only/Both] <Insulation only>:
Enter insulation thickness: *enter thickness (ex: 2")*
Enter insulation density in lbs/ft3 <11.5>: *enter density or enter*
Select one side of pipe: *pick one side of the pipe (prompt only shown when a CADWorx component is not selected)*
Select other side of pipe: *pick other side (prompt only shown when a CADWorx component is not selected)*
Attach to components with same line numbers [Yes/No] <Yes>: *(prompt shown if component with a line number is selected)*
Total = .. *(prompt shown if component with a line number is selected)*

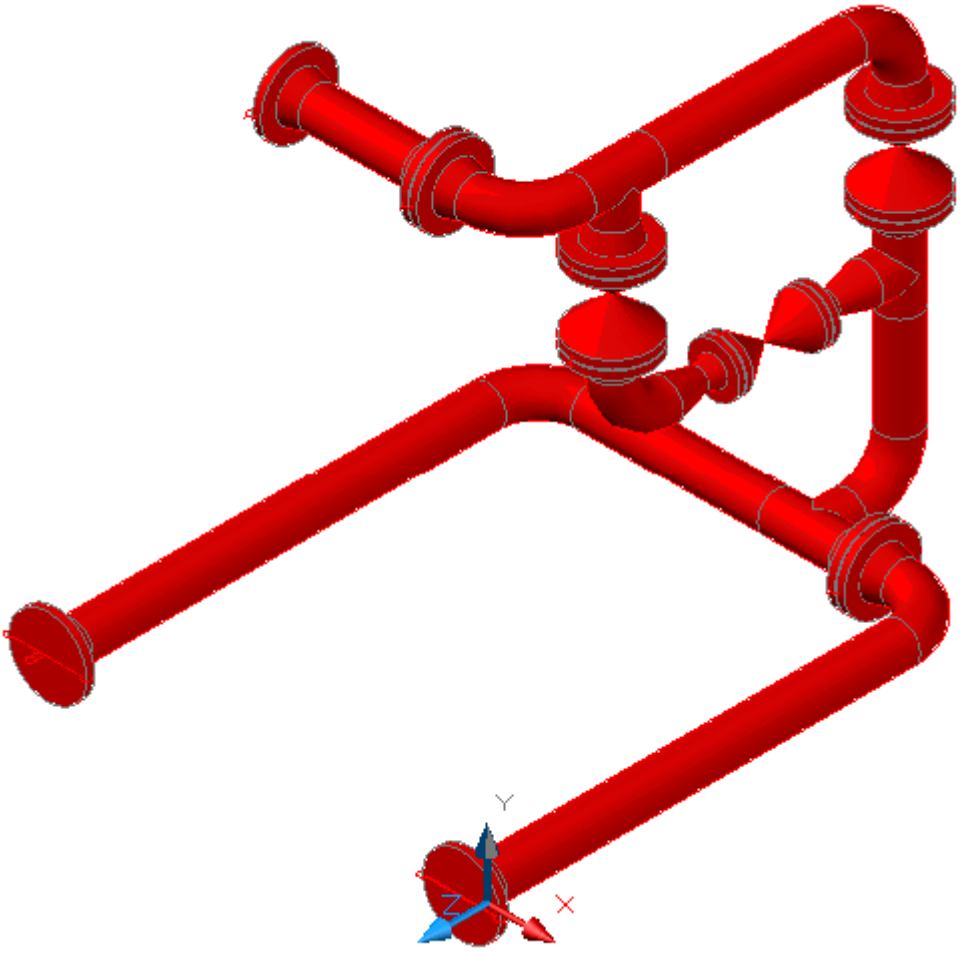
Insulation View Palette

Command line: INSULATIONVIEW

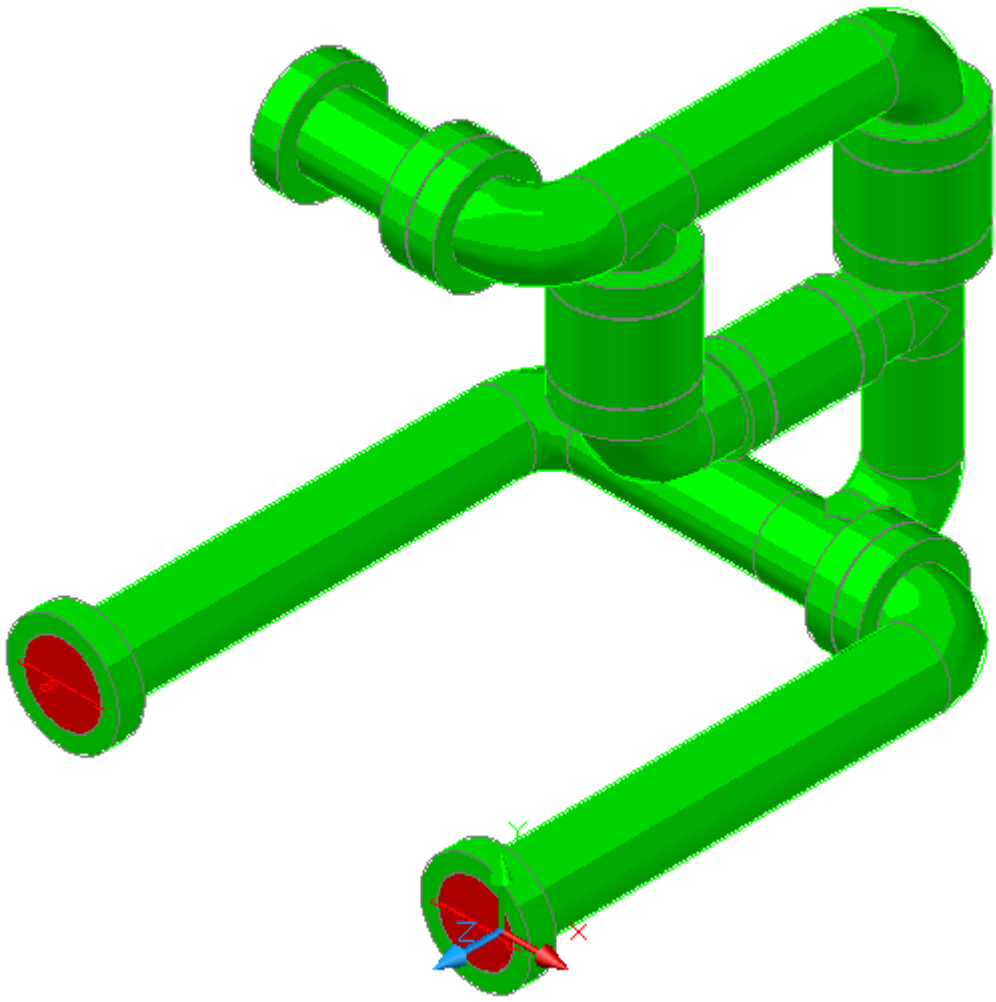
There are two tabs for the Insulation View: Settings and Apply

The Settings tab allows the user to create a data file with an extension ".inu". In this file the user can add, modify and remove insulation configurations that can be applied to CADWorx components. After the insulation configuration data file has been created, the Apply tab can be used to apply these configurations to the components selected.

Model:

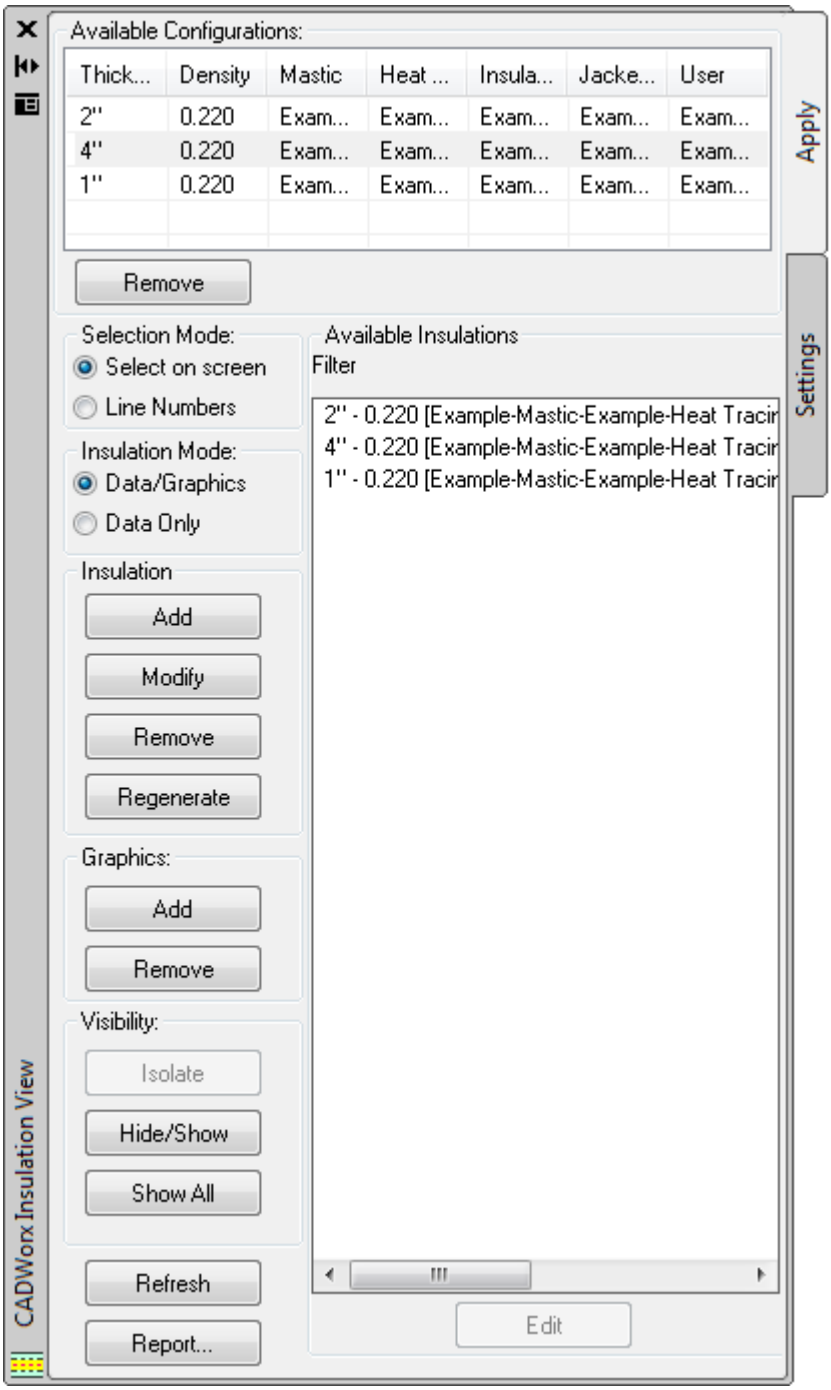


Insulation:



Apply Tab

After the insulation configuration data file has been created using the Settings tab, the Apply tab can be used to apply any of these configurations to CADWorx components.



Note: For most CADWorx components, insulation is approximated with the use of single or multiple AutoCAD cylindrical components.

Available Configurations - Grid: Displays the currently saved insulation configurations in the current AutoCAD drawing.

Remove: Removes the highlighted row in grid from the current drawing.

Selection Mode: There are two modes by which the user can apply insulation. All buttons in the dialog will be based on these two modes.

- Select on screen - select components in the drawing window.
- Line Numbers - select by line numbers

Insulation Mode: There are two options by which to apply insulation data and graphics.

- Data/Graphics - Add both the insulation graphic and data
- Data Only - Add only the insulation data and no graphic.

Insulation:

- Add - Adds the insulation currently selected in the "Available Configurations" grid to components in the drawings. The user will be prompted to select CADWorx components. If any of the selected components are already insulated, this action is skipped.
- Modify - Modifies existing insulation graphics and data.
- Remove - Removes existing insulations graphics and data.
- Regenerate - If insulation graphics were deleted or lost, this button can be used to generate the graphics from the insulation data stored in the components center line.

Graphics:

- Add - Adds insulation graphics to components that have had their graphics previously removed. The data previously stored in the components will be used to generate the graphics.
- Remove - Removes the graphics from selected components but insulation data will still remain. These components will not show the insulation solid even if the user later uses the regenerate option.

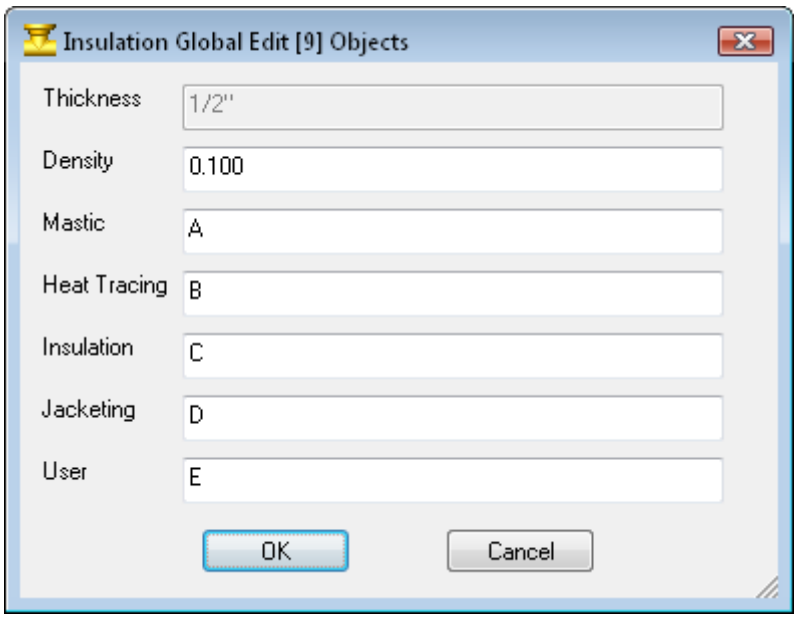
Visibility:

- Isolate - Isolates insulation graphics by either selecting components on screen or by selecting line numbers from the list.
- Hide/Show - Hides or shows the insulation graphics by selecting components on screen or by selecting line numbers from the list.
- Show All - Shows all objects, that may have been previously hidden, in the current drawing.

Refresh: Although all lists are dynamically updated as needed. This button is used to manually update the "Available Configurations" grid and the "Available Insulations" list.

Report: Creates a complete report in HTML format about all insulation component in the model. The report is itemized by the different types of insulation data and calculates the total weight, volume, and length.

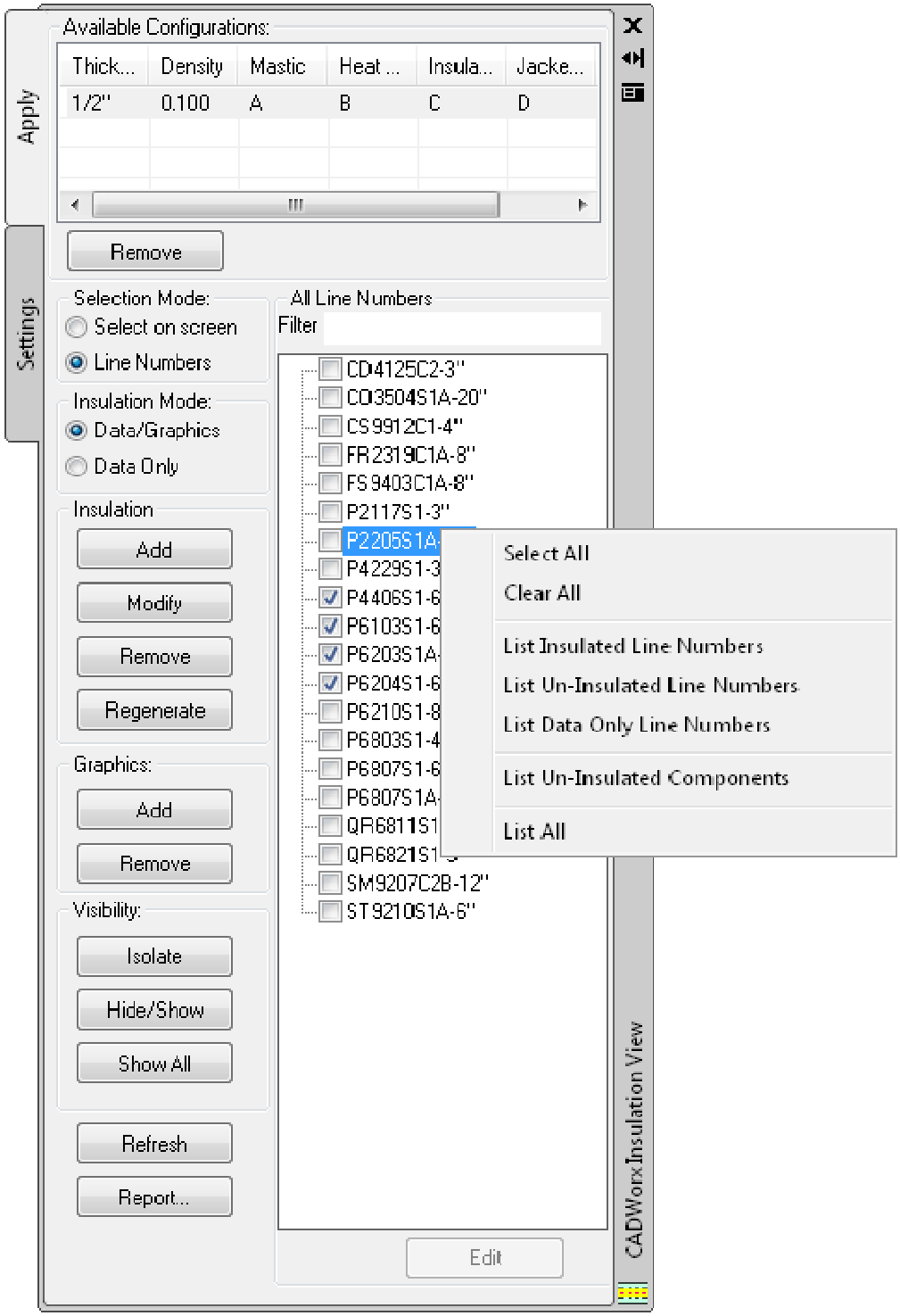
Edit: Globally edit the insulation data to all components having this particular insulation configuration. Using the dialog box below, changes can be made to all insulation data except the "Thickness" field.



After all fields have been edited and "OK" is clicked, all insulations with the same data will be updated as well as the insulation configurations saved in the drawing.

Filter: Filter the list of line numbers by using wild cards. For example, entering *25-1B* will shrink the line number list to only line numbers having 25-1B as part of its name. The text filter will occur regardless of where the text is in the line number name.

Right Click Menu: In Line Number selection mode, a right click menu is available with additional options. The right click menu will change what is displayed in the edit box. The figure below shows the edit box in the "Line Numbers" mode. In this mode, the user can insulate components by checking the line numbers in the list. All the buttons to the left of the edit box will function the same way except the user will not be asked to select components on the screen. If no line number is selected, clicking any of the buttons will result in a prompt for the user to select one object to get the line number. The corresponding operation will then be applied to all components that have that line number.


Apply
Settings
Available Configurations:
Thick...
Density
Mastic
Heat ...
Insula...
Jacke...
1/2"
0.100
A
B
C
D
Remove
Selection Mode:
Select on screen
Line Numbers
Insulation Mode:
Data/Graphics
Data Only
Insulation
Add
Modify
Remove
Regenerate
Graphics:
Add
Remove
Visibility:
Isolate
Hide/Show
Show All
Refresh
Report...
All Line Numbers
Filter
CD4125C2-3"
CD3504S1A-20"
CS9912C1-4"
FR2319C1A-8"
FS9403C1A-8"
P2117S1-3"
P2205S1A-
P4229S1-3
P4406S1-6
P6103S1-6
P6203S1A-
P6204S1-6
P6210S1-8
P6803S1-4
P6807S1-6
P6807S1A-
QR6811S1
QR6821S1
SM9207C2B-12"
ST9210S1A-6"
Edit
Select All
Clear All
List Insulated Line Numbers
List Un-Insulated Line Numbers
List Data Only Line Numbers
List Un-Insulated Components
List All
CADWorx Insulation View

Select All / Clear All: This will select or un-select all line numbers displayed in the list.

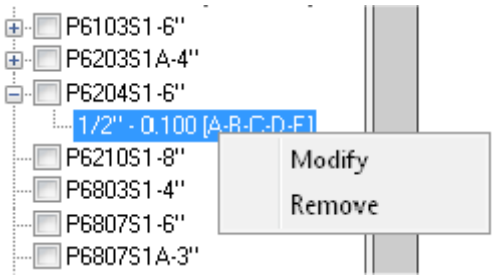
List Insulated Line Numbers: List only insulated line numbers.

List Un-Insulated Line Numbers: List only Un-Insulated line numbers.

List Data Only Line Numbers: List only line numbers that have insulation data but have no insulation graphics

List Un-Insulated Components: List only Un-Insulated components.

List All: This has the same functionality as the “Refresh” button.



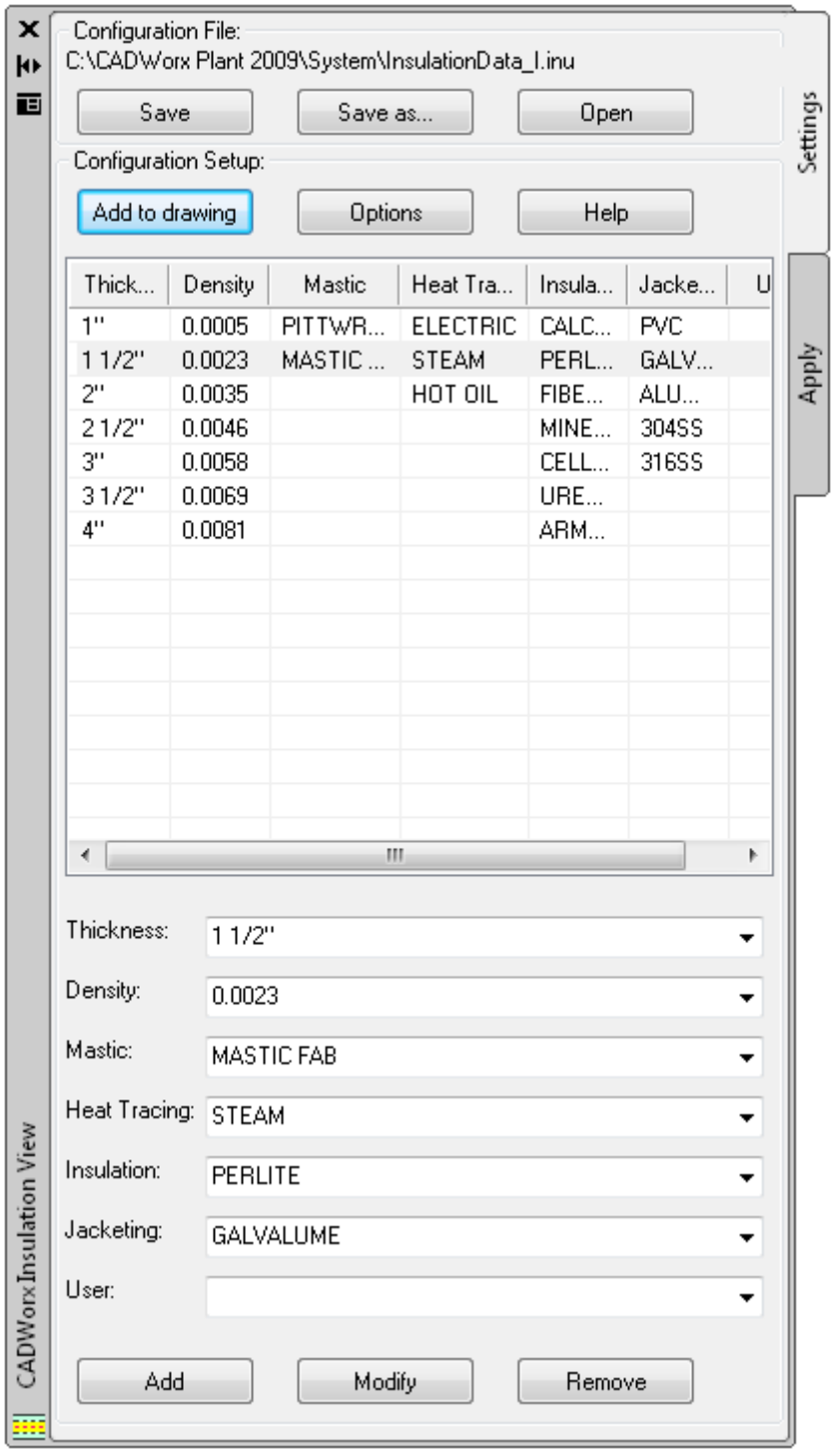
If any line number in the list has one or more insulation configuration, there will be a plus (+) sign displayed to left. When the user expands the list, the insulation data will be displayed. Right clicking an insulation configuration, a menu will appear.

Modify: Replaces the selected insulation data with the highlighted insulation in the “Available Configurations” grid. This change will apply to all components in the selected line number.

Remove: Removes this insulation from all components in the selected line number.

Settings Tab

The Settings tab allows the user to create a data file with an extension of ".inu". In this file the user can add, modify and remove insulation configurations that can be applied to CADWorx components.


Configuration File:
C:\CADWorx Plant 2009\System\InsulationData_I.inu
Save
Save as...
Open
Settings
Configuration Setup:
Add to drawing
Options
Help
Apply
Thick... Density Mastic Heat Tra... Insula... Jacke... U
1" 0.0005 PITTWR... ELECTRIC CALC... PVC
1 1/2" 0.0023 MASTIC ... STEAM PERL... GALV...
2" 0.0035 HOT OIL FIBE... ALU...
2 1/2" 0.0046 MINE... 304SS
3" 0.0058 CELL... 316SS
3 1/2" 0.0069 URE...
4" 0.0081 ARM...
Thickness: 1 1/2"
Density: 0.0023
Mastic: MASTIC FAB
Heat Tracing: STEAM
Insulation: PERLITE
Jacketing: GALVALUME
User:
Add
Modify
Remove
CADWorx Insulation View

Save: Save the current data in the "Configuration Setup" grid to the configuration file.

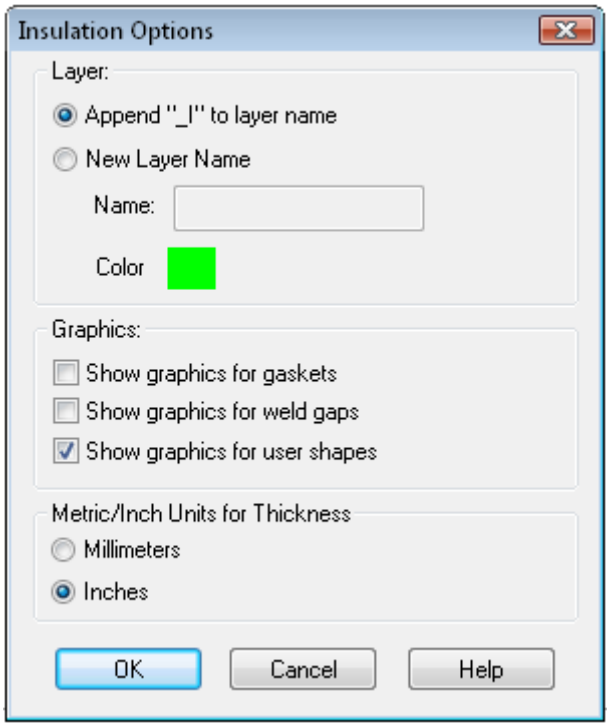
Save As: Brings up a file selection dialog to save the current data in the "Configuration Setup" grid to a new file.

Open: Brings up a file selection dialog to load an insulation data file. Once selected, the data contained in that file will be displayed in the "Configuration Setup" grid as well as listed individually in the drop down categories.

Configuration Setup - Grid: Displays the content of the insulation data file in grid form.

Add to drawing: Saves the grid selected row(s) to the current AutoCAD drawing. If similar data already exists in the current drawing, it will not be overwritten. All configuration that has been added, will appear in Apply tab of the palette.

Options: The Options button displays the dialog box shown below:



Layer:

Append "_I" to layer name: The insulation solid will be grouped with the component solid and center line. The insulation solid layer name is based on the layer of the solid with an "_I" appended to the end (Example: If solid is on layer 1"-150-CWS, the insulation solid will be on 1"-150-CWL_I)

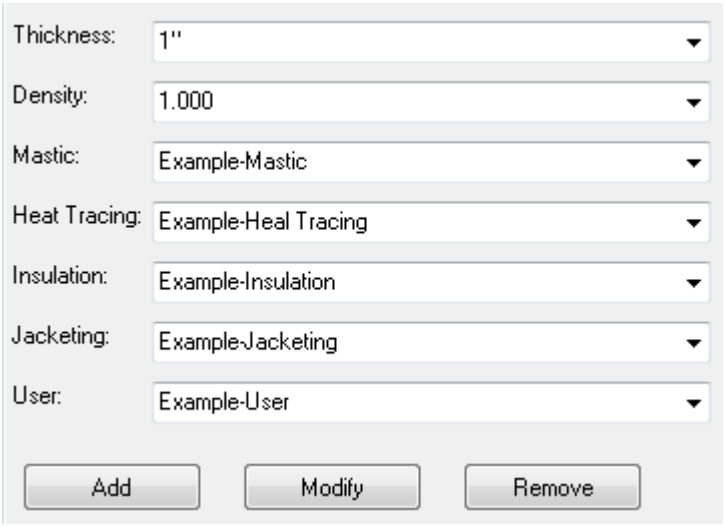
New Layer Name: All insulation solids will be placed on this layer.

Color: Starts a standard AutoCAD color selection dialog. This setting will set the default layer color.

Graphics: The option to insulate gaskets, weld gaps, and user shapes can be switched on or off.

Metric/Inch Units for Thickness: In mixed metric units (Imperial pipe size / metric dimensions), thickness can be entered in millimeter or inches.

Configuration Setup - Drop Down Lists



Thickness: Accepts only numeric values greater than zero. This value is the actual insulation thickness. Select an existing value, if applicable, or enter a new value. This parameter is required to create a insulation configuration.

- In Imperial units (Imperial pipe size / Imperial dimensions), enter a value in inches.
- In mixed metric units (Imperial pipe size / metric dimensions), enter a value in inches or millimeter based on the setting from the Options dialog. If inch values is entered, the program will automatically convert to millimeter when the insulation solid is created in the model. If millimeter value is entered, no conversion is done.
- In pure metric units (metric pipe size / metric dimensions), enter a value in millimeters.

Density: Accepts only numeric values greater than zero. The density will be used to calculate the insulation weight by multiplying it with the insulation volume. Select an existing value, if applicable, or enter a new value. This parameter is required to create a insulation configuration.

- In Imperial units (Imperial pipe size / Imperial dimensions), this value has units of lb / in^3.
- In mixed metric (Imperial pipe size / metric dimensions) the value has units of g / cm^3.
- In pure metric (metric pipe size / metric dimensions), the value has units of g / cm^3.

Mastic - Heat Tracing - Insulation - Jacketing - User: Each category represents a specific description of the insulation type being used. They accept any entry from the user. Entering a value for these is optional.

Add: Adds the current values from the drop down to the grid. If similar data exists, the value will not be added to the grid.

Modify: Modifies the currently selected row in the grid using the values currently shown in the drop down lists. Each drop down list corresponds to a column in the grid.

Remove: Deletes the currently selected row in the from the grid.

Note: The Add, Modify and Remove buttons will only change the data currently displayed in the grid. The actual data in the insulation file is not updated with the unless the user clicks "Save" or Save As".

Dim Adjustment

Plant menu: ▶ **Utility** ▶ **Dim Adjustment**

Command line: DIMADJUST

This routine provides a method of adjusting a dimension when the pipe segments of a particular run have been adjusted or modified with the ISO flag routine. This routine will work with associate dimensioning setvar “DIMASSOC” on. The routine will also adjust the pipe’s stored xdata length. This is preformed by using the dimension to adjust the pipe’s xdata. Therefore, the first prompt asks whether the user is adjusting the dimension, or pipe (adjust the pipe with Dimension option, adjust the dimension with Pipe option). Both options of this routine will read and write fractional characters used with the dimension routines.

Command: **DIMADJUST**
Pick correct length from **[Dimension/Pipe]** <Dimension>:

Dimension

This option provides a method of adjusting the pipe from an existing dimension. The user would select the dimension, then select the pipe’s centerline.

Pick dimension: *select the dimension to adjust pipe by*

For this option, only one prompt will appears. The user cannot select multiple pieces of pipe.

Select pipe centerline to be adjusted:
Select objects: *pick pipe centerline*
Select objects: *hit enter*
Adjusted pipe to 10'-2 1/2"

Pipe

This option adjusts the dimension from the stored length in the pipe’s xdata.

Pick reference points [**Line**]:

Line

Instead of having to pick two reference points with the option below, the user can use this option to select a line. The line must be the same length as the pipe distance. AutoCAD dimension lines stop at the back of the arrow; therefore would not be an accurate line to select.

Select line: *Pick a line*

Pick reference points

Pick two reference points. This the most commonly used option and can be used to pick each end of a pipe.

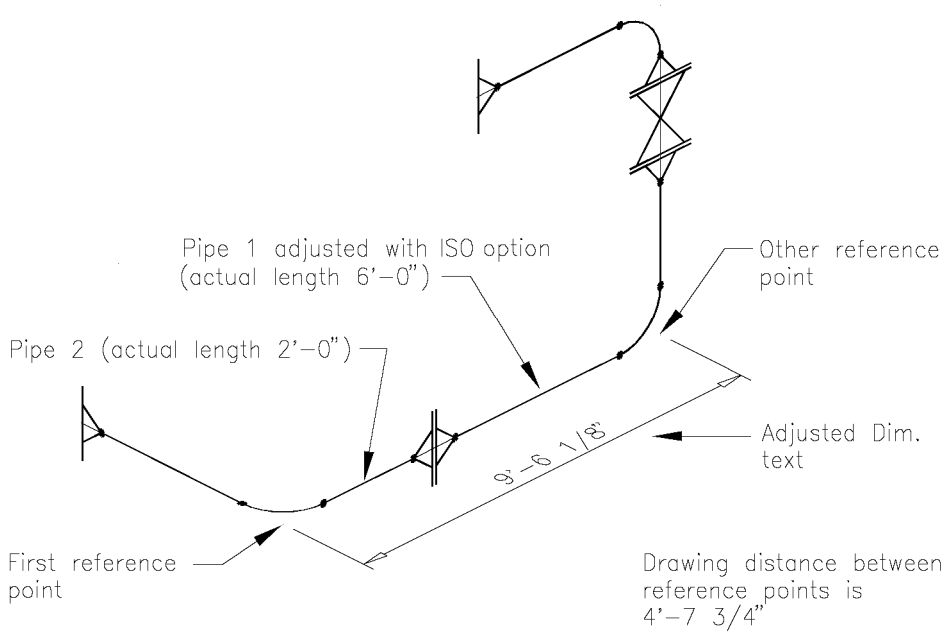
For this option, the following prompts requests the selection of the pipe(s) that were adjusted. Selection can be made by crossing, window or individually selecting the centerlines. Other components which might have been selected with crossing or window will simply be ignored. This will only detect adjusted pipe.

Select pipe centerline(s) that were adjusted...
Select objects: **C** *select all the components by crossing*
First corner: *pick point* Other corner: *pick another point*
20 found (not necessarily the number of pipe centerlines)
Select objects: *select more or hit enter to end*

A list will appear indicating the listed length (xdata) and the actual length (on drawing) for each pipe selected. It will prompt the "Dimensioned reference =" which is the distance between the two reference points selected above. The "Adjusting dimension by" will be the amount that the dimension needs to be increased (or decreased). The total of these two will be the new dimension text.

Pipe 1:
Listed length = 6'-0"
Actual length = 2'-0 5/16"
Pipe 2:
Listed length = 1'-0"
Actual length = 1'-1 5/16"
Dimensioned reference = 4'-7 3/4"
Adjusting dimension by = 4'-10 3/8"

Select dimension text...
Select objects: select dimension text
Select objects: hit enter



Line Isolate

Plant menu: ➤ **Utility** ➤ **Line Isolate**

Command line: LINEISOLATE

This function allows the user to isolate components based on their line numbers (if assigned) or their specifications. It will isolate an entire line number or it can be used to isolate one line number and include surrounding buildings or equipment. It can also turn off any isolation of components and/or surrounding equipment and building.

Command: **LINEISOLATE**
Enter an option **[Isolate/Partial isolate/Show all]** <Show all>:

Isolate

This option allows the user to isolate only components with particular specifications or line numbers. This option hides all objects including non-CADWorx Plant objects in the drawing. To make them visible again, run this command again and select the *Show all* option.

Enter an option **[Spec/Line number]** <Line number>:

Spec

This option list all specs used in the model from which the user can select one or more specifications to isolate.

Enter an option [**List**/**Select component**] <Select component>:

List

This option shows a dialog listing either the specfications or line numbers within the drawing. In this dialog, the user can select one or more items using standard Windows selection methods.

Select component

This option allows the user to select a component within the drawing.

Line number

This option list all line numbers used in the drawing from which the user can select one or more line numbers to isolate. This option will then present the same additional options as the *Spec* option above.

Partial isolate

This option works similar to the *Isolate* option above but it leaves the surrounding graphics (building, equipment, dimensions, etc.) visible. This option has the same additional options as the *Isolate* option above.

Show all

This option turns off any isolation and makes all objects in the drawing visible.

Line View Palette

Plant menu: ➤ **Utility** ➤ **Line Isolate Palette**

Command line: LINEVIEW

The Line View palette is comprised of two tabs: Line Isolate and Find.

Note: Line View will not process any entities that are on locked layers.

Line Isolate Tab

Plant menu: ▶ **Utility** ▶ **Line Isolate Palette**

Command line: LINEVIEW

The Line Isolate tab allows the user to isolate components based on their line number or their specification. The functionality available in this tab allows the user to isolate an entire line number, or to isolate one line number and include surrounding buildings or equipment.

The available toolbar buttons are: *Refresh List, Show All Components, Select Line Number by Component, Isolate Selected Line Numbers, Set Layer Options,* and *Help*.

- *Refresh List:* This option will refresh the line number list in the dialog. Use this option if new line numbers do not appear in the list.
- *Show All Components:* This option makes all objects in the drawing visible.
- *Select Line Number by Component:* This option allows the user to select a component within the drawing. The line number of this component will be used to isolate the model.
- *Isolate Selected Line Numbers:* This option will isolate the selected line numbers.
- *Set Layer Options:* This dialog allows the user to include/exclude certain layers when Isolate Type is set to Layer Isolate. By default, all layers are visible for line isolate. The user can exclude certain layers by selecting them in the dialog.

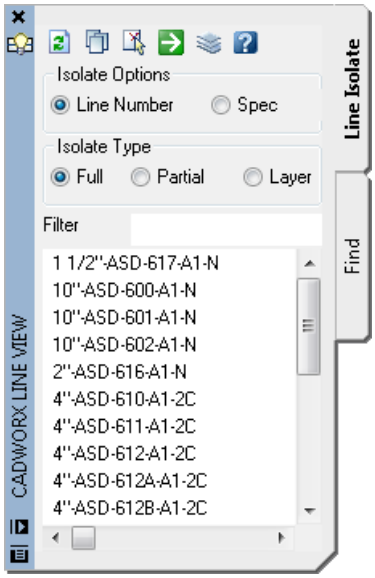
Isolate Options:

- *Line Number:* This option lists all line numbers used in the drawing from which the user can select one or more line numbers to isolate.
- *Spec:* This option lists all specs used in the model from which the user can select one or more specifications to isolate.

Isolate Type:

- *Full:* This option will display the selected CADWorx components and hide all non-CADWorx components.
- *Partial:* This option will display the selected CADWorx components and also show all non-CADWorx components (building, equipment, dimensions, etc.)
- *Layer:* This option will display the selected CADWorx components and hide all non-CADWorx components based on the layer setup.

Filter: For details on this option, review this topic in Appendix F (see page 647).



Right clicking on the line number list also displays the *Isolate Selected Line Numbers, Show All Components,* and *Refresh List* options.



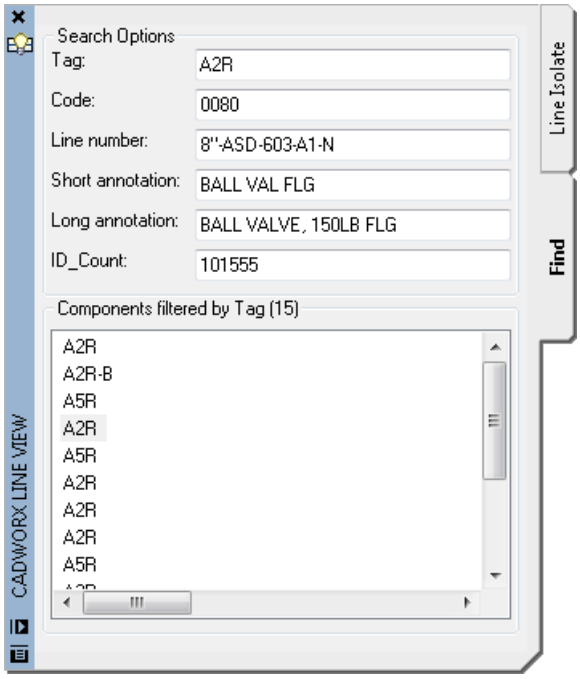
Note: If components have been hidden using the Line Isolate tab, these components will not be listed in the Find tab.

Find Tab

Plant menu: ▶ **Utility** ▶ **Line Isolate Palette**

Command line: LINEVIEW

The Find tab allows the user to locate components based on their description information. Clicking in the edit boxes will refresh the list of components. Entering text in each individual edit boxes will dynamically filter the list of available components based on the category that the edit box represents. If components have been hidden using the Line Isolate tab, these components will not be listed in the Find tab.



Right clicking on the list displays the *Zoom, CEdit,* and *Refresh* options.



Zoom: This option will zoom the current model space view to the component. Double clicking in the list of components will also zoom to the component. Zoom will not work with XREFs that have been inserted into the base drawing with a rotation.

CEdit: This option will start the CEdit (see page 303) dialog. CEdit on xref component will not allow any changes to the component descriptions

Refresh: This option will refresh the list in the dialog. Use this option if new components do not appear in the list.

Layer Control

Previous

Plant menu: ▶ Utility ▶ Layer Control ▶ Previous

Command line: LAYERRESTORE

This is used to set the current layer back to the previous layer when dimensioning or text has been used. It will only work after these two items are used. It will not restore a previous layer that has been changed with the Layer command or Ddlmodes.

Command: **LAYERRESTORE**

Change

Plant menu: ▶ Utility ▶ Layer Control ▶ Change

Command line: LAYERCHANGE

This is used to change an entity's layer by selecting an entity on another layer. The original entities' selection can be made by crossing a window or by selecting individually.

Command: **LAYERCHANGE**
Select entity to change layer...
Select objects: *select entity*

Select objects: *select more or enter to end*

Select entity with desired layer: *select entity*
Moved to _150 layer...

Match

Plant menu: ▶ Utility ▶ Layer Control ▶ Match

Command line: MATCH

This routine will change an object's properties by selecting another object. It will ask the user to select the object they want to change, then the object to match properties to.

Command: **MATCH**
Select entities to change:
Select objects: *select the object desired*

Select objects: *enter to complete selection*

Select entity to match: *select the object*

Set

Plant menu: ▶ Utility ▶ Layer Control ▶ Set

Command line: LAYERSET

This is used to set the current layer by selecting an entity. Individual selection is required.

Command: **LAYERSET**
Select entity layer to set: *select entity*
Layer set to _150...

Off

Plant menu: ▶ Utility ▶ Layer Control ▶ Off

Command line: LAYEROFF

This is used to turn a layer off by selecting an entity. Individual selection is required.

Command: **LAYEROFF**
Select entity layer to off: *select entity*
Layer _150 turned off...

On

Plant menu: ▶ Utility ▶ Layer Control ▶ On

Command line: LAYERON

This is used to turn a layer back on that was turned off with the routine above. No selection is required.

Command: **LAYERON**
Layer _150 turned on...

Isolate

Plant menu: ▶ **Utility** ▶ **Layer Control** ▶ **Isolate**

Command line: LAYERISOLATE

This is used to isolate a layer. Selecting an individual entity will turn all layers off, except the layer of the entity selected.

Command: **LAYERISOLATE**
Select entity layer to isolate: *select entity*
Layer _150 isolated...

Un-Isolate

Plant menu: ▶ **Utility** ▶ **Layer Control** ▶ **Un-Isolate**

Command line: LAYERUNISO

This is used to un-isolate a layer which has been turned off with the above routine. This will turn all the layers back on. No selection is required.

Command: **LAYERUNISO**
All layers turned back on...

Delete

Plant menu: ▶ **Utility** ▶ **Layer Control** ▶ **Delete**

Command line: LAYERDELETE

This is used to delete a desired layer. For example: deleting all the entities on layer 0. This will delete all lines, text, etc., on the layer selected. Individual selection is required.

Command: **LAYERDELETE**
Select entity layer to delete: select entity
Layer _150 deleted...

Selecting the “VIEWL” layer will generate the following warning message. This will delete a block called “CPM” which is inserted when beginning a new drawing. This block contains information (sizes, specification, etc.) that will be used when re-entering an existing drawing. If the user deletes this layer, this information will not be available for the next drawing session. The user can re-enter this information.

Cannot save setting for this drawing session if this layer is deleted...
Delete anyway **<No>**: *enter for no or type* **Y** *for yes*

Drawing Control

Fitting Width

Plant menu: ▶ **Utility** ▶ **Drawing Control** ▶ **Fitting Width**

Command line: FITTINGWIDTH

This routine controls the appearance of all flanged, socket weld and threaded fittings. For example; drawing single line socket weld fittings at 100% will not produce a legible component outline due to the hubs overlapping. This can be changed so that the component's hub will be drawn at 70% of the actual diameter. For double line components, the user should leave it set at 100%, unless, there is a special reason for reducing the hub.

Command: **FITTINGWIDTH**
Enter fitting width factor <100%>: *enter real (ex: 70 for 70%)*

Single Line Width

Plant menu: Utility ▶ Drawing Control ▶ Fitting Width

Command line: LINEWIDTH

Command: **LINEWIDTH**
Enter single line pipe width or **[Change]** <0.02>: *enter real or C*

Change

Use this option to change existing polyline widths already on the drawing.

Change polyline width for **[Drawing/Layer/Individual]** <Individual>:

Drawing

This option will change all the polylines on the drawing to a new width. It will not affect polylines in inserted blocks.

Enter new width for selected polylines: *enter real number*

Layer

This option will change all the polylines on a particular layer to the new width.

Enter name of layer: *enter layer name (ex: 150 or _150)*
Enter new width for selected polylines: *enter real number*

Individual polylines

This option will change individual polylines to the new width prompted for after the selection is finished. Selection can be achieved by crossing, window or picking.

Select objects: *select desired polylines*
Select objects: *enter to end selection*
Enter new width for selected polylines: *enter real number*

Enter single line pipe width

Enter single line pipe width <0.2000>: *enter real number*

This option controls the width of all single line components prior to insertion into the drawing. The use of this variable can control the single line flanges that are drawn based on this width (flange face lines drawn off the actual face location which allows for the gap between two adjoining faces). The user may require an adjustment to this gap. By resetting this width variable and using Mode Convert on the components, a new polyline width and a gap (based on this variable between the flanges) will occur.

Weld Size

Plant menu: ▶ **Utility** ▶ **Drawing Control** ▶ **Weld Size**

Command line: WELDSIZE

This routine enters the default size of the weld dots that accompany single line components entered into the drawing. It will also allow for the change of weld dots already placed in the drawing.

Command: **WELDSIZE**
Enter weld dot size for single line pipe or **[Change]** <1.00>:

Change

Change weld dot size for **[Drawing/Individual]**:

Drawing

Select to change all the weld dots on the entire drawing.

Individual weld dots

Use this option to change individual dots. Selection can be made by window, crossing or selecting individual weld dots.

Select objects: *select weld dots*
Select objects: *select more or enter to end*
Enter new size for weld dots: *enter real number*

Enter weld dot size for single line pipe

Enter weld dot size for single line pipe : *enter real*

If no weld dots were found, when using the change option, the following message will appear.

Weld dots not found or no weld dots selected...

Break Distance

Plant menu: ▶ **Utility** ▶ **Drawing Control** ▶ **Break Distance**

Command line: BREAKDIST

This variable is used in the Manual-Auto Break (see page 332) routine. When using the Break routine in the Auto mode, the distance that the intersecting pipes will be separated from each other will be provided. For example, if this distance was set to 2", the pipe above was a 2", and the pipe below was a 10"; using the Break routine on the pipe above will place a gap in the pipe 14.75" long (10.75 + 2 * 2.0).

Command: **BREAKDIST**
Enter break distance for AUTOBREAK <2.00> : *enter real*

3D Follow

Plant menu: ▶ **Utility** ▶ **Drawing Control** ▶ **3D Follow**

Command line: 3DFOLLOW

This will change the present UCS to a new plane and location with the insertion of every component. It will draw a single line extending out of the UCS icon. This single line indicates a positive Z direction. The user can use this indicator for determining which direction (Up or Down) to place the next component. It will allow the user to draw pipe in any direction X, Y, or Z without ever relocating the UCS. It will only work in 3D mode. Select 3D Follow to turn this mode on and off. This is for advanced users.

Command: **3DFOLLOW**
3D follow mode is set on...

Selecting the option again will turn it off.

Command: **3DFOLLOW**
3D follow mode is set off...

Single Line Pipe

Plant menu: ▶ **Utility** ▶ **Drawing Control** ▶ **Single Line Pipe**

Command line: LINEPIPE

When in the 2D double line mode, this toggle forces all pipe components (pipe only, not elbows, flanges, etc.) to be drawn in single line. This is very useful for engineering standards that require orthographic drawing be drawn this way. After selecting this toggle:

Command: **LINEPIPE**
Single line pipe will be forced...

Selecting the option again, will turn it off.

Command: **LINEPIPE**
Single line pipe will not be forced...

Elbow Centerline

Plant menu: ▶ **Utility** ▶ **Drawing Control** ▶ **Elbow Center Line**

Command line: ELBOWCL

This function toggles elbow centerlines for use with isometrics. When toggled on, it provides square elbow centerlines which can be used to dimension to. This also produces attractive isometrics.

Command: **ELBOWCL**
Centerlines will be drawn...

Selecting this option again will turn it off.

Command: **ELBOWCL**
Centerlines will not be drawn...

Auto Gasket

Plant menu: ▶ **Utility** ▶ **Drawing Control** ▶ **Auto Gasket**

Command line: AUTOGASKET

This function toggles automatic gaskets on or off. When on, this function will place a gasket at every component that has a flange face being placed last. For example, when placing a weld neck flange, insertion from buttweld end to face would place a gasket. From face to buttweld would not place a gasket.

Command: **AUTOGASKET**
Gaskets will be automatically placed...

Selecting this option again will turn it off.

Command: **AUTOGASKET**
Gaskets will not be automatically placed...

Node Placement

Plant menu: ▶ **Utility** ▶ **Drawing Control** ▶ **Node Placement**

Command line: NODEPLACE

This function toggles component node placement. When on, it will place a node at each end (and corner, if applicable) of the component as it's inserted into the drawing. This can be used with the NODe of Osnap function to properly direct dimensions or other drawing entities.

Command: **NODEPLACE**
Nodes will be placed at component ends...

Selecting this option again will turn it off.

Command: **NODEPLACE**
Nodes will not be placed at component ends...

Auto Insertion

Plant menu: ▶ **Utility** ▶ **Drawing Control** ▶ **Auto Insertion**

Command line: INSERTMODE

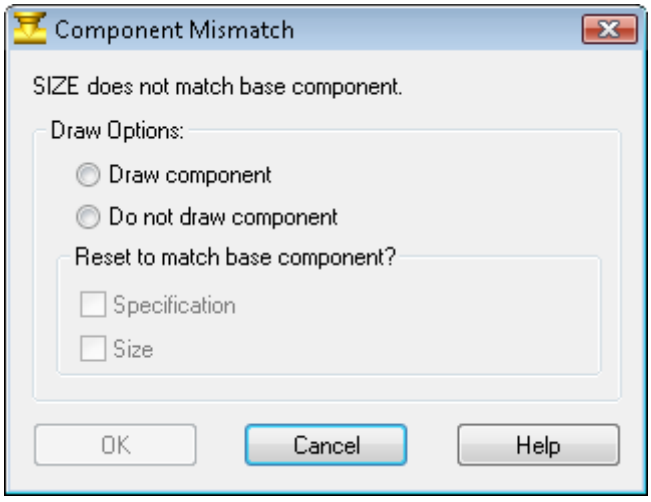
This function provides the capability of inserting components into existing pipe components. If two pieces of pipe components were drawn to a corner, this function could insert an elbow into these pipe pieces. A branch could be drawn from a main and then a tee could be inserted into both the main and the branch piping. This function automatically breaks the pipe and inserts the tee component. When placing a flange with a automatic gasket, the function will break the pipe component for both the flange and the gasket.

When selecting the function from the pulldown, there are three different options that will be available.

Command: **INSERTMODE**
Insertion mode ON and check dialog ON...

This default mode provides both insertion of components into pipe components and also checks to verify that the component is of the same size and/or specification. The pipe will require the save vector as the component being placed. If a component has two different vectors such as a tee, this function checks and breaks both pieces of pipe in the way of the component. If no pipe is detected, then the component will be placed as normal. The component and the pipe will require the exact same vectors. Although AutoCAD maintains points which have extreme accuracy, comparing vectors of different components still might not match due to variances in drawings. If this mis-match occurs, then this function will not break the pipe or even detect the pipe.

If the size and/or specification does not match, the following dialog appears and allows the option to skip or draw the component. If skipping the component is chosen, toggles for resetting the size and/or specification are available upon exiting the dialog. If the user requests that the size and/or specification be reset, then choose Skip, an enter could be issued and the component would restart with the desired size and/or specification. Shown below is the dialog from which the toggles may be executed.



Command: **INSERTMODE**
Insertion mode ON and check dialog OFF...

This mode provides insertion of components into pipe components, but it will not check the size and/or specification. It will just insert the component without any questions.

Command: **INSERTMODE**
Insertion mode OFF and check dialog OFF...

The third and final mode will not insert components into existing pipe components or provide any checking.

When the insertion mode is on, this function provides a distance correction feature. If a component is attached to the end of an existing pipe, this function will not interfere and the component will be placed as normal. If the component is attached to the pipe at a distance from the end less than or equal to the length of the component, then this function will just place the component in the required position and proceed. If the component is placed at a distance from the end greater then the length of the component, the following prompt will occur requiring an user response. If the distance displayed in the prompt is adequate, then an enter is all that is needed to proceed. The following prompt demonstrates a typical tee being placed.

Command: 4" TEE, STR. S/40, ASTM A-234 GR WPB

Pick start point or [Branch/Center] <last point>: *pick a point at the intersection of two pieces of pipe*
Pick main end direction: *pick a direction for the main run along the pipe*
Pick branch end direction: *pick a direction for the branch*

As indicated, the placement of the tee is the same as just picking a point without the presence of any pipe. The next prompt indicates that a pipe component was detected in the way of the component and then prompts for a distance from the end of the pipe. It would also show a hidden or highlighted line drawn from the closer end of the pipe to indicate which end the reference is from.

Enter distance or [**Round distance**] <1'-6 3/8">:

Enter distance

Enter distance <actual pick distance>: *enter a real or hit enter*

The user can enter either a new distance or just enter to accept the present value. Even though this distance might indicate 1’-6 3/8” (18.3750), it could be 18.3597. If the system was drawn at increments of 1/16, or the Snap option was set to 1/16”, this would not occur.

Round

This option will take the default distance (the actual distance picked from the end of the pipe) and round it to the nearest unit. This option uses the AutoCAD setvar command LUNITS to determine the nearest unit or measure and follows the same rules.

Note: If the graphics associated with the centerline (lines, or solids indicating the outline of the pipe, not the centerline) are not found, this function will prompt for selection so that they can be deleted.

Please select pipe outline graphics...
Select objects: *select the outline of the pipe (or graphics)*

This only deletes the component outline. Copying the solids, or double lines (outlines of pipe) will cause this problem. Also, breaking the two lines associated with a double line mode causes problems. Mode Convert will re-link the graphics with the component's centerline and will correct this problem.

This function uses a special selection process to find the correct component which filters out all the other entities. Sometimes the component insertion will not break the piping as expected. If this happens, turn off the layers of the entities that surround or overlap the centerline of the pipe.

Auto Connect

Plant menu: ▶ **Utility** ▶ **Drawing Control** ▶ **Auto Connect**

Settings toolbar:

Command line: CONNECTTOG

Auto Connect is activated by picking the button on the end of the settings toolbar. It can be disabled by picking the same button again.

Command: **CONNECTTOG**
Auto connection is activated...

This function will provide minimum prompts and make certain assumptions for the user. The first assumption to be made will be the location of the last component. The last component placed in the drawing will provide the starting point for the next component selected from the upper tablet menu. The direction of the last component will also be used to indicate the direction of the new component. When selecting components which require a second direction, such as elbows, tees, etc., there will be additional prompts requesting the direction. The typical elbow would prompt the following.

Pick direction: *pick a direction*

A pipe with an arbitrary length would prompt for an end point.

Pick ending point: *pick a point*

When a component is selected such as an elbow pointing up or down the prompt would require a response of up or down.

Enter direction [Up/Down] <Up>: *D for down and enter for up*

This function is extremely useful for placing components down a linear line. It can dramatically decrease the time required to place components in assemblies such as control valve bypass loops and others.

If any type of Mode Convert has been run, the last point will no longer be available. This will produce the following prompt.

Point not available. Please pick point:

This will require the user to pick the starting point.

Note: Auto Connect does not work in conjunction with Auto Route (see page 293) or Auto Weld Gap (see page 51).

Auto Weld

Plant menu: ▶ **Utility** ▶ **Drawing Control** ▶ **Auto Weld**

Command line: AUTOWELD

This function toggles automatic weld gaps on for buttweld only, on for buttweld and socket weld, and off. This function tries to place a weld gap on every end of any buttweld or socket weld component. There are three different situation which occur when the weld gaps are placed.

The first condition is when weld gaps are automatically placed on the ends of a buttweld or sock weld components and there is nothing but space around it. In this situation, a weld gap is placed and the next component can easily be placed at the weld gap by simply hitting a return or picking the end point of the weld gap symbol.

The second condition is when a component (such as a tee) is inserted into a pipe and the weld gaps are placed at each end of the tee. Here the weld gaps automatically break and cut back the pipe on both sides of the tee. It would also break the branch connection pipe if a one existed there.

The third condition is when a component is placed up against something which cannot be cut back (such as a flange against an elbow). Here, the end on the elbow must have existing a weld gap to place the flange to. If not, the flange will be placed against the elbow without any gap provided. It is the responsibility of the user to place a weld gap prior to trying to place a flange as such. Most often, when weld gaps are on, the elbow should have a weld gap automatically placed.

Selecting this option the first time turns automatic weld gaps on for buttweld only.

Command: **AUTOWELD**
Weld gaps will be placed for buttweld only...

Selecting this option again will turn it on for both buttweld and socket weld components.

Weld gaps will be placed for buttweld and socket weld...

Selecting this option one more time will turn weld gaps off.

Weld gaps will not be placed...

Note: See AutoWeldGap (see page 51) in the configuration file.

Note: The following information only applies to ISOGEN and weld gap components: If "Misc" BOM Item Type (see page 303) is set in CEDIT (see page 303) for weld gaps, ISOGEN will represent this weld gap as a "Field Fit Weld". Use ISOGEN Option Switch 22 to increase cut piece add on allowances for field fit welds.

Note: Auto Weld does not work in conjunction with Auto Connect (see page 366) or Auto Route (see page 293)

Group On/Off

Plant menu: ▶ **Utility** ▶ **Drawing Control** ▶ **Group On/Off**

Command line: GROUPTOG

Settings toolbar: 

This command toggles the grouping of the centerline of the component with the accompanying graphics. It is useful for selecting any piece of the component rather than only the centerline. By default, grouping is turned on.

Running the command once turns grouping off.

Grouping turned off...

Running it once again turns grouping on again.

Grouping turned on...

Select Control

Plant menu: ▶ **Utility** ▶ **Drawing Control** ▶ **Select Control**

Command line: SELECTCONTROL

Settings toolbar: 

This toggle controls whether grip points are displayed on solids. To change the length of pipe or steel, the user can grip stretch the pipe or steel center line. However, it is difficult to select the center line endpoint grip for steel shapes. Whit this toggle on, the grips on solids will not appear thus making it easier to grip stretch components.

Command: **SELECTCONTROL**
Grips will not be displayed for steel and pipe solids...

Selecting this toggle again will turn grips on for solids.

Grips will be displayed for steel and pipe solids...

System Visibility

Plant menu: ▶ **Utility** ▶ **Drawing Control** ▶ **System Visibility**

Command line: SYSVIS

This toggle controls whether certain components usually located on the system layer are visible are not. The components affected by this command are:

- Weld gaps
- Pads / Saddles
- Bolts
- Gaskets

This command will toggle the visibility of the above components.

Command: **SYSVIS**
Setting system visibility OFF...

Selecting this toggle again will turn grips on for solids.

Setting system visibility ON...

MVSetup

CADWorx Plant uses the MVSETUP.LSP supplied by AutoCAD which can be found in the pulldown menus. We have also supplied a stripped down version of the MVSETUP.DFS already created in the SUPPORT directory. If this file is deleted, the program will re-create it.

The MVSETUP.DFS can be modified to include any user title blocks. CADWorx Plant uses these title block with the Setup (see page 18) function described earlier. The MVSetup routine has options that allow the user to add and delete title blocks. CADWorx Plant has supplied its own title blocks for convenience. Our title blocks have attributes attached for ease of labeling.

Zoom Lock

Plant menu: ▶ **Utility** ▶ **Zoom Lock**

Command line: ZOOMLOCK

Settings toolbar: 

This function is a utility that will lock the AutoCAD Zoom command to allow zooming in paper space only. This means while the user is in paper space, with the Mspace command activated (the cursors are in the Mview), the user can zoom and it will not resize the model space viewport. Instead, this command will intercept the ZOOM command and change to paper space prior to zooming. When finished zooming, it will return the cursor and environment back to the Mview. Manually, the user would have to change to Pspace, zoom, and then change back to Mspace. The latter is an annoying process.

This command makes working on CADWorx Plant isometrics really easy. The automatic isometric generator will automatically leave the drawing with this command on when it finishes an isometric.

Command: **ZOOMLOCK**
Paperspace zoom lock toggled on...

Hit the command again and the ZOOMLOCK command will be turned off.

Paperspace zoom lock toggled off...

This command is very similar to the new Mview lock option.

Note: If running CADWorx Equipment, run the command EQUIPZOOMLOCK

Zoom Factors

This pulldown menu allows the user to zoom an Mview to a desired scale. This function will zoom to a perfect scale for the user's current Mview. If the user executes this function from model space (tilemode 1), it will produce a warning that tilemode has to be set to 0 (paper space). After selecting the scale from one of the menus, a prompt will appear requesting the center of the desired view.

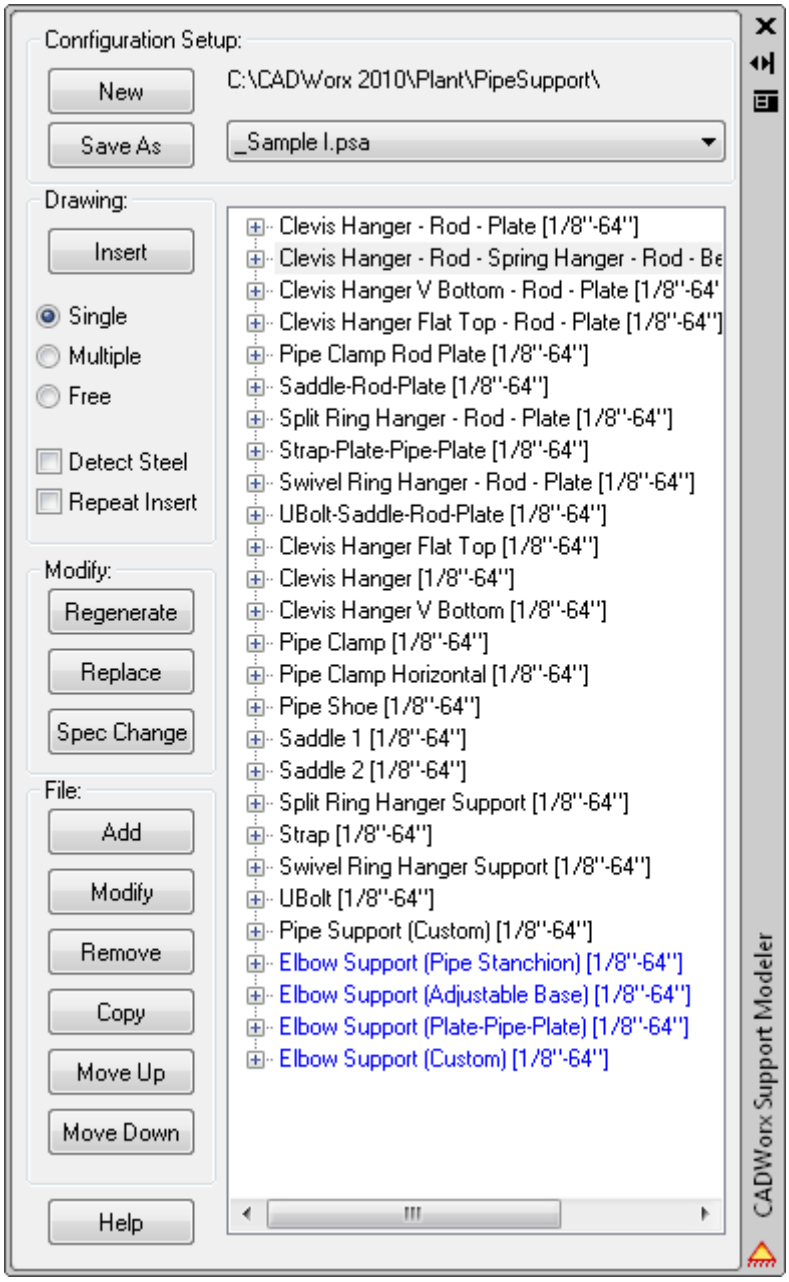
Command: **ZOOMF**
Pick center point: *pick view center*

Pipe Support Modeler Palette

Plant menu: ➤ Palettes ➤ Pipe Support Modeler

Command line: **PIPESUPPORTVIEW**

This command will start the Pipe Support Modeler palette. This palette can be used to insert pipe and elbow support components. Upon running the command, a palette is displayed as shown below. Supports that are set up for pipe components are shown in black. Supports that are setup for elbow components are shown in blue.


Configuration Setup:
New
Save As
C:\CADWorx 2010\Plant\PipeSupport\
_Sample I.psa
Drawing:
Insert
Single
Multiple
Free
Detect Steel
Repeat Insert
Modify:
Regenerate
Replace
Spec Change
File:
Add
Modify
Remove
Copy
Move Up
Move Down
Help
Clevis Hanger - Rod - Plate [1/8"-64"]
Clevis Hanger - Rod - Spring Hanger - Rod - Be
Clevis Hanger V Bottom - Rod - Plate [1/8"-64'
Clevis Hanger Flat Top - Rod - Plate [1/8"-64"]
Pipe Clamp Rod Plate [1/8"-64"]
Saddle-Rod-Plate [1/8"-64"]
Split Ring Hanger - Rod - Plate [1/8"-64"]
Strap-Plate-Pipe-Plate [1/8"-64"]
Swivel Ring Hanger - Rod - Plate [1/8"-64"]
UBolt-Saddle-Rod-Plate [1/8"-64"]
Clevis Hanger Flat Top [1/8"-64"]
Clevis Hanger [1/8"-64"]
Clevis Hanger V Bottom [1/8"-64"]
Pipe Clamp [1/8"-64"]
Pipe Clamp Horizontal [1/8"-64"]
Pipe Shoe [1/8"-64"]
Saddle 1 [1/8"-64"]
Saddle 2 [1/8"-64"]
Split Ring Hanger Support [1/8"-64"]
Strap [1/8"-64"]
Swivel Ring Hanger Support [1/8"-64"]
UBolt [1/8"-64"]
Pipe Support (Custom) [1/8"-64"]
Elbow Support (Pipe Stanchion) [1/8"-64"]
Elbow Support (Adjustable Base) [1/8"-64"]
Elbow Support (Plate-Pipe-Plate) [1/8"-64"]
Elbow Support (Custom) [1/8"-64"]
CADWorx Support Modeler

Configuration Setup: This section of the palette allows for creating new Pipe Support Assembly (*.psa) files. The PSA file saves the various support assembly configuration created. The path to the PSA file is set from the current configuration file setting of PipeSupportDirectory. The drop down list will show all PSA files that are available in the directory.

Drawing:

Insert button: This button will insert the support selected in the list. Based on the type of support selected (pipe or elbow), the AutoCAD command line will ask to select the appropriate CADWorx component (pipe or elbow)

Pipe type support prompts:

Select a pipe: *(select a pipe)*
Specify insertion point: *(specify an insert point along the pipe)*
Specify direction angle: *(specify a rotation around the pipe and the length of the support)*

Elbow type support prompts: Elbow support can be located at the corner point or the center line mid point. Center line mid point is the default for an elbow that is in the world x-y UCS plane (flat). Corner point is the default for an elbow not in the world x-y UCS plane (vertical).

Select elbows:
Select objects: *(select elbows)*
Start point is set to elbow corner point. *(elbow support will be located at corner point)*
Specify end point [MidPoint]: *(specify the support rotation / length or change insertion to center line mid point)*

Start point is set to elbow midpoint point. *(elbow support will be located at center line mid point)*
Specify end point [Corner]: *(specify the support rotation / length or change insertion to corner point)*

Single / Multiple / Free radio button: These options are only available for pipe type supports.

Single option will insert one support onto the pipe.

Free option will insert a support without selecting a CADWorx component. The support will be inserted using the current UCS settings.

The multiple option will prompt for start point, end point, and support spacing.

Select a pipe: *(select a pipe)*
Specify start point: *(select a start point along the pipe)*
Specify end point: *(select an end point along the pipe)*
Specify support spacing: *(specify the support spacing)*
Specify direction angle: *(specify a rotation around the pipe and the length of the support. All support will have the same rotation)*
5 supports of 5 have been successfully regenerated

Detect Steel check box: This option will detect CADWorx Steel components to set the correct size for the beam clamp if the beam clamp is used as the last component in the pipe support assembly.

Repeat Insert check box: This option will repeat the insert command for pipe type supports when in the single or free insert mode.

Modify:

Regenerate button: This button will ask to select support components from the model and recreate the support graphics based on what is set in the current PSA file. Use the regenerate button any time you need to recreate a support based on a new assembly sequence or if the parametric values for the support have changed since the support was originally inserted into the model.

Replace button: This button will replace the selected components in the model with the support that is selected from the list.

Spec Change button: This button will update the spec information for the support based on what is currently set in the PSA file.

File:

Add button: This button will start a dialog that will allow you to add a new assembly to the PSA file.

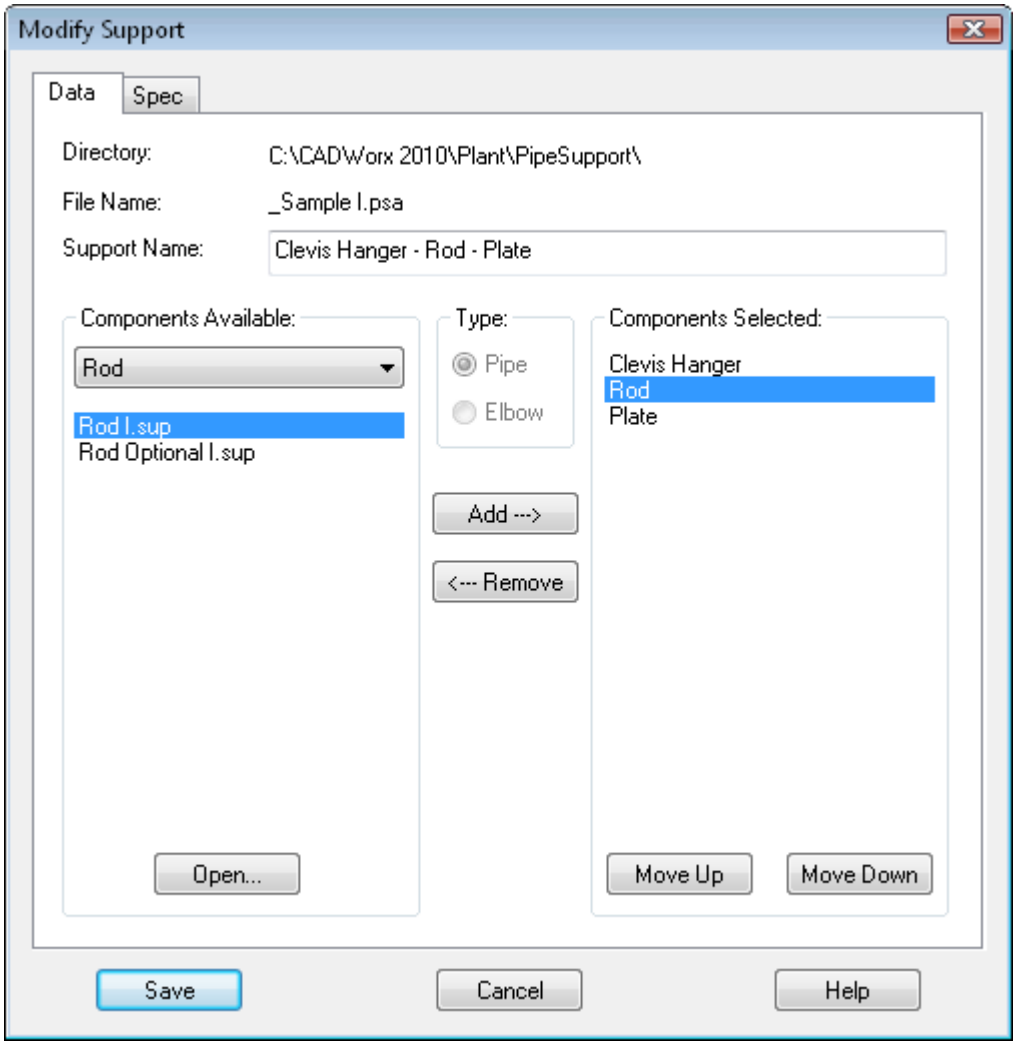
Modify button: This button will start a dialog that will allow you to modify a new assembly to the PSA file.

Remove button: This button will remove an assembly from the PSA file

Copy button: This button will copy an assembly within the PSA file

Move Up / Down buttons: These buttons will move an assembly up or down within the available support list.

Add or Modify Support: The following dialog will be displayed when you add a new support assembly or modify an existing support assembly.

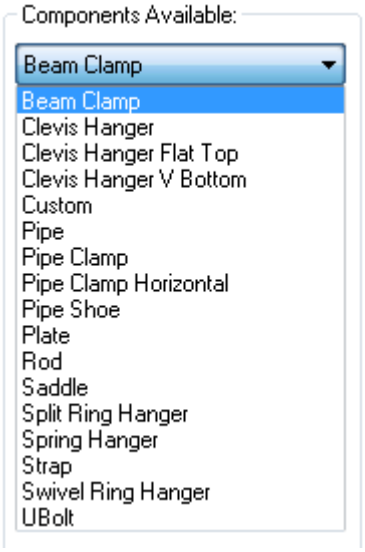

Modify Support
Data
Spec
Directory:
C:\CADWorx 2010\Plant\PipeSupport\
File Name:
_Sample I.psa
Support Name:
Clevis Hanger - Rod - Plate
Components Available:
Rod
Rod I.sup
Rod Optional I.sup
Open...
Type:
Pipe
Elbow
Add --->
<--- Remove
Components Selected:
Clevis Hanger
Rod
Plate
Move Up
Move Down
Save
Cancel
Help

Data tab: This section of the dialog will allow the creation of pipe support assemblies.

Support Name edit box: This edit box allows you to set the name of the support. This name will appear in the palette. In modify mode, you can rename a support assembly using this edit box.

Components Available drop down list: This drop down will show the different support components that can be used to make an assembly. After selecting a type, the list box will show the the parametric data files (*.sup) for the that type of support that are available. For elbow type supports, only Custom, Pipe, Plate, Rod, and Spring Hanger are available.

Open button: This button will open the select parametric data file (*.sup) selected with NOTEPAD.EXE

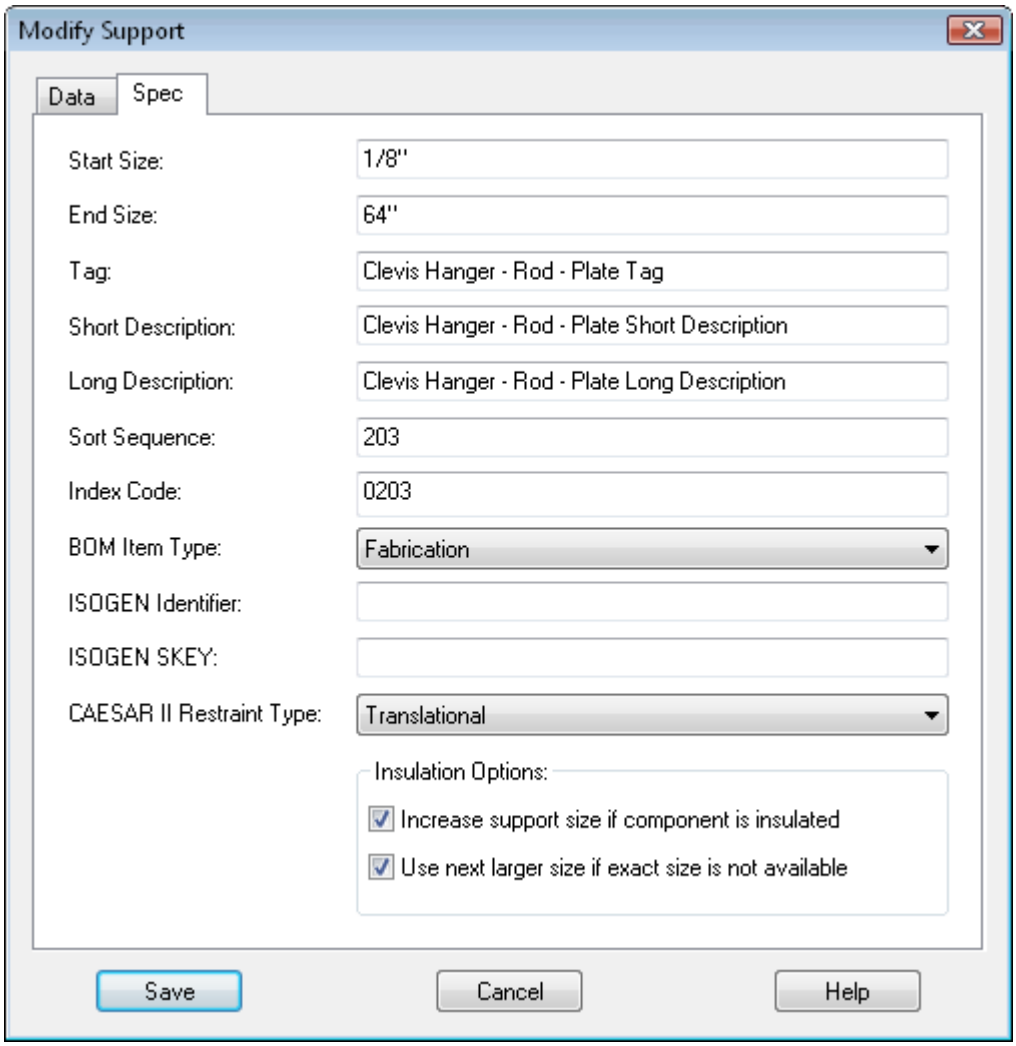


Type Pipe or Elbow radio button: These radio buttons will allow you to set the type of support (pipe or elbow). This option is only available when the support is first created.

Add and Remove button: These button will add or remove a component from the available list to the selected list.

Components Select list box: This box will list the componentS in the assembly. The support will be built using the selected components in the order shown.

Move Up / Down buttons: These buttons will move a component up or down in the assembly sequence.



Spec tab: This section of the dialog will allow setting the specification information for the support component. See the Spec Edit (see page 261) section of the help for more information on the edit boxes in this dialog.

Insulation Options: If the component is insulated, these options can be used to automatically select a larger size support component based on the insulation thickness. For example: A 6 inch nominal size (6.625 inch actual OD) has 4 inch thick insulation, 14.625 OD (6.625 actual OD + 8 insulation OD) support will be looked up in the data file. If a 14.625 OD is not available in the data file, the next larger size support can be used if the option is selected.

Support Parametric Data Files (*.sup):

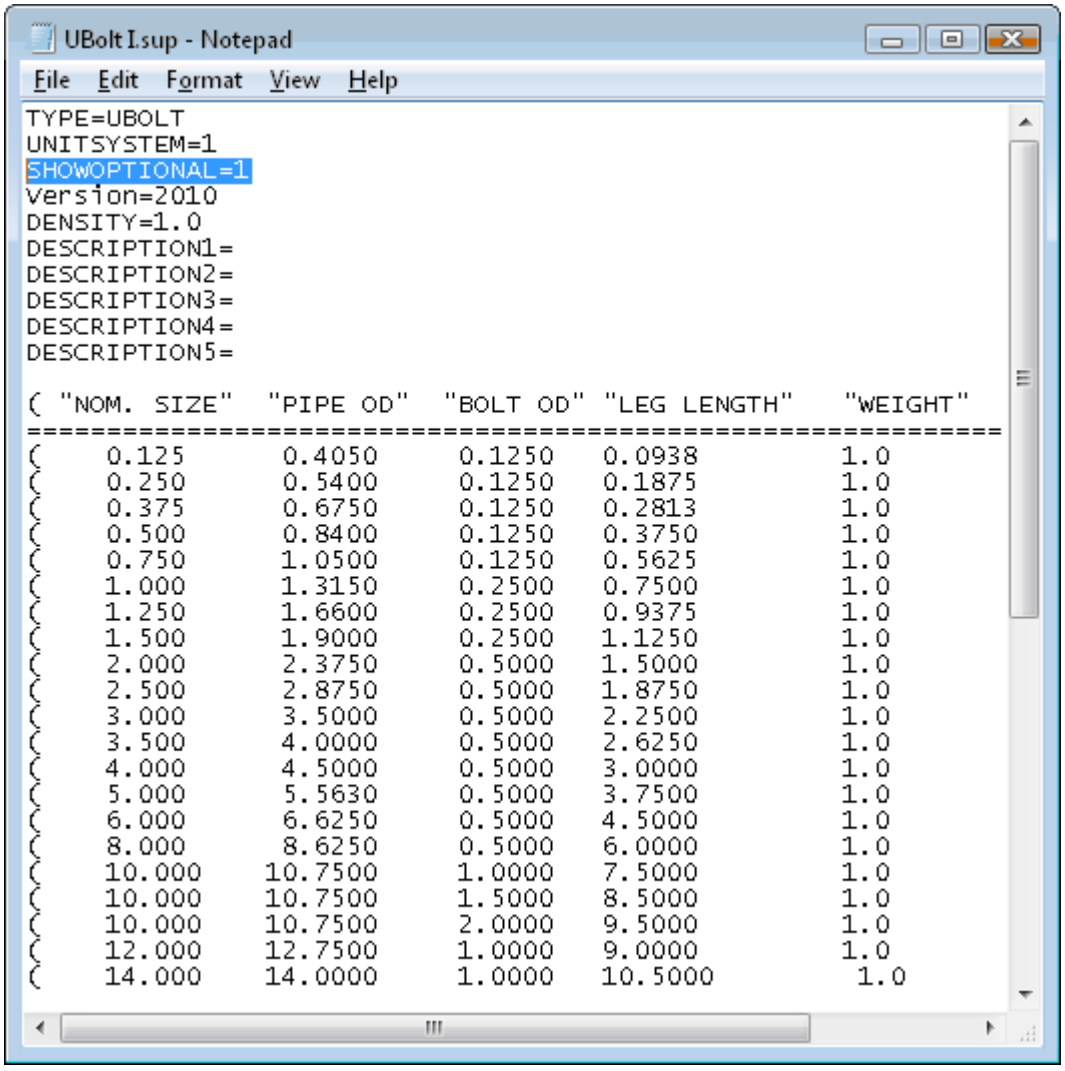
There are several types of support components that can be used to make an assembly. Each type has a data file which lists the parametric data needed to build the support. Sample data files are located in the ..\CADWorxPlant\PipeSupport\ directory. For each type, a sample drawing file is located in the ..\CADWorxPlant\PipeSupport\ directory that shows how the data file parameters are used to draw the shape. For custom (TYPE=CUSTOM), a drawing file cant be created for each size.

The top of the SUP file will have information about the support type (TYPE=) and unit system of the file.

The SHOWOPTIONAL setting can be used to control how different sizes are selected when the support is inserted.

SHOWOPTIONAL = 0 : The exact size match will be used if found in the data file. The optional selection dialog will not be shown.

SHOWOPTIONAL = 1 : If multiple sizes are listed, a dialog will be shown to select a size. In the example image below, all 10" U Bolts will be shown in the dialog for selection.

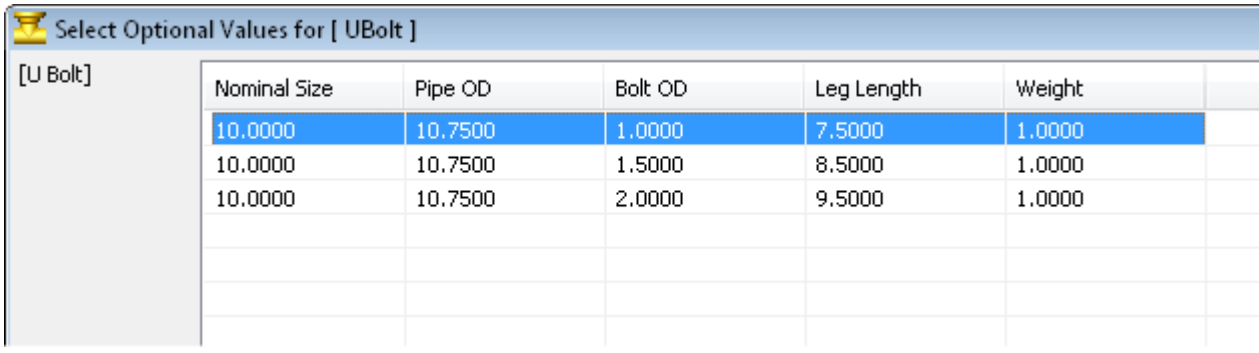


UBoltI.sup - Notepad

File Edit Format View Help

```
TYPE=UBOLT
UNITSYSTEM=1
SHOWOPTIONAL=1
Version=2010
DENSITY=1.0
DESCRIPTION1=
DESCRIPTION2=
DESCRIPTION3=
DESCRIPTION4=
DESCRIPTION5=

( "NOM. SIZE"   "PIPE OD"   "BOLT OD" "LEG LENGTH"    "WEIGHT"
=============================================================
(     0.125       0.4050      0.1250   0.0938           1.0
(     0.250       0.5400      0.1250   0.1875           1.0
(     0.375       0.6750      0.1250   0.2813           1.0
(     0.500       0.8400      0.1250   0.3750           1.0
(     0.750       1.0500      0.1250   0.5625           1.0
(     1.000       1.3150      0.2500   0.7500           1.0
(     1.250       1.6600      0.2500   0.9375           1.0
(     1.500       1.9000      0.2500   1.1250           1.0
(     2.000       2.3750      0.5000   1.5000           1.0
(     2.500       2.8750      0.5000   1.8750           1.0
(     3.000       3.5000      0.5000   2.2500           1.0
(     3.500       4.0000      0.5000   2.6250           1.0
(     4.000       4.5000      0.5000   3.0000           1.0
(     5.000       5.5630      0.5000   3.7500           1.0
(     6.000       6.6250      0.5000   4.5000           1.0
(     8.000       8.6250      0.5000   6.0000           1.0
(     10.000     10.7500      1.0000   7.5000           1.0
(     10.000     10.7500      1.5000   8.5000           1.0
(     10.000     10.7500      2.0000   9.5000           1.0
(     12.000     12.7500      1.0000   9.0000           1.0
(     14.000     14.0000      1.0000   10.5000           1.0
```

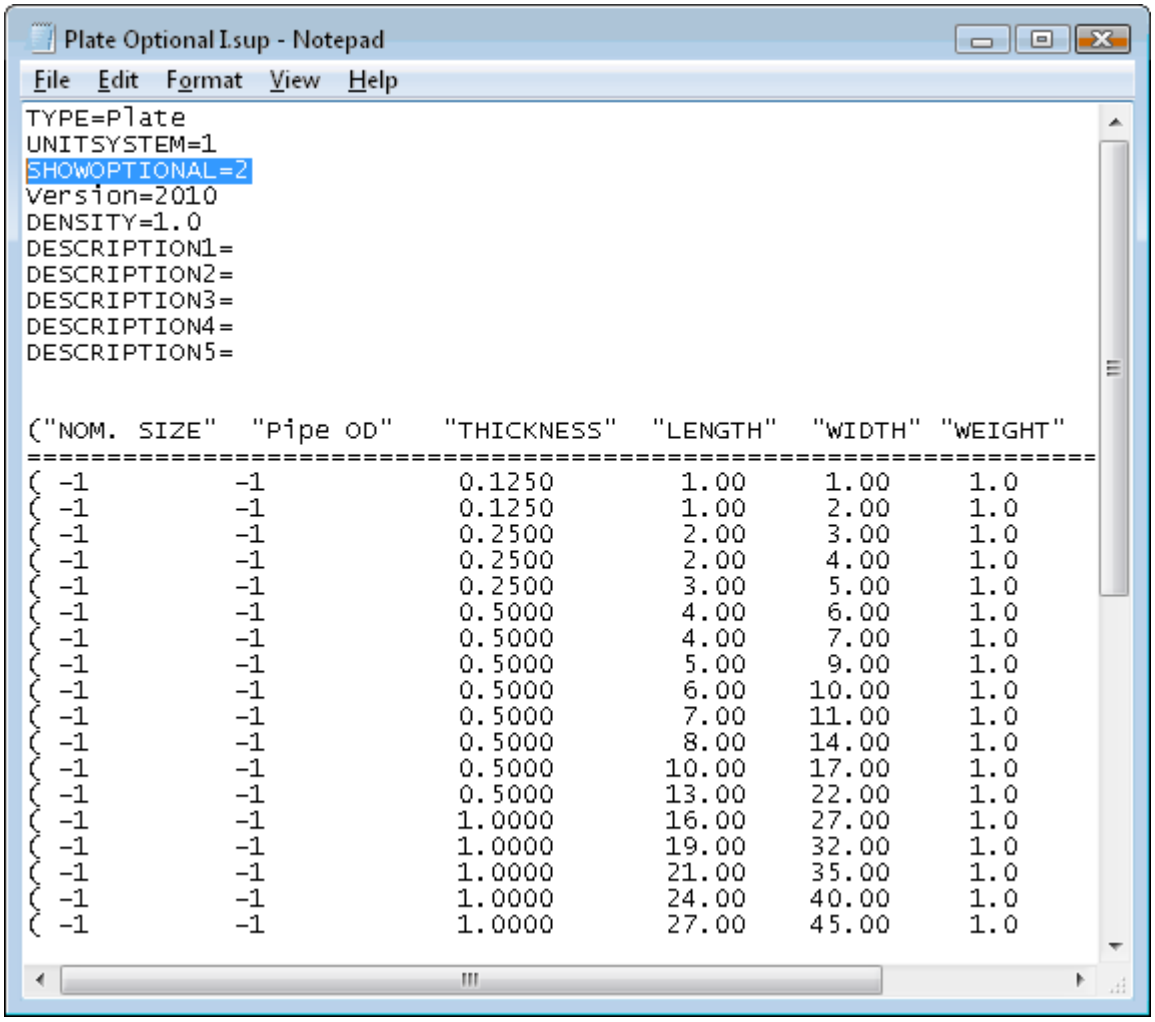


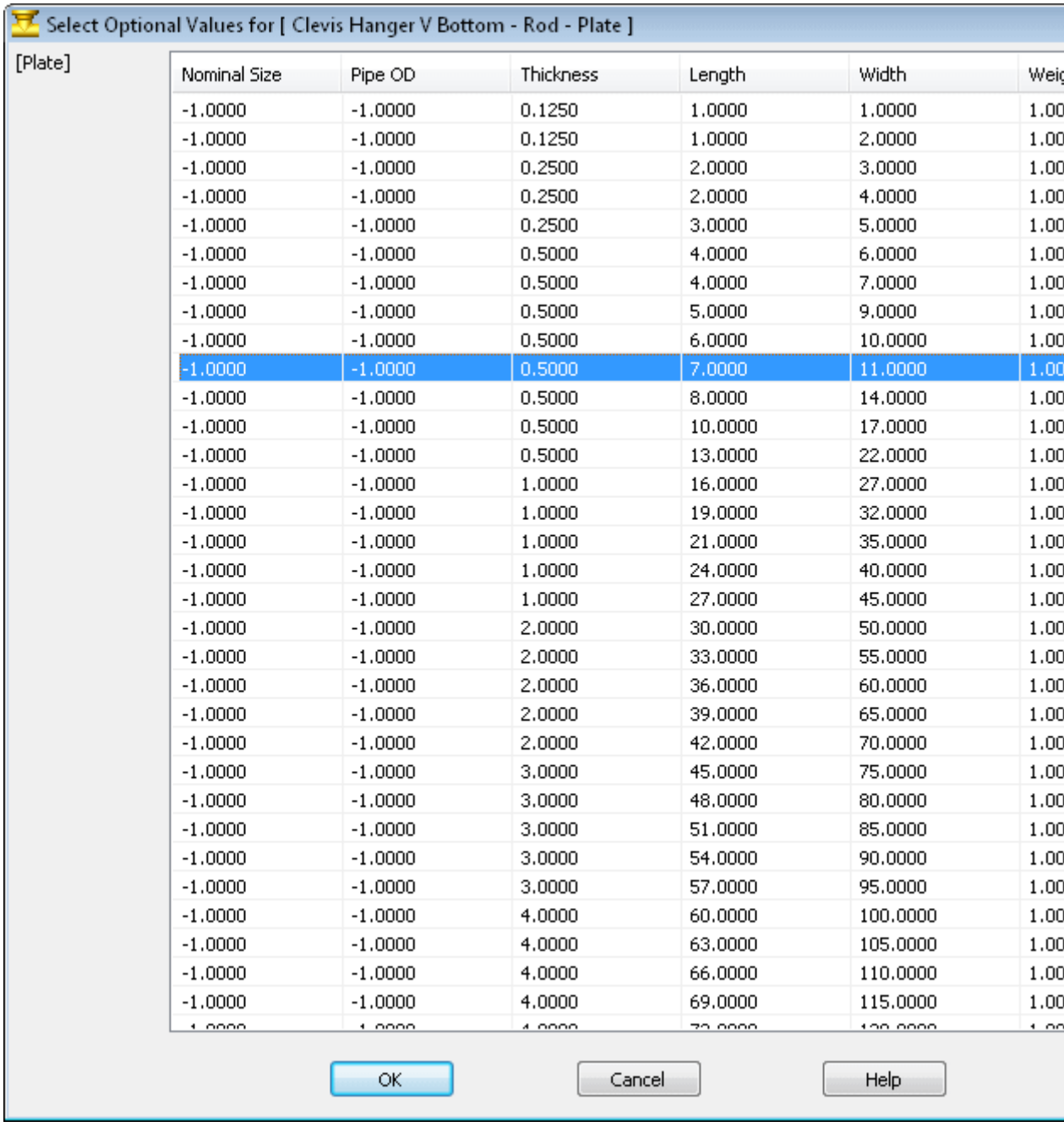
Select Optional Values for [UBolt]

[U Bolt]

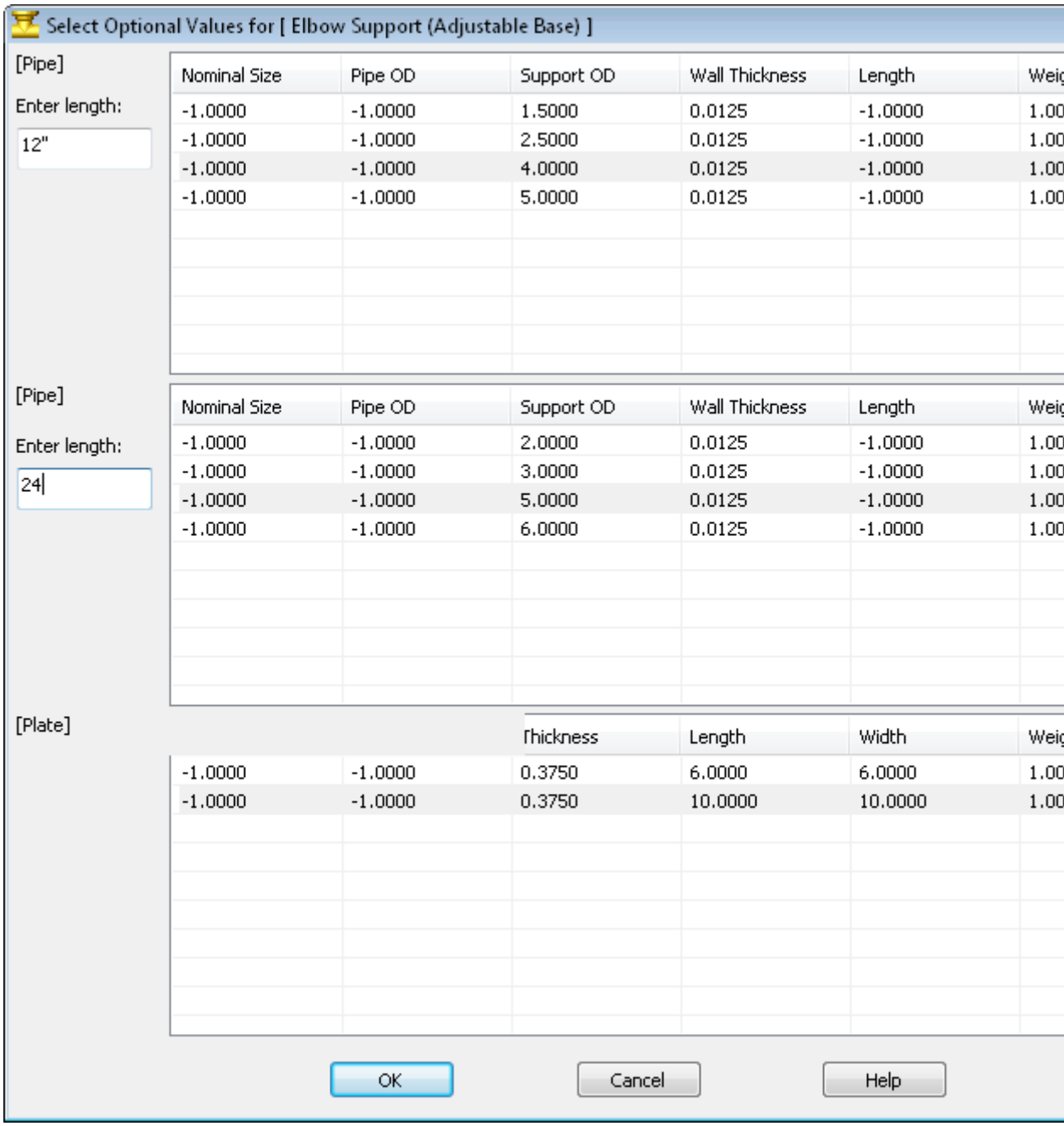
Nominal Size	Pipe OD	Bolt OD	Leg Length	Weight
10.0000	10.7500	1.0000	7.5000	1.0000
10.0000	10.7500	1.5000	8.5000	1.0000
10.0000	10.7500	2.0000	9.5000	1.0000

SHOWOPTIONAL = 2 : The entire content of the file will be shown. This setting is useful for components that are not based on the connecting pipe or elbow (plates, rods, spring hangers).




Select Optional Values for [Clevis Hanger V Bottom - Rod - Plate]
[Plate]
Nominal Size
Pipe OD
Thickness
Length
Width
Weig
-1.0000 -1.0000 0.1250 1.0000 1.0000 1.00
-1.0000 -1.0000 0.1250 1.0000 2.0000 1.00
-1.0000 -1.0000 0.2500 2.0000 3.0000 1.00
-1.0000 -1.0000 0.2500 2.0000 4.0000 1.00
-1.0000 -1.0000 0.2500 3.0000 5.0000 1.00
-1.0000 -1.0000 0.5000 4.0000 6.0000 1.00
-1.0000 -1.0000 0.5000 4.0000 7.0000 1.00
-1.0000 -1.0000 0.5000 5.0000 9.0000 1.00
-1.0000 -1.0000 0.5000 6.0000 10.0000 1.00
-1.0000 -1.0000 0.5000 7.0000 11.0000 1.00
-1.0000 -1.0000 0.5000 8.0000 14.0000 1.00
-1.0000 -1.0000 0.5000 10.0000 17.0000 1.00
-1.0000 -1.0000 0.5000 13.0000 22.0000 1.00
-1.0000 -1.0000 1.0000 16.0000 27.0000 1.00
-1.0000 -1.0000 1.0000 19.0000 32.0000 1.00
-1.0000 -1.0000 1.0000 21.0000 35.0000 1.00
-1.0000 -1.0000 1.0000 24.0000 40.0000 1.00
-1.0000 -1.0000 1.0000 27.0000 45.0000 1.00
-1.0000 -1.0000 2.0000 30.0000 50.0000 1.00
-1.0000 -1.0000 2.0000 33.0000 55.0000 1.00
-1.0000 -1.0000 2.0000 36.0000 60.0000 1.00
-1.0000 -1.0000 2.0000 39.0000 65.0000 1.00
-1.0000 -1.0000 2.0000 42.0000 70.0000 1.00
-1.0000 -1.0000 3.0000 45.0000 75.0000 1.00
-1.0000 -1.0000 3.0000 48.0000 80.0000 1.00
-1.0000 -1.0000 3.0000 51.0000 85.0000 1.00
-1.0000 -1.0000 3.0000 54.0000 90.0000 1.00
-1.0000 -1.0000 3.0000 57.0000 95.0000 1.00
-1.0000 -1.0000 4.0000 60.0000 100.0000 1.00
-1.0000 -1.0000 4.0000 63.0000 105.0000 1.00
-1.0000 -1.0000 4.0000 66.0000 110.0000 1.00
-1.0000 -1.0000 4.0000 69.0000 115.0000 1.00
OK
Cancel
Help

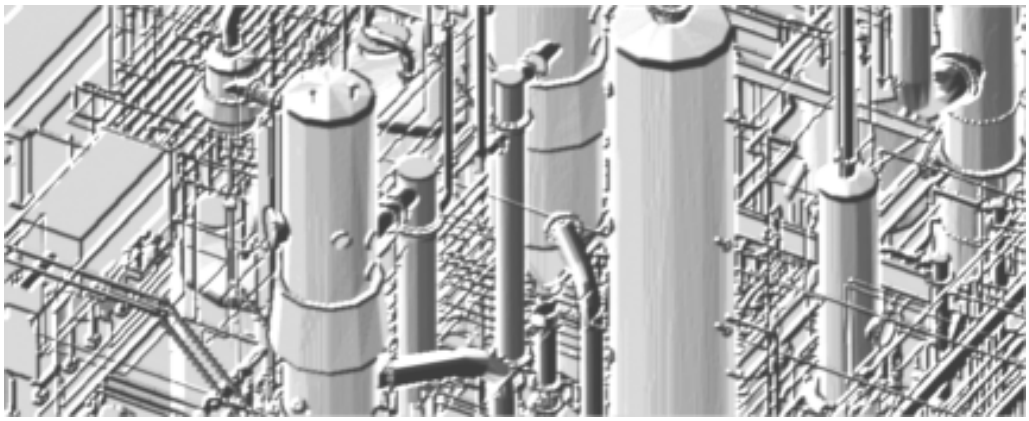
If several components in the assembly have the SHOWOPTIONAL setting of 1 or 2, a dialog will be displayed similar to the one below to select the different components that make up the assembly. The rod, pipe, and spring hanger have an additional option for the length column in the data file. If the length column is set to "-1", use the "Enter length" edit box to enter the length of the component.



If SHOWOPTIONAL = 0 for all components in the assembly, the dialog will be shown to enter the length of the component. The length for the last variable length component will be prompted for during the assembly insertion into the model.

CHAPTER 6

Bill of Material



In This Chapter

Setup 386
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Setup

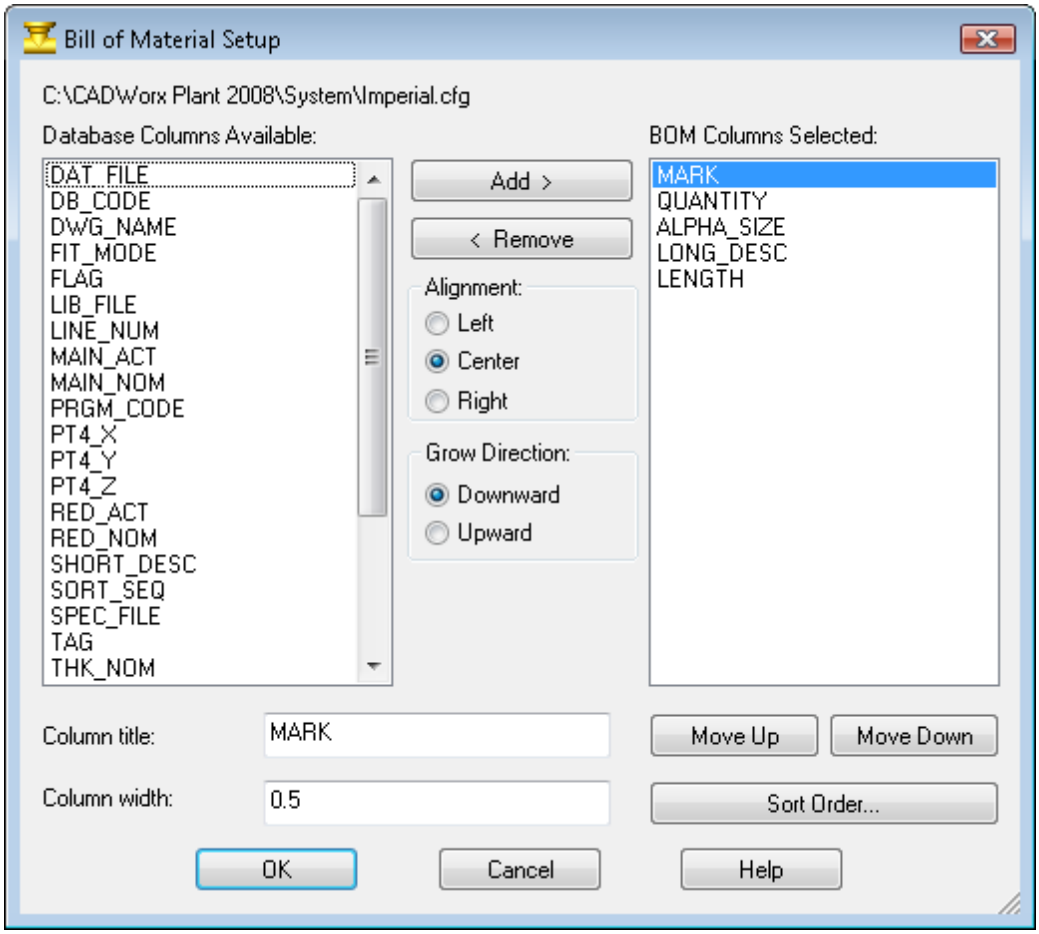
Plant menu: ➤ **Bill of Material** ➤ **Setup**

Command line: BOMSETUP

This dialog provides the user the ability to customize the Bill of Material schedule to any style or layout desired. This utility can specify column widths, provide the schedule direction (up or down), provide user specified headings for columns, allow alignment of columns and most of all allow the selection of over 30 different entries into the schedule. The columns can be arranged in any order. Select (click on the desired columns) an entry on the left side of the dialog and use the Add and Remove button in the middle to add these columns to any Bill of Material schedule.

The template for this information will be stored in the configuration file as listed at the top of the dialog.

Note: Bill of material lengths are based on DIMLFAC, DIMDEC, and DIMLUNIT.



Database Columns Available

The *Database columns available* list box contains a list of columns that can be added to a BOM. The column names represent data that is stored within each component.

There are six special columns: USER1, USER2, USER3, USER4, USER5, UNIT. These columns will be will be blank in the BOM. The user will have to manually enter data for these columns after the BOM is generated.

PT4_X, PT4_Y and PT4 _Z are the world coordinate system location of the approximate center point of the component.

BOM Columns Selected

The *BOM columns selected* list box shows all the columns that have been selected and their order of appearance in the schedule from left (top of list box) to right (bottom of list box).

The Sort Order (see page 388) button should be used for setting the order by which these columns are sorted.

Add & Remove

The *Add* and *Remove* buttons in the middle of the list boxes copy columns from the columns available list to the column selected list box. These buttons allow individual columns to be placed or the user can select all to be added to the right list box. The facility to remove the columns is also available.

The user can also add a column by double-clicking on the Columns Available list. In addition, they can remove a column by double-clicking on the Columns Selected list.

Alignment & Grow Direction

The *Alignment* toggle specifies alignment for a particular highlighted column. Highlight the column and then select its alignment.

The *Grow Direction* toggle allows the schedule to start at the top and work its way downward or start at the bottom and work its way upward.

Move Up/Down

The Move Up and Move Down buttons allow the user to order the columns highlighted in the right list box. Highlight a column in the right list box and then click the move up button (to place it further left) or click on the move down button (to place it further right).

Note: Move Up and Move Down do not set the sort order. Click on Sort Order (see page 388) for setting the sort order.

Column Width and Title

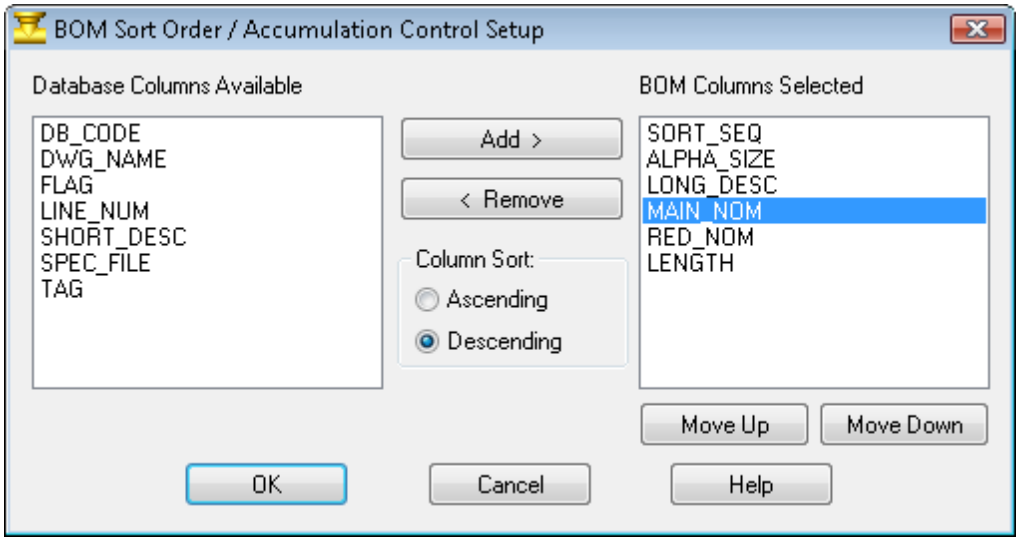
The *Column Title* edit box allows the user to specify a column heading for the highlighted column in the *Column Selected* list box.

The *Column Width* edit box allows the user to specify a column width for the highlighted column in the *Column Selected* list box.

Sort Order

Note: The BOM Sort Order also controls **accumulation** of piping components.

For example: If two valves have different TAG values and you do not want these to accumulate, TAG would have to be added to the BOM Sort Order.



The list box on the right hand side of the dialog lists all the columns that are selected. The user can select one or more columns to sort by. They can also use the Move Up and Move Down buttons to change the order in which the BOM is sorted. Column Sort allows additional sorting on each column. Ascending sorts from small numbers to larger numbers, Ex: 1 to 100 or from A to Z. Descending sorts from large numbers to small numbers, Ex. 100 to 1, or from Z to A.

Note: If you want the BOM to sort from large size piping to small size piping, use Descending for MAIN_NOM and RED_NOM

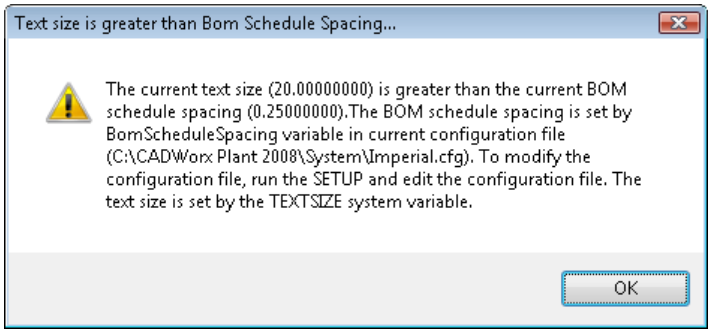
Run

Plant menu: ➤ **Bill of Material** ➤ **Run**

Command line: BOMCUT, BOMTOTAL, **or** BOMSINGLE

This is the main Bill of Material generator. The Total option provides one tag for a total length of pipe in each size. The Cut option provides each piece of pipe with a length and a tag. There are various options that will enhance the Bill of Material generator that will be discussed after each prompt. By selecting Cut or Total from the pulldown, the following prompts will appear. If no components are detected or selected, the generator will halt. The component could have been converted with the Existing option.

The schedule's text size is based on the AutoCAD setvar TEXTSIZE. If this text size is larger than the schedule spacing, the following message will appear and the Bill of Material generation will halt. The schedule spacing is controlled by the BomScheduleSpacing (see page 52) variable in the Configuration file.



Note: A common mistake in AutoCAD is to apply a text height while setting up a text style. If the height is specified in the text style dialog, then there is no other way of controlling it. It will be set to the size in the dialog regardless of what the TEXTSIZE variable is.

CADWorx Plant performs special optimization for automatic tag placement while in paper space only. This optimization will place tags in non-overlapping positions. This optimization is controlled by the BomTagOptimize variable in the current configuration file. For more information regarding this functionality and its controls, please refer to BomTagOptimize (see page 53) variable.

This function is also available as **BOMTOTAL** and **BOMSINGLE**.

Command: **BOMCUT**
Enter tag option [**Counter**/**Manual placement**/**Automatic placement**] <Automatic>:

Counter

This option allows the user to start the tag counter at any positive nonzero integer. Once set, the command resumes requesting the user pick Manual or Automatic placement of tags.

Manual placement

This option inserts tags manually. It prompts the user for the location of the tag and there will be a rubber band line attached to the component extending to the desired tag location. The process will continue until the last component's tag has been placed.

Enter an option [Line number/Select components] <Select components>: *select components or line number*

If the BOM is being generated from paper space, the following prompt would ask whether to turn the snap option on and to change the cursor to an isometric flat view. This works well for placing text, tags, etc. in paper space for isometric models.

Pspace isometric and snap <Yes>: *enter or type* **N** *for no*

This option requires the user to pick a point on the drawing that will be the upper right corner of the users Bill of Material's heading and schedule.

Select upper right corner of BOM heading:

At this point the BOM generator will step through each component selected and prompt for a tag location.

(PIPE, S/40 SMLS, ASTM A-106 GR B) Pick tag location or [Skip]:
(PIPE, S/40 SMLS, ASTM A-106 GR B) Pick tag location or [Skip]:
Etc...

Automatic placement

This option inserts tags automatically. In model space, it places the tag in the direction (tag to component) provided in the next prompt. In paper space, this prompt will not appear if the tag optimization discussed earlier is set on. Please refer to BomTagOptimize (see page 53) for more information.

Enter tag placement angle <45>: *enter angle or enter*

If the user is in paper space, it will switch to the last Mview for component selection. If the components are not in the last Mview, just pick the desired Mview with the arrow indicator that is provided in paper space. Selection can be achieved by using the crossing, window, or individual selection methods.

Select upper right corner of BOM heading: *pick a point*
Enter an option [Line number/Select components] <Select components>:

The user can select components by normal selection methods or select one or more line numbers using the line number option.

Note: Bill of material lengths are based on DIMLFAC, DIMDEC, and DIMLUNIT.

Tag

Toggle

Plant menu: ▶ **Bill of Material** ▶ **Tag** ▶ **Toggle**

Command line: TAGTOGGLE

This option toggles tags on or off. By selecting the toggle option from the pulldown the user can turn the tags off. This provides the Bill of Material schedule without the tags. By selecting the toggle option again, tags are turned back on.

Command: **TAGTOGGLE**
Tags are turned off...

Hit enter or run this command again and the tags are turned back on.

Tags are turned on...

Location

Plant menu: ▶ **Bill of Material** ▶ **Tag** ▶ **Relocate**

Command line: TAGRELOCATE

This option will relocate a tag. It is useful when tags are all placed on top of each other. Optionally, it will allow the connector line (from item mark to component) to be deleted. This is useful when multiple items are required for one location. A good example would be gaskets and bolts associated with one flange.

Note: Tags and the associated leader lines are grouped. This function works well for their manipulation. The user can also use the AutoCAD Group command to un-group them and manually re-arrange them.

Command: **TAGRELOCATE**

Select tag entity to relocate: *pick a tag on the drawing*
Select new location: *pick a new point*
Delete connector line [Yes/No] <No>: *enter* **Y** *(for yes) or enter for No*

Note: If running CADWorx Equipment, run the command EQUIPTAGRELOCATE

Insert

Plant menu: ▶ **Bill of Material** ▶ **Tag** ▶ **Insert**

Command line: TAGINSERT

This option places tags on the drawing without having to run the main Bill of Material generator. The tag will be scaled according to the present text size.

Command: **TAGINSERT**

Enter tag number: *type in any alphanumeric value*
Select component for tagging: *select any component*

Note: If running CADWorx Equipment, run the command EQUIPTAGINSERT

Export

Plant menu: ➤ **Bill of Material** ➤ **Export**

Command line: BOMEXPORT

This utility exports the bill of material to various file formats. This facility will export the bill of material to a file that will be based on the columns that have been chosen in BOMSETUP (see page 386). It will only use the column names. CADWorx Plant can export the bill of material to the following file formats:

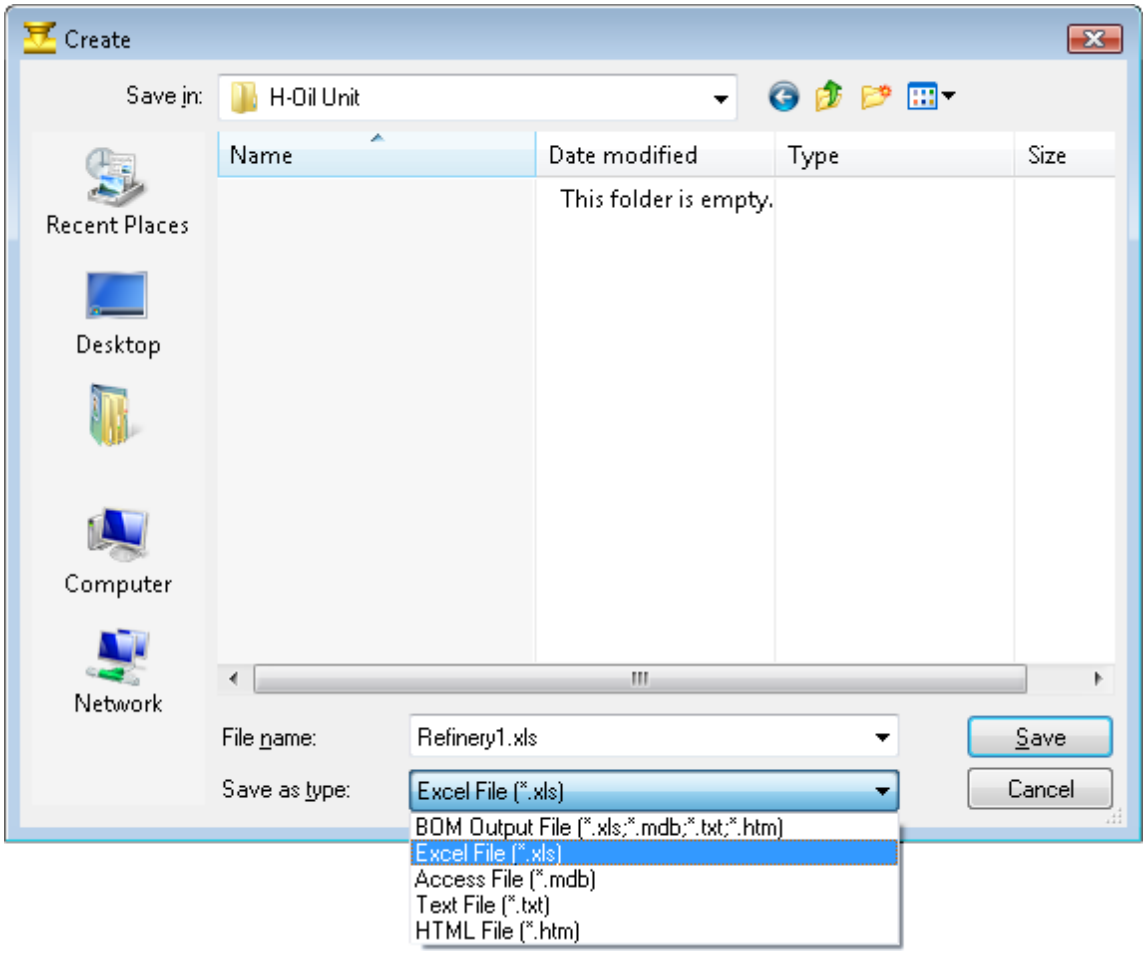
- Access file (*.mdb)
- Excel file (*.xls)
- Text file (*.txt)
- HTML file (*.htm)

Command: **BOMEXPORT**
Enter an option **[Single item/Total length/Cut length]** <Cut>:

These options are the same as the run options in the Bill of Material generator (see page 389).

Enter an option **[Line number/Select components]** <Select components>: *select component or pick line number*

After the components or line numbers have been selected, the program displays the file selection dialog that allows the name to be specified. At the bottom of the dialog, in the *Save as* type list box, notice the different file types. One of these types should be selected and then a file name is needed.



A final prompt indicates that the file was exported.

BOM output file created: E:\CADWorx Pipe\Refinery1.mdb

Note 1: An additional command is available called BOMEXTERNAL. This command allows the user to export a project database to a file based on the columns chosen in the BOMSETUP.

Note 2: When exporting data to Excel, numeric data will be exported as text and will be preceded by a single quote. This single quote will prevent you from totaling the column in Excel. To correct this in Excel, first select the column that needs to be updated. Then, from the Data pulldown menu in Excel, select Text To Columns option. Go through the wizard selecting the "Delimited" radio button and "Tab" as the Delimiter check box. This will cause Excel to change the data from text to numbers. In Imperial/Inch settings, length is exported in feet and inches which cannot be totaled in Excel. To export length in decimal format, change the current AutoCAD Dimension Style settings "Primary Units" format to decimal.

Import

Plant menu: ▶ **Bill of Material** ▶ **Import**

Command line: BOMIMPORT

This utility allows the user to import a database into a bill of material schedule on the present drawing. The schedule will be generated according to the BOMSETUP parameters. It can ONLY import Access files (*.mdb) formats created with the DBFGEN (see page 236) command. This function cannot be used to import data from the files generated with the BOMEXPORT function. BOMEXPORT exports only partial field information, whereas the BOMIMPORT function expects all the fields to be present.

Command: **BOMIMPORT**
Enter an option **[Single item/Total length/Cut length]** <Cut>:

These options are the same as the run options in the Bill of Material generator (see page 389). If the database source is not set, it will request the user to select a database.

Current BOM source database is not set...
Select database **[Yes/No]** <No>: *select yes and pick database*

At this point, it will provide the same prompts as the BOM generator.

Delete

Plant menu: ▶ **Bill of Material** ▶ **Delete**

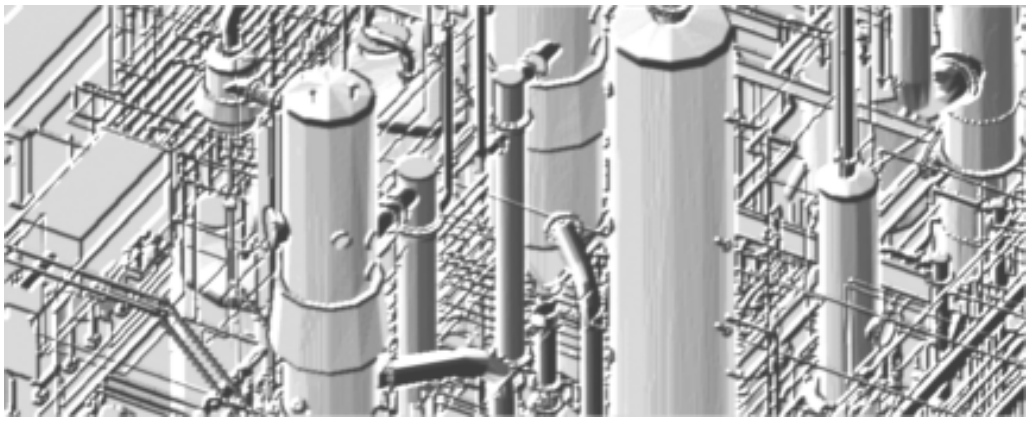
Command line: BOMDELETE

This option deletes all entities that are on the layer designated for Bill of Materials in the current configuration file. This option cannot be used for deleting partial tags or for deleting the schedule only.

Command: **BOMDELETE**
Deleting all Bill of Material tags and schedule...

CHAPTER 7

Graphics



In This Chapter

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Introduction

These dialogs provide the following graphics for use with CADWorx Plant. The graphic drawings are located in the <INSTALL-DIR>\SUPPORT directory and can be user modified. If they are modified, they must incorporate the same amount of attributes presently in the drawing.

A user can also add more graphic to these dialogs or even make their own. The function that produces this dialog is (viewslide [string]). The string argument needs to be in quotes (i.e.: “Section”). This functions then examines the <INSTALL-DIR>\SUPPORT\PIPE_ALT.DAT (or Steel_ALT.dat for CADWorx Steel) file to locate a group the same name as the string.

In this file, columns must start at locations 1, 40, and 75. If they are not at these positions, the function that draws them will fail. Also, there must not be any blank lines located within a section. There must be a blank line at the end and the beginning. Shown below is the group that creates the first “Sections” dialog.

COL1(1) Drawing name used in the dialog for preview image.

COL2(40) Name to be displayed in dialog for graphic name.

COL3(75) Function used to draw the graphic in the drawing with arguments.

```
[Sections]
cpm2.slb(s1s)     Head                    graph1 "s1" 1
cpm2.slb(s2s)     Head w/ Attrib          graph1 "s2" 2
cpm2.slb(s3s)     Head w/ 2 Attrib        graph1 "s3" 3
cpm2.slb(s4s)     End w/ Attrib           graph1 "s4" 4
```

Suppose a new group was required (Widgets). The preview drawing needed in the dialog was named wid1.dwg and the dialog prompt need to be “Draw widget 1”. The program that draws widget 1 would be (widdraw). The drawing needed is w1.dwg. There is one argument, scale size 32.

The PIPE_ALT.DAT (or Steel_ALT.dat) file would need a new group call “[Widgets]” as shown below.

```
[Widgets]
wid1.dwg          Draw widget 1           widdraw "w1" 32
```

The command to execute (widdraw “w1” 32) is shown next.

```
Command: (viewslide "Widgets")
```

Please make a backup copy of PIPE_ALT.DAT (or Steel_ALT.dat) prior to making any modifications to this file. The dialog can be accessed from the pulldown under Plant -> Graphic (or Steel -> Graphics).

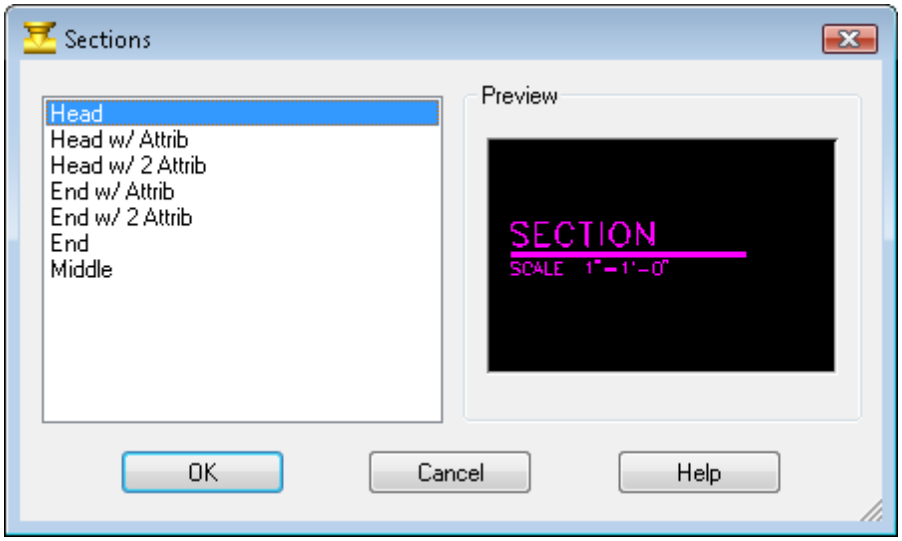
Note: The scale of the graphics inserted into the drawing from this facility is based upon the value of the AutoCAD setvar DIMSCALE. DIMSCALE can be controlled through the Setup (see page 18) dialog with the *Scale* button or from the command line. If the DIMSCALE is set to 0, as it is set to in paperspace, the program will insert the graphics at a scale of 1.

Section

Plant menu: ➤ **Graphics** ➤ **Sections**

Command line: SECTIONS

Please see the note regarding scaling on the introduction page of this chapter.



For the graphic drawing…

S1	prompt 1
S2	prompt 2
S3	prompt 3
S4	prompt 4
S5	prompt 5
S6	prompt 6
S7	prompt 6

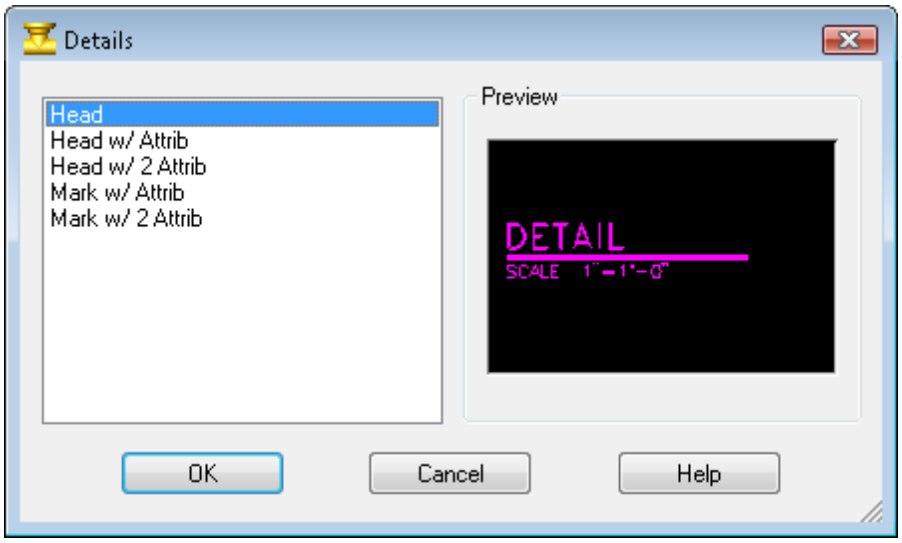
…would be displayed.

Detail

Plant menu: ▶ **Graphics** ▶ **Details**

Command line: DETAILS

Please see the note regarding scaling on the introduction page of this chapter.



For the graphic drawing…

D1	prompt 1
D2	prompt 2
D3	prompt 3
D4	prompt 7
D5	prompt 8

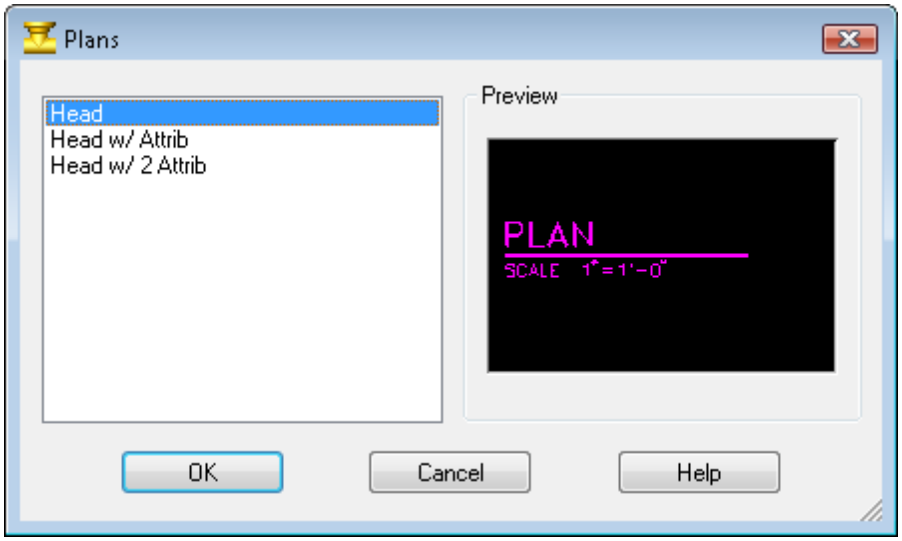
… would be displayed.

Plan

Plant menu: ➤ **Graphics** ➤ **Plans**

Command line: PLANS

Please see the note regarding scaling on the introduction page of this chapter.



For the graphic drawing…

P1 prompt 1

P2 prompt 2

P3 prompt 3

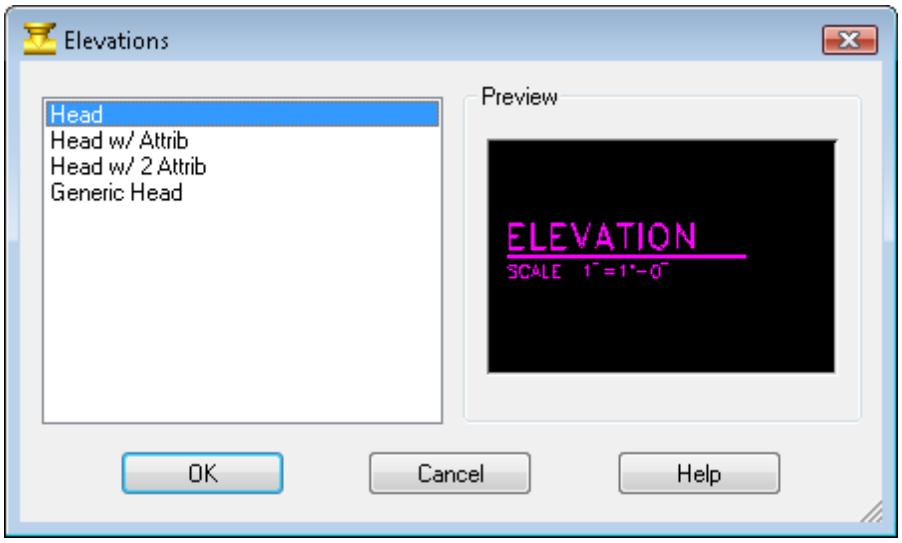
…would be displayed.

Elevation

Plant menu: ➤ **Graphics** ➤ **Elevations**

Command line: ELEVATIONS

Please see the note regarding scaling on the introduction page of this chapter.



For the graphic drawing…

E1	prompt 1
E2	prompt 2
E3	prompt 3
L1	prompt 9

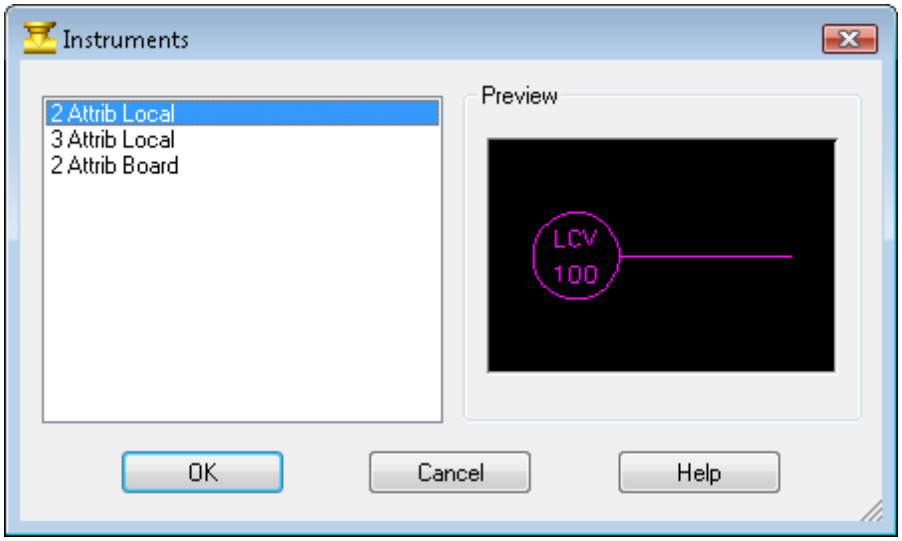
…would be displayed.

Instrument

Plant menu: ➤ **Graphics** ➤ **Instruments**

Command line: INSTRUMENTS

Please see the note regarding scaling on the introduction page of this chapter.



For the graphic drawing…

F1 prompt 10

F2 prompt 11

F3 prompt 12

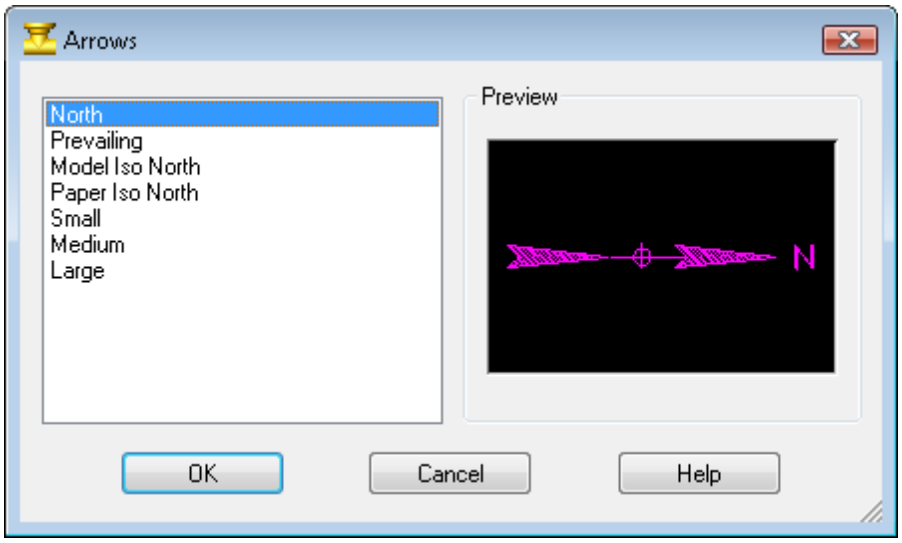
… would be displayed.

Arrow

Plant menu: ▶ **Graphics** ▶ **Arrows**

Command line: ARROWS

Please see the note regarding scaling on the introduction page of this chapter.



For the graphic drawing…

N1	prompt 14
N2	prompt 12
N3	prompt 12
N4	prompt 14
A1	prompt 12
A2	prompt 12
A3	prompt 12

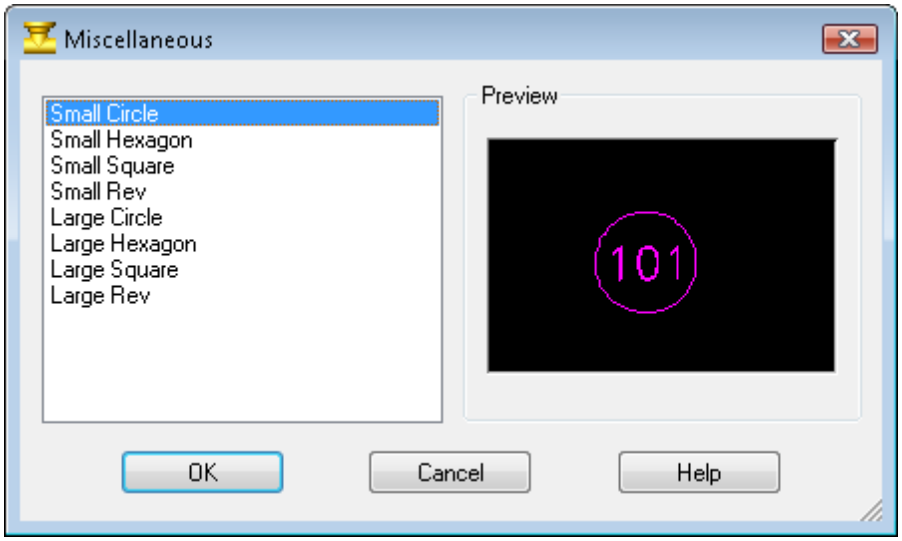
… would be displayed.

Misc

Plant menu: ➤ **Graphics** ➤ **Misc**

Command line: MISC

Please see the note regarding scaling on the introduction page of this chapter.



For the graphic drawing…

M1	prompt 15
M2	prompt 15
M3	prompt 15
M4	prompt 15
M5	prompt 15
M6	prompt 15
M7	prompt 15
M8	prompt 15

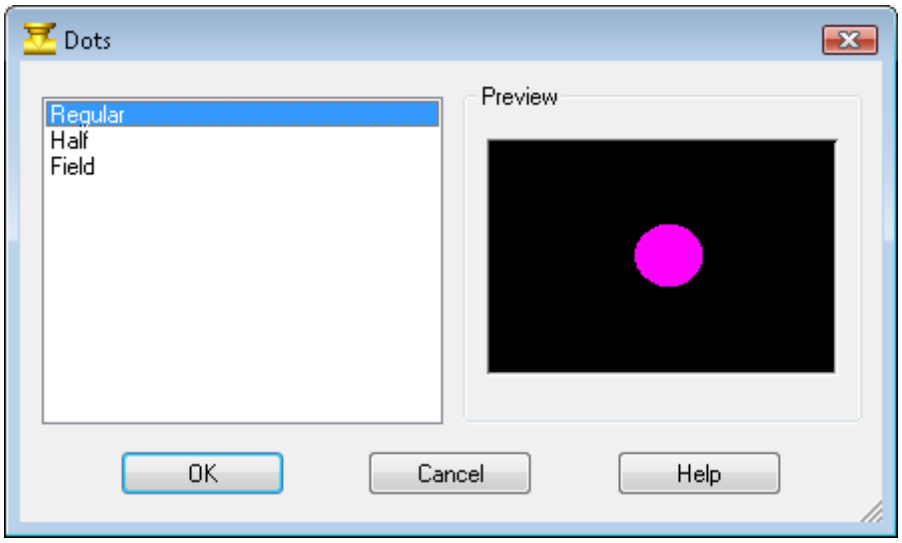
… would be displayed.

Dots

Plant menu: ➤ **Graphics** ➤ **Dots**

Command line: DOTS

Please see the note regarding scaling on the introduction page of this chapter.



For the graphic drawing…

W1 prompt 15

W2 prompt 15

W3 prompt 15

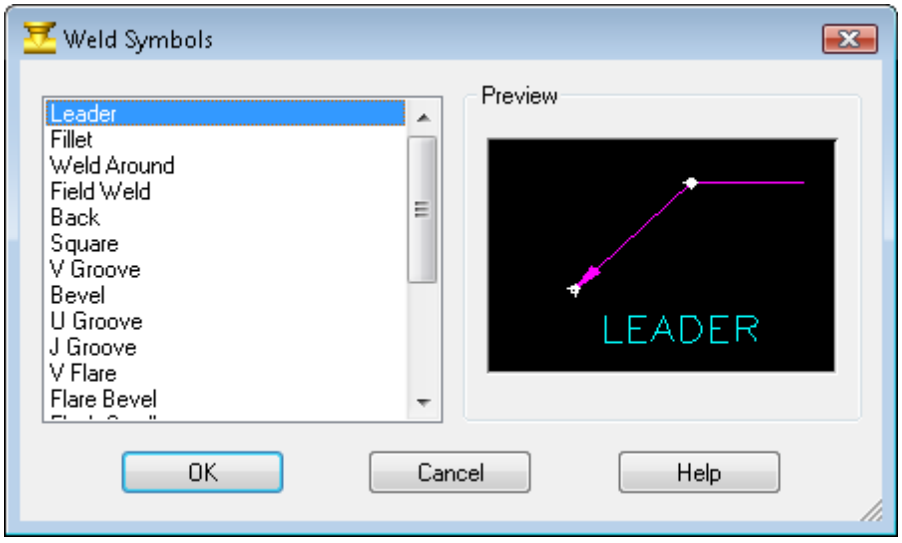
…would be displayed.

Weld

Plant menu: ➤ **Graphics** ➤ **Weld**

Command line: WELDS

Please see the note regarding scaling on the introduction page of this chapter.



For the graphic drawing…

Leader	prompt 16
Fillet	prompt 17
Around	prompt 18
Field	prompt 17
Back	prompt 17
Square	prompt 17
V Groove	prompt 17
Bevel	prompt 17
U Groove	prompt 17
J Groove	prompt 17
V Flare	prompt 17
Flare Bevel	prompt 17
Plug	prompt 17
Flush Small	prompt 17
Flush Large	prompt 17
Convex Small	prompt 17
Convex Medium	prompt 17
Convex Large	prompt 17
Backing	prompt 19
Spacer	prompt 19

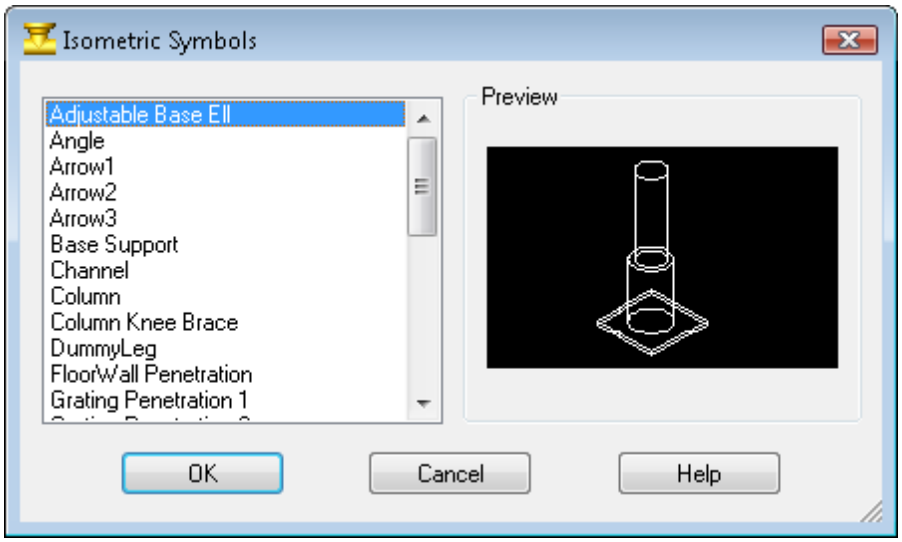
…would be displayed.

Iso

Plant menu: ➤ **Graphics** ➤ **ISO**

Command line: ISOS

Please disregard the note on scaling in the introduction page of this chapter. These graphics will be inserted at a scale of 1.



See prompt 20 for all items in this category.

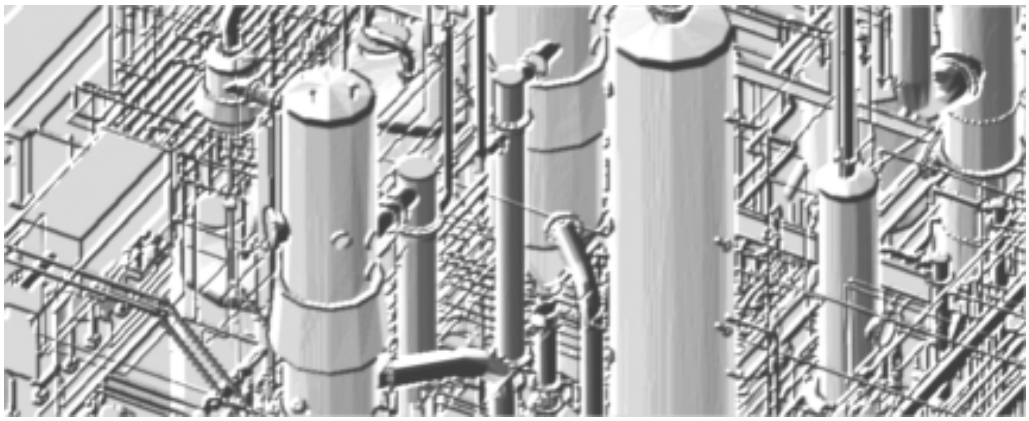
Prompts

Prompt Number	Prompt
Prompt 1	Pick left justified insertion point: *pick point* Rotation angle: *pick direction or enter angle* Enter scale: *enter string*
Prompt 2	Pick left justified insertion point: *pick point* Rotation angle: *pick direction or enter angle* Enter mark: *enter string* Enter scale: *enter string*
Prompt 3	Pick left justified insertion point: *pick point* Rotation angle: *pick direction or enter angle* Enter mark: *enter string (ex: A)* Enter drawing: *enter string (ex: 106)* Enter scale: *enter string (ex: 3/8"=1'-0")*
Prompt 4	Pick insertion point: *pick point* Section direction: *pick direction* Projection direction: *pick direction* Enter mark: *enter string (ex: A)*

Prompt 5	Pick insertion point: *pick point* Section direction: *pick direction* Projection direction: *pick direction* Enter mark: *enter string (ex: A)* Enter drawing: *enter string (ex: 106)*
Prompt 6	Pick insertion point: *pick point* Section direction: *pick direction* Projection or other direction: *pick direction*
Prompt 7	Pick object to mark: *pick object* Pick mark location: *pick point* Enter mark: *enter string (ex: A)*
Prompt 8	Pick object to mark: *pick object* Pick mark location: *pick point* Enter mark: *enter string (ex: A)* Enter drawing: *enter string (ex: 106)*
Prompt 9	Pick left end point: *pick point* Pick right end point: *pick point* Enter title: *enter string (ex: ELEVATION NORTH)* Enter scale: *enter string (ex: 3/8"=1'-0")*
Prompt 10	Pick instrument to label: *pick point* Pick mark location: *pick point* Enter first level label: *enter string (ex: SDV)* Enter second level label: *enter string (ex: 1)*
Prompt 11	Pick instrument to label: *pick point* Pick mark location: *pick point* Enter first level label: *enter string (ex: PI)* Enter second level label: *enter string (ex: 106)* Enter third level label: *enter string (ex: B)*
Prompt 12	Pick center insertion point: *pick point* Rotation angle: *pick direction or enter angle*
Prompt 13	Pick center insertion point: *pick point*
Prompt 14	Pick center insertion point: *pick point* Pick direction: *pick direction*
Prompt 15	Pick insertion point: *pick point (not center insertion, outside edge)* Pick direction: *pick direction* Enter mark: *enter string (ex: 101)Weld*
Prompt 16	Pick leader start point: *pick point* Pick intermediate point: *pick point* Pick direction for tail: *pick direction*
Prompt 17	Pick insertion point: *pick point* Pick rotation: *pick direction*
Prompt 18	Pick insertion point:
Prompt 19	Pick insertion point on line: *pick point* Pick rotation: *pick direction*
Prompt 20	Pick location: *pick point* Pick rotation or [Up/Down/Roll/mirrorX/mirrorY/mirrorZ]: *chose an option or pick the rotation*

CHAPTER 8

Equipment



In This Chapter

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Circle

Plant menu: ▶ **Equipment** ▶ **Circle**

Command line: VESSELOD

This routine is used for drawing circles with centerlines. The most common use of this routine would be providing a vessel in plan view. The circle will be placed on the “EQUIP” layer and the centerlines will be on the “CL” layer. In the 2D Mode the circle will have 0" thickness. In the 3D Mode the circle will provide for a thickness. In 3D mode the object will start at the present UCS elevation and extend upward or in the positive Z direction.

In the 2D Mode, the following prompts will appear.

Command: **VESSELOD**
Pick point at center: *pick point*
Enter radius of vessel or pick point: *enter real number or drag and point for radius*

In the 3D Mode, an additional prompt will appear.

Enter thickness: *enter the desired thickness or elevation (ex. 5')*

Rectangle

Plant menu: ➤ **Equipment** ➤ **Rectangle**

Command line: RECT

This routine draws rectangles that can be used to represent different equipment. The rectangle will be placed on the “EQUIP” layer. In the 2D Mode the rectangle will have 0" thickness. In the 3D Mode, the rectangle will provide for a thickness. In 3D mode, the object will start at the present UCS elevation and extend upward or in the positive Z direction.

In the 2D Mode, the following prompts will appear.

Command: **RECT**
Pick lower left corner: *pick point or enter desired coordinates*
Pick upper right corner: *pick point or enter @x,y*

In the 3D Mode, an additional prompt will appear.

Enter thickness: *enter the desired thickness or elevation (ex. 5')*

Vessel

Plant menu: ▶ **Equipment** ▶ **Vessel**

Command line: VESSEL

Please refer to Heads later in this chapter, for figures explaining the different head prompts for the vessel routine.

Command: **VESSEL**
Enter head type [**Flat/Dished/Hemispherical/Ellipical**] <Elliptical>:

Flat

This option allows for a vessel with flat heads. Distances can be entered from the keyboard or by picking two points.

Enter O.D. of vessel: *enter real number (ex: 48.5, 4', 0-1/2")*
Enter vessel seam-seam: *enter real number*
Enter head straight flange distance <0">: *enter real number*

Dished

This option allows for a vessel with Flanged and Dished heads. Distances can be entered from the keyboard or by picking two points. Please have the head information (ex: Luken's head catalog) handy prior to executing this routine. Guessing at the prompts will provide strange results.

Enter head straight flange distance <0">: *enter real number*
Enter knuckle radius: *enter real number*
Enter dished radius: *enter real number*
Enter inside depth of dish (not including straight flange): *enter real number*

Hemispherical

This option allows for a vessel with Hemispherical heads. Distances can be entered from the keyboard or by picking two points.

Enter head straight flange distance <0">: *enter real number or enter for 0"*

Ellipical

This option allows for a vessel with 2:1 Elliptical heads. Distances can be entered from the keyboard or by picking two points. Note, for vessel heads to conform with the exact ASME 2:1 ellipsoidal formula, the user must provide the I.D. and offset the vessel outline to obtain the O.D. Failure to do so will not provide the exact formula for vessel head dimensioning.

Enter head straight flange distance <0">: *enter real number*

The following prompts appear after the head type options.

Enter O.D. of vessel: *enter real number (ex: 48.5, 4', 0-1/2")*
Enter O.D. of vessel: *pick a point* Second point: *pick other point*

After the preceding prompts, the user will be prompted for the location. They will have two choices of placement.

[**Middle point**] Pick head center at seam:

Middle point

This option allows the user to locate the vessel from the center. They will be prompted for the center point, then the vessels longitudinal direction.

Pick center point of vessel: *pick point*
Pick vessel direction: *pick direction*

Pick head center at seam

This option allows the user to locate the vessel from the seam at center. This location is not at the head's tangent line, but at the seam. If the user entered 0" for the straight flange, this location would be at the head's tangent line.

Pick vessel direction: *pick direction*

Tank

Plant menu: ▶ **Equipment** ▶ **Tank**

Command line: TANK

This routine draws tanks in 2D or 3D Mode. Options provide for three different roofs and a variety of shapes. The bottom or roof plate lap prompts show the overlap outside the tank. For example; entering 48" for O.D. and 2" for bottom plate lap will give the user a bottom plate 52" in diameter. If a flat floating roof is desired, the user can enter 50' for O.D. and -3" for roof plate lap. This gives the user a floating roof of 49'-6" for a 50' diameter tank. This roof can be moved in a negative Z direction to indicate a floating roof.

In 3D Mode, the following prompts will appear. Please refer to the 2D Mode section for prompt descriptions.

Command: **TANK**
Enter roof option [**Flat/Conical/Open**] <Open>:

Flat

This option provides a flat roof.

Enter tank OD: *enter real number*
Enter height of tank: *enter real number*
Enter bottom plate underlap: *enter real number (+ or -)*
Enter roof plate overlap: *enter real number (+ or -)*
Pick center point of bottom: *pick point*

Conical

This option provides a conical roof.

Enter O.D. of tank: *enter real number*
Enter height of tank: *enter real number*
Enter bottom plate underlap: *enter real number (+ or -)*
Enter roof plate overlap: *enter real number (+ or -)*

Enter an option [**Slope/Height**] <Height>:

Slope

Enter degrees of slope: *enter real number*
Pick center point of bottom: *pick point*

Height

Enter height from side: *enter real number*
Pick center point of bottom: *pick point*

Open

This option provides an open roof.

Enter O.D. of tank: *enter real number*
Enter height of tank: *enter real number*
Enter bottom plate underlap: *enter real number (+ or -)*
Pick center point of bottom: *pick point*

For 2D Mode, the following prompts will appear.

Command: **TANK**

Enter an option [**Plan**/**Elevation**] <Elevation>:

Plan

This option draws a tank in plan view showing the required overlap. It will show 2 circles with centerlines.

Enter O.D. of tank: *enter real number*
Enter bottom or roof plate lap: *enter real number (+ or -)*
Pick center point of bottom: *pick point*

Elevation

This option draws a tank in an elevation view showing any of the 3 following roof options.

Enter roof option [**Flat**/**Conical**/**Open**] <Open>:

Flat

This option provides a flat roof.

Enter tank OD: *enter real number*
Enter height of tank: *enter real number*
Enter bottom plate underlap: *enter real number (+ or -)*
Enter roof plate overlap: *enter real number (+ or -)*
Pick center point of bottom: *pick point*
Pick direction to top: *pick direction*

Conical

This option provides a conical roof.

Enter O.D. of tank: *enter real number*
Enter height of tank: *enter real number*
Enter bottom plate underlap: *enter real number (+ or -)*
Enter roof plate overlap: *enter real number (+ or -)*

Enter an option [**Slope**/**Height**] <Height>:

Slope

Enter degrees of slope: *enter real number*
Pick center point of bottom: *pick point*
Pick direction to top: *pick direction*

Height

Enter height from side: *enter real number*
Pick center point of bottom: *pick point*
Pick direction to top: *pick direction*

Open

This option provides an open roof.

Enter O.D. of tank: *enter real number*
Enter height of tank: *enter real number*
Enter bottom plate underlap: *enter real number (+ or -)*
Pick center point of bottom: *pick point*
Pick direction to top: *pick direction*

Head

Plant menu: ▶ **Equipment** ▶ **Head**

Command line: HEAD

This routine is used for drawing heads. The user will be prompted for the type of head required. This routine will provide either 2D or 3D Mode heads. Setting the desired mode will be required first. For example; to draw a 2D head, the user will have to be in the 2D Mode. When selecting the Dished option, as with the vessel routine, the user needs to have the exact dimensions for the Dished Head or the user might get strange results.

For 2D and 3D Mode, the following prompts will appear.

Command: **HEAD**
Enter head type [**Flat**/**Dished**/**Hemispherical**/**Ellipical**] <Elliptical>:

Flat

Pick point of head @ tangent, or seam, and centerline: *pick point*
Pick head @ side: *pick point*

Dished

[Defaults] Enter knuckle radius: *enter real number or enter*

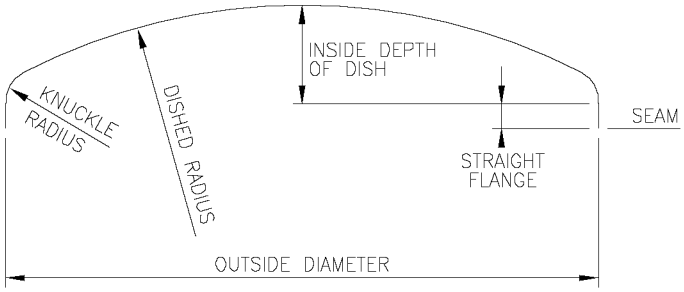
Defaults
Knuckle = 0.06 * OD; Dished radius = OD...

Enter knuckle radius

Enter a knuckle radius. Make sure correct values are entered.

Enter dished radius: *enter real number*
Enter inside depth of dish (not including straight flange): *enter real number*

Pick point of head @ tangent, or seam, and centerline: *pick point*
Pick head @ side: *pick point*
Pick direction: *pick direction*
Enter head straight flange distance <0>:

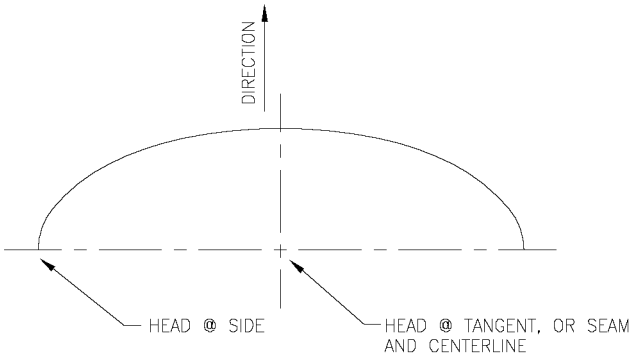


Hemispherical

Pick point of head @ tangent, or seam, and centerline: *pick point*
Pick head @ side: *pick point*
Pick direction: *pick direction*
Enter head straight flange distance <0>:

Elliptical

Pick point of head @ tangent, or seam, and centerline: *pick point*
Pick head @ side: *pick point*
Pick direction: *pick direction*
Enter head straight flange distance <0>:



Pump

Plant menu: ▶ **Equipment** ▶ **Pump**

Command line: PUMP

Pumps can be drawn in 2D or 3D mode. Pumps and other equipment, such as, vessels and tanks, are non-intelligent. This means that they cannot be mode converted to another mode, such as, 2D to 3D or vice-versa. The pump routine will get its' flange information from the flange data files in the <INSTALL-DIR>\SYSTEM directory; FLG.150, FLG.300, FLG.400, FLG.600, and FLG.900. If the user requires a flange that is not one of these ratings, they will have to construct it. The routine will also prompt the user for an electrical motor size. This file is in the same directory and is labeled EMOTOR.TBL. We recommend using the exact pump dimensions to avoid undesirable results.

The prompts for this routine in the 3D Mode will remain the same, but in the 2D Mode they will change depending on the view selected. 2D will be explained after the 3D prompts. Abbreviations are; b.o.b. (bottom of base); f.o.f. (face of flane); and O.D. (outside diameter).

For 3D Mode, the following prompts will appear.

Command: **PUMP**
Enter nominal suction size: *enter a real (i.e. 6")*
Enter nominal discharge size: *enter a real (i.e. 4")*
Enter flange rating file (i.e. 150, 300m, 300mm or NP100) <150>: *return*
Electric motor frame number or enter for list: *enter a alpha-numeric (i.e. 364T)*
Discharge direction [**Horizontal**/**Vertical**] <Vertical>: *return*

Horizontal

This option places the discharge outlet in the horizontal position.

Discharge position [**Bottom**/**Center**/**Top**] <Top>:

Bottom

Choose this option for the discharge outlet to be on the bottom of the impeller.

Center

Choose this option for the discharge outlet to be centered on the impeller.

Top

Choose this option for the discharge outlet to be on the top of the impeller.

These options allow the outlet to be placed on either side.

Discharge side [**Left**/**Right**] <Right>:

Left

Facing the pump suction, this option places the discharge on the left side, in a horizontal position.

Right

Facing the pump suction, this option places the discharge on the right side, in a horizontal position.

Vertical

This option places the discharge outlet in the horizontal position. It will only allow the placement of the outlet in the upper position.

Discharge position [**Right**/**Center**/**Left**] <Left>:

Right

Facing the pump suction, this option places the discharge on the right side, in a vertical position.

Center

Facing the pump suction, this option places the discharge on the centerline, in a vertical position.

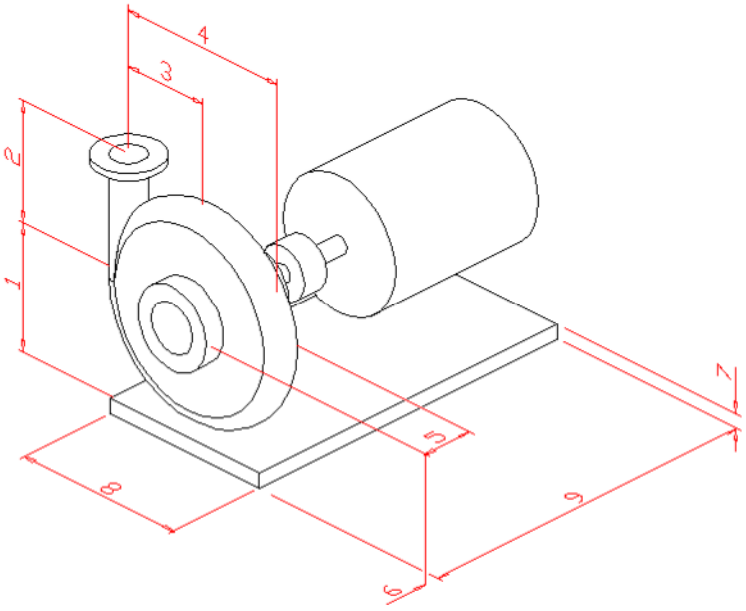
Left

Facing the pump suction, this option places the discharge on the left side, in a vertical position.

<1> Enter distance, b.o.b. to centerline suction: *enter a real (i.e. 16)*
<2> Enter distance, centerline suction to f.o.f. discharge: *enter a real (i.e. 16)*
<3> Enter distance, centerline suction to centerline discharge: *enter a real (i.e. 12)*
<5> Enter distance, f.o.f. suction to centerline discharge: *enter a real (i.e. 5)*
<6> Enter distance, suction f.o.f. to front edge of base (+ or -): *enter a real (i.e. 1)*
<7> Enter base thickness: *enter a real (i.e. 2)*
<8> Enter base width: *enter a real (i.e. 24)*
<9> Enter base length: *enter a real (i.e. 48)*

After the numerical prompts the final location and direction prompt will be required.

Pick front end of base start point: *pick point*
Pick direction to back end of pump: *pick direction*


4
3
2
1
7
5
8
9
6

For 2D Mode, the following prompts will appear.

Command: **PUMP**

An alert will be presented asking whether to view a slide showing prompts. Select Yes or No.

Enter view [**Plan**/**Front**/**Elevation**] <Elevation>:

Plan

This option draws a 2D pump in a plan view

Enter nominal suction size: *enter a real (i.e. 6")*
Enter nominal discharge size: *enter a real (i.e. 4")*
Enter flange rating file (i.e. 150, 300m, 300mm or NP100) <150>: *return*
Electric motor frame number or enter for list: *enter a alpha-numeric (i.e. 364T)*
Discharge direction [**Horizontal**/**Vertical**] <Vertical>:

Horizontal

This option allows for a horizontal discharge. Facing the pump suction; it will ask the user for the left or right discharge direction.

Discharge side [**Left**/**Right**] <Right>:

Left

Facing the pump suction, this option places the discharge on the left side, in a horizontal position.

Right

Facing the pump suction, this option places the discharge on the right side, in a horizontal position.

Vertical

This option allows for a vertical discharge. Facing the pump suction; it will ask the user for the right, center, or left discharge position.

Discharge position [**Right**/**Center**/**Left**] <Left>:

Right

Facing the pump suction, this option places the discharge on the right side, in a vertical position.

Center

Facing the pump suction, this option places the discharge on the centerline, in a vertical position.

Left

Facing the pump suction, this option places the discharge on the left side, in a vertical position.

Front

This option draws a 2D pump from the front view.

Enter nominal suction size: *enter a real (i.e. 6")*
Enter nominal discharge size: *enter a real (i.e. 4")*
Enter flange rating file (i.e. 150, 300m, 300mm or NP100) <150>: *return*
Discharge direction [**Horizontal**/**Vertical**] <Vertical>:

Horizontal

See the description in the Plan view option.

Discharge side [**Left**/**Right**] <Right>:

Left

See description in the Plan view option.

Right

See description above in the Plan view option.

These options allow placement at the bottom, center, and top.

Discharge position [**Bottom**/**Center**/**Top**] <Top>:

Since this is an elevation view and a horizontal discharge direction, the user will be prompted for the discharge location; bottom, center or top. The user will not be prompted for this location in the plan view because it would not make any difference which way the view was drawn.

Bottom

This option places the discharge on the bottom, tangent with the impeller housing, in a horizontal position.

Center

This option places the discharge on the centerline, in a horizontal position.

Top

This option places the discharge on the top, tangent with the impeller housing, in a horizontal position.

Vertical

This option allows placement of the outlet in vertical position.

Discharge position [**Right**/**Center**/**Left**] <Left>:

Right

See the description in the Plan view option.

Center

See the description in the Plan view option.

Left

See the description in the Plan view option.

Elevation

This option draws a 2D pump in an elevation view.

Enter nominal suction size: *enter a real (i.e. 6")*
Enter nominal discharge size: *enter a real (i.e. 4")*
Enter flange rating file (i.e. 150, 300m, 300mm or NP100) <150>: *return*
Electric motor frame number or enter for list: *enter a alpha-numeric (i.e. 364T)*
Discharge direction [**Horizontal**/**Vertical**] <Vertical>:

Horizontal

See the description above in the Plan view option.

Discharge position [**Bottom**/**Center**/**Top**] <Top>:

Bottom

See the description above in the Plan view option.

Center

See the description above in the Plan view option.

Top

See the description above in the Plan view option.

Vertical

No additional prompts

After the previous prompts, the user will be prompted for various dimensional information. Selected views would prompt for some or all of the following numeric prompts. The following prompts are numerically correlated with the view slide to help the user determine the required input. Enter a real number, for example: 16-1/4, 16-1/4", 16.25, or 16.25"

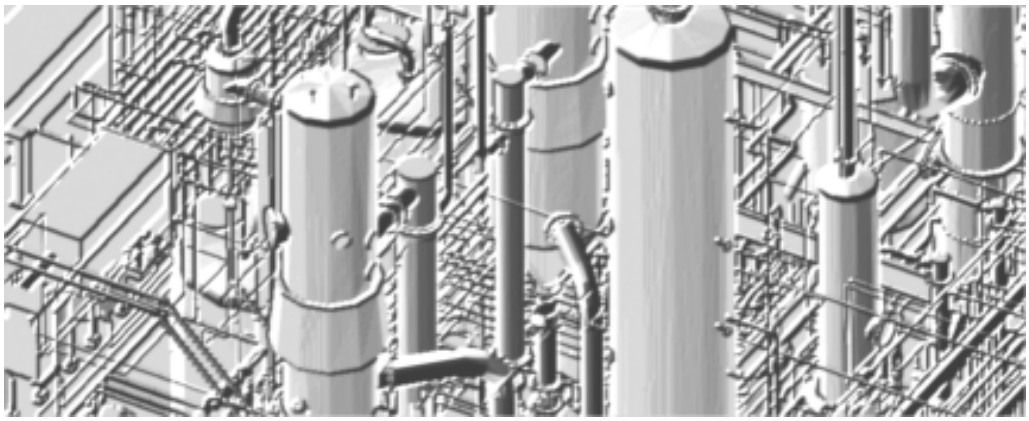
<1> Enter distance, b.o.b. to centerline suction: *enter a real (i.e. 16)*
<2> Enter distance, centerline suction to f.o.f. discharge: *enter a real (i.e. 16)*
<3> Enter distance, centerline suction to centerline discharge: *enter a real (i.e. 12)*
<5> Enter distance, f.o.f. suction to centerline discharge: *enter a real (i.e. 5)*
<6> Enter distance, suction f.o.f. to front edge of base (+ or -): *enter a real (i.e. 1)*
<7> Enter base thickness: *enter a real (i.e. 2)*
<8> Enter base width: *enter a real (i.e. 24)*
<9> Enter base length: *enter a real (i.e. 48)*

After the numerical prompts the final location and direction prompt will be required.

Pick front end of base start point: *pick point*
Pick direction to back end of pump: *pick direction*

CHAPTER 9

Steel



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Shapes

Plant menu: ➤ **Steel** ➤ **Shapes**

Command line: STEEL

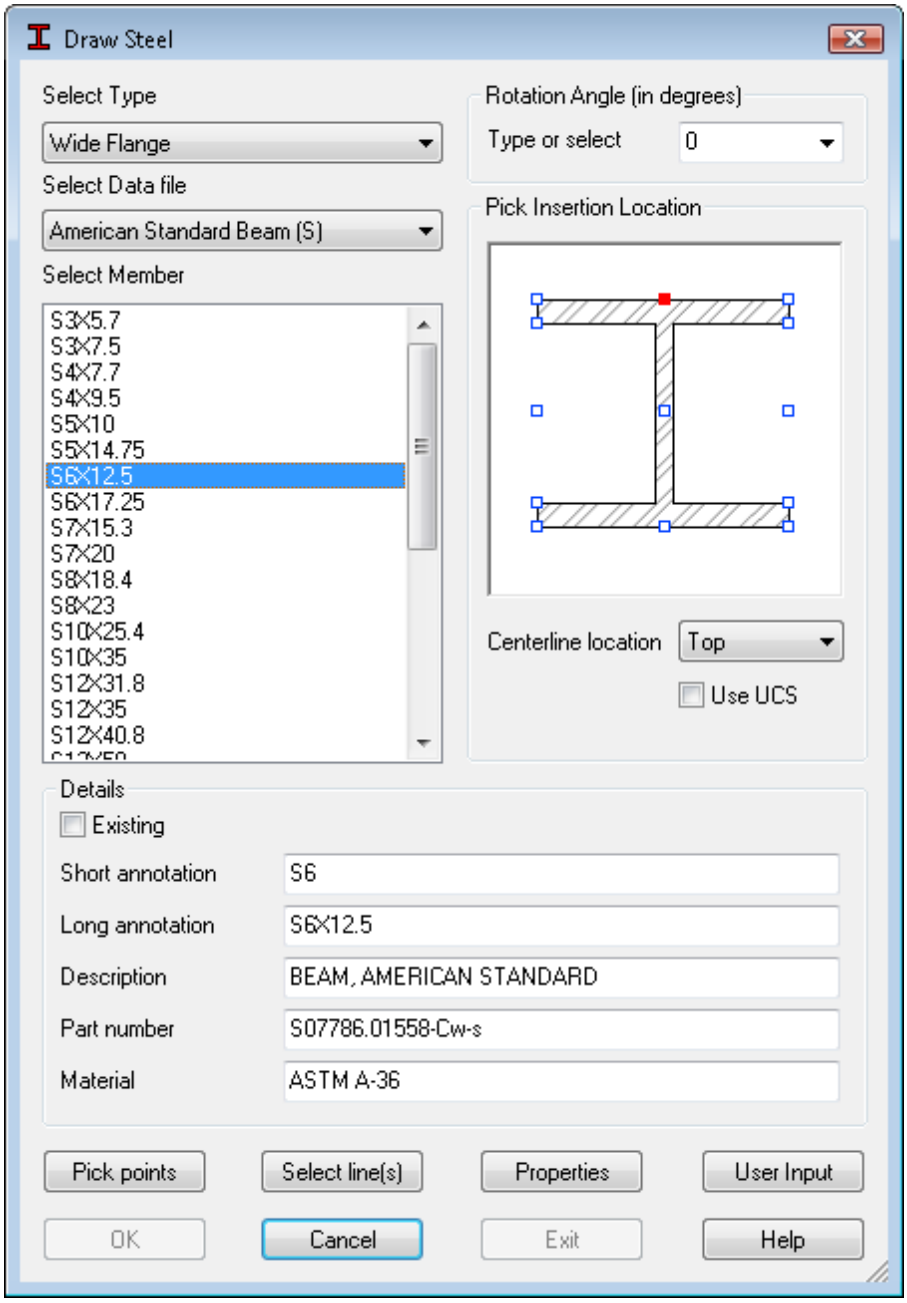
Steel toolbar:



This dialog allows the user to draw many different steel shapes into the drawing. They can be attached to lines or individual points can be picked. The ability to place members into the current UCS is also available. The location of the centerline can be specified or the angle about the centerline can be specified. Members can be placed on arcs. Members can be copied and stretched and even the size can be changed with the edit function discussed below.

Utilize the SteelLibraryDirectory (see page 63) startup variable to provide the location of the the data files associated with the steel program within CADWorx Plant. The default libraries shipped are Steel_I and Steel_M. Steel_I has available most imperial type data files all listed in inches while Steel_M has available most other metric data files. Located at the top of each data file is an entry which tells the program what type of data file it is - metric or imperial. When the program reads the data file, it looks at this entry and determines if it needs to be converted. For example, when CADWorx Plant's SystemMeasure (see page 66) is 2 (metric system with nominal inch input) and the user wants to use a imperial data file from Steel_I, the system automatically converts the data file from inches into millimeter. Therefore, it is not necessary to build a set of metric data file from the imperial data files. Just set the proper environment with CADWorx Plant and pick whatever data files desired (Steel_I or Steel_M - use the SteelLibraryDirectory setting).

Note: Steel channel and angle shapes cannot be modified using the AutoCAD MIRROR command. Use the "Filp Flange" option in the dialog to mirror channel and angle shapes.



Select Type

This drop down list will display the name for each directory pointed to with the SteelLibraryDirectory (see page 63) startup variable. Within each of these directories there is a Type.ini file which dictates what type of members are in the directory (W, T, TS, etc.). If the directory is empty (there are no data files) you will get a message indicating that. If there are more than one type W directories and the first alphabetical named directory is empty, you will get an error indicating that. You will be able to proceed and chose another directory.

Select Data File

This drop down list displays all the data files located in the directory chosen (plate will not have the option to chose). You could have a mix of metric and imperial data files in the same directory. It will automatically convert the selected data file to the proper drawing units.

Select Member

Once the data file is selected above, this list box displays the members within the data file. The default data files are supplied in alphabetical order but if desired, the data files can have the members in any order.

Details

The Existing toggle allows the user to change the existing status of the member.

The Short annotation in the first edit box is derived programmatically. The name under the MEMBER column in the data file will be stripped up to the very first "X" found and this portion of the member name will be used for the short annotation.

The Long annotation will be the whole name retrieved from the data file under the MEMBER column.

The Description is retrieved from the header in the data file. There will be a "DESCRIPTION=" which will dictate the type of the member.

The Part Number can be manually entered or can be automatically assigned by using the SteelPartNumberFile (see page 64) startup variable. Please refer to that topic for more information about part numbers.

The Material is retrieved from the header in the data file. There will be a "MATERIAL=" which will dictate the type of material.

Rotation Angle

This drop down box allows the user to indicate what angle the member should be rotated about the centerline. The drop down can be used to pick 0, 90, 180, or 270 or the user can manually enter any desired rotation angle.

Pick Insertion Location

The user can pick the desired insertion point of the member being drawn. By passing the cursor over the grip boxes, a hand cursor will appear allowing the user to select the desired point. When picked, the grip point will change color to that chosen in the drawing environment for grip colors.

The Centerline location drop down box allows the centerline of the member to be placed at the top, center, or bottom location of the member being drawn. The centerline location cannot be located at a corner, only the three locations along the Y axis of the member.

The Use UCS toggle allows the user to use the current UCS as the XY plane. For example, if the UCS is located in the same plane as a slope roof, members are easily added at this angle. All that is required is to select two points within the plane. The program will then determine it's rotation automatically.

Pick Points

This option allows the user to pick two points on the screen which will be used to locate the member.

Command: **SHAPE**
Pick start point <last point>: *pick a starting point or enter for last point*
Pick end point: *pick the ending point*

Select Line(s)

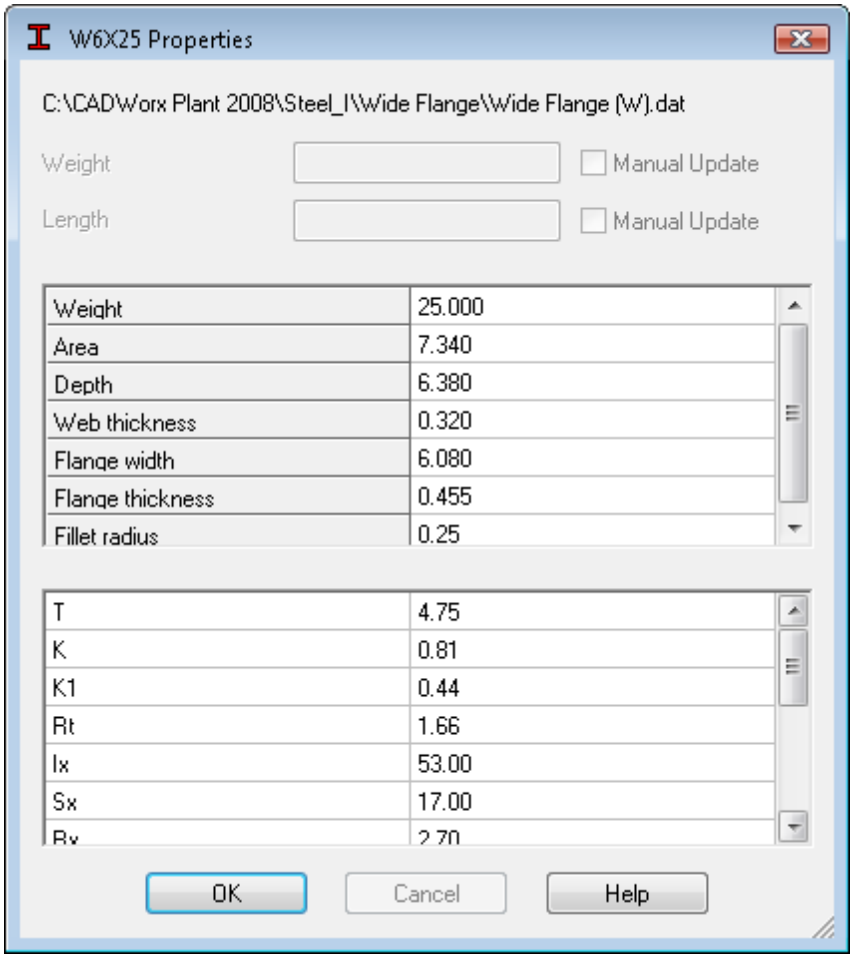
This button allows the user to select a line which will be used to draw the member.

Command: **SHAPE**
Select objects: *select a line*
Select objects: *return to end selection*

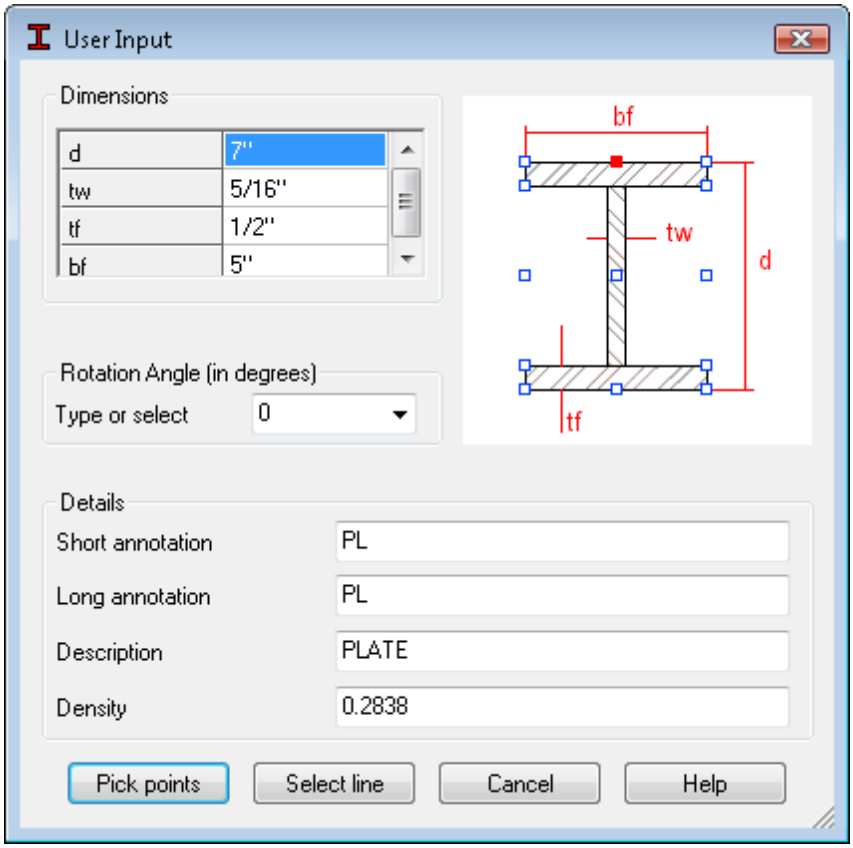
Properties

This dialog displays all the available properties of the selected member. The top of this dialog shows the reference data file. The Weight and Length entries can be manually updated by selecting the Manual Update toggle on the side of these edit boxes. The length of the member is calculated based on the shape of the solid - not the length of the centerline. For example if one end of the member was a 45 degree cut, the full length would be to the very end of the 45 degree cut - not the centerline.

The first list box's information is retrieved from the data file and is necessary for the program to draw the member. It must have at least these entries in the data file for the program to function correctly. The properties in the second list box are also retrieved from the data file and can be as many or few as the user wishes. For example, the user could add a column in the data file for the gage distance of a member. It is very important that the first list box's columns remain as is or the member will not draw correctly.



User Input



This dialog allows the user to create user defined plates. Each piece of the shape will be constructed from plate. Editing any part of the user shape later provides the plate dialog editing facility. The thickness, length and width are appended to the long annotation as they are drawn and placed.

Plate

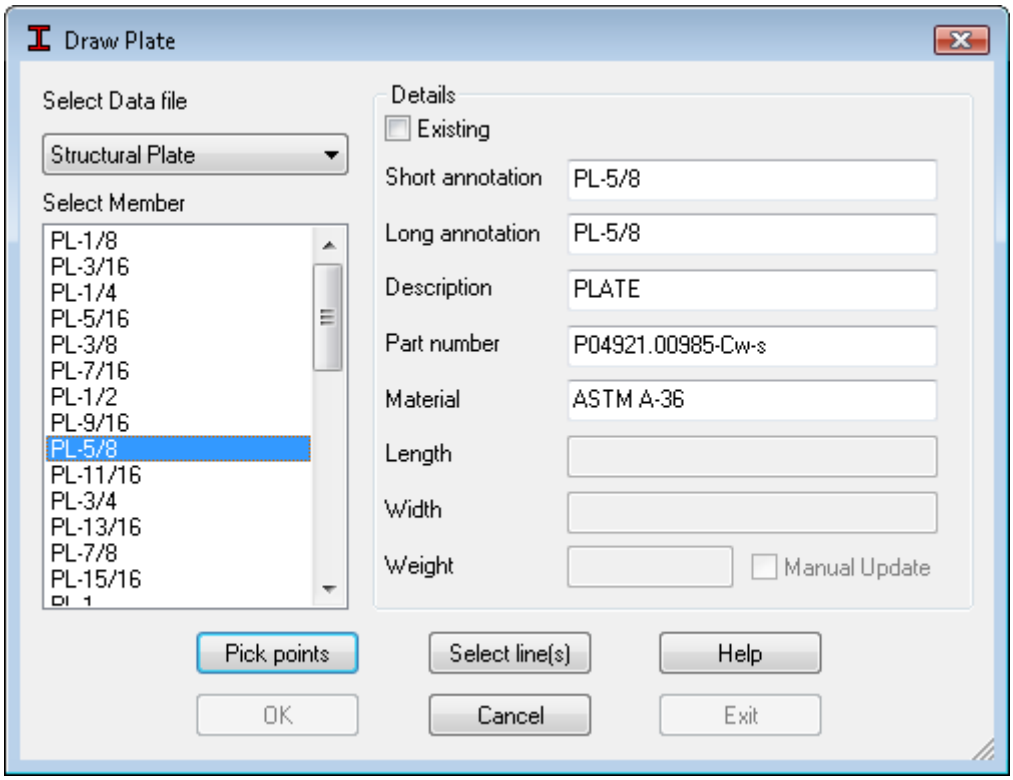
Plant menu: ▶ **Steel** ▶ **Shapes**

Command line: SPLATE

Steel toolbar:

This dialog allows the user to create plate shapes. The user must simply select a closed polyline to attach the plate to, or specify a minimum of three points to indicate the corners of the plate shape. The user will be asked for the extrusion direction after the polyline is selected, or the points have been picked.

As with the steel member dialog, utilize the SteelLibraryDirectory (see page 63) startup variable to provide the location of the the plate data files. Please refer to the Steel dialog information about how data files are automatically converted from Metric to Imperial.



Select Data File

This drop down list displays all the data files located in the directory chosen (plate will not have the option to chose). You could have a mix of metric and imperial data files in the same directory. It will automatically convert the selected data file to the proper drawing units.

Select Member

Once the data file is selected above, this list box displays the members within the data file. The default data files are supplied in alphabetical order but if desired, the data files can have the members in any order.

Select Line(s)

This button allows the user to select a closed polyline which will be used to draw the plate.

Command: **SPLATE**
Select objects: *select a closed polyline*
Select objects: *return to end selection*
Plate extrude direction [Positive/Negative] <Positive>: *enter for a positive Z direction or type N for negative*

Pick Points

This button allows the user to pick multiple points which will then be used to draw the plate.

Command: **SPLATE**
Select objects: *select a closed polyline*
Select objects: *return to end selection*
Plate extrude direction [Positive/Negative] <Positive>: *enter for a positive Z direction or type N for negative*

Details

The Existing toggle allows the user to change the existing status of the member.

The Short annotation in the first edit box is derived programmatically. The name under the MEMBER column in the data file will be stripped up to the very first "X" found and this portion of the member name will be used for the short annotation.

The Long annotation will be the whole name retrieved from the data file under the MEMBER column.

The Description is retrieved from the header in the data file. There will be a "DESCRIPTION=" which will dictate the type of the member.

The Part Number can be manually entered or can be automatically assigned by using the SteelPartNumberFile (see page 64) startup variable. Please refer to that topic for more information about part numbers.

The Material is retrieved from the header in the data file. There will be a "MATERIAL=" which will dictate the type of material.

The Length and Width is computed by finding a bounding box around the plate aligned along a alignment axis. The alignment axis through the plate will be the axis formed between the long segment along the polyline which the plate was made from.

The Weight is computed from the actual shape of the plate based on the weight recorded in the data file. The Weight entry can be manually updated by selecting the Manual Update toggle.

Cope

Plant menu: ▶ **Steel** ▶ **Cope**

Command line: SCOPE

Steel toolbar: 

This command allows steel members to be coped to other steel members. When coping, the user has two options: bolted coping and welded coping. The bolted coping is controlled by the SteelCopeBoltDistance (see page 63) startup variable and will provide an offset or gap between the coped member and the boundary member. The welded cope is controlled by the SteelCopeWeldDistance (see page 63) startup variable and will provide an offset or gap between the coped member and the boundary member.

A single member can be coped to another single member or many members can be coped to many other members. When coping a single member, the user simply selects one member to be coped and another member which will provide the boundary. When coping many members to many other members, the user selects all the members to be coped and then select all the boundary members. When members cross other members, they will be cut into two pieces and will become separate members.

Command: **SCOPE**
Select objects to cope: *pick the object to cope*
Select objects: *select other objects if desired*

Select objects: *return to end selection set*

Select coping boundaries: *select the objects to cope against*
Select objects: *select other object if desired*

Select objects: *return to end the selection set*

Bolted or welded cope? [**Bolt**/**Weld**] <Bolt>:

Bolt

This option copes the member(s) with an offset as specified in the SteelCopeBoltDistance startup variable.

Weld

This option copes the member(s) with an offset as specified in the SteelCopeWeldDistance startup variable.

Cut

Plant menu: ▶ **Steel** ▶ **Cut**

Command line: SCUT

Steel toolbar: 

This command allows steel members to be cut along a plane. The user is required to select a line or pick points and then select the members to be cut. This command will cut the steel members along the plane that is formed from the line or the points picked. The new pieces will be updated with new lengths, weights, etc.

Command: **SCUT**
Cut by [**Line**/**Points**] <Line>:

Line

This option determines the plane to cut by using the two end points of the line and its extrusion direction (the Z axis of the plane the line was drawn in).

Select cutting line: *pick the line desired*
Select objects: *return to finish selection*

Points

This options allow the user to pick three point from which the plane will be determined.

Pick first point on plane: *pick the first point on the plane*
Pick second point on plane: *pick the second point on the plane*
Pick third point on plane: *pick the third point on the plane*

After either of the two options above, the final steps will require a steel members to be selected. More than one steel member can be selected.

Select CADWorx steel objects to cut: *select as many members as desired*
Select objects: *return to finish selection*

Note: This command cannot cut plate members.

Miter

Plant menu: ➤ **Steel** ➤ **Miter**

Command line: SMITER

Steel toolbar: 

This command miters two structural pieces to fit each other. The two structural pieces must be in the same plane and the ends of the members must connect.

Command: **SMITER**
Select members to miter: *pick two or several members at a time*
Select objects: *enter to finish selection set*

2 members were mitered...

An error message will be displayed if the members could not be mitered.

Selected members could not be mitered...

Union

Plant menu: ▶ **Steel** ▶ **Union**

Command line: SUNION

Steel toolbar:

This command unions two steel members into one member. The following items must be identical for two members to be joined into one.

- Steel type (i.e. W Shape, C Shape etc.)
- Insertion location
- Centerline location
- Rotation angle
- Data file
- Member size
- Short annotation
- Long annotation
- Description
- Material
- Part number

Most importantly, the two center lines of the beams must meet each other and cannot intersect.

Command: **SUNION**
Select members to union: *select one steel member*
Select objects: *select the other steel member*
Select objects: *return to finish selection set*

2 members were combined...

If some condition above is not satisfied, an error message would occur and the members will not be mitered.

Note: Curved members cannot be joined with straight members.

CG Generator

Plant menu: Steel C.G. Generator

Command line: SCG

Steel toolbar:

This command allows the user to calculate the center of gravity(CG) of the steel members (including plate) in a drawing. The user must select the desired members.

```
Command: SCG
Select objects: pick a point or type A for all
Specify opposite corner: pick another point
Select objects: return to finish selection
```

After selecting the desired components, a prompt will ask whether the report need to be saved.

```
Save report [Yes/No] <No>:
```

If the user answers Yes to the above prompt, a standard file selection dialog will appear requesting a file name. The user can save the report as a text or html file. Next, the command will prompt the user whether they wish to view the report at the command line.

```
View file [Yes/No] <Yes>:
```

If the the user answers Yes to the above prompt, the command will process the selection and display the report as shown below.

```
W6X20          W6X20
Length = 14'-1 5/8"     Weight = 282.67
X = 66.87 ft    Y = 47.49 ft    Z = -0.26 ft
```

At the end of the command, the following information will be displayed.

```
Weight Summation          =     13000.00 lb
Summation (X * Weight)    =     9041625.65 ft-lb
Summation (Y * Weight)    =     1190958.29 ft-lb
Summation (Z * Weight)    =     -51285.00 ft-lb
Center of Gravity X, Y, Z =     57.9591 ft    7.6343 ft    -0.3287 ft
```

A prompt will ask whether a marker should be placed in the drawing and then the final prompt will tell where the output file(if any) was written.

```
Insert drawing marker [Yes/No] <Yes>:
Center of gravity report written to D:\Dwg\Drawing1.txt
```

Note: To combine center of gravity reports for piping and steel components, take the weight summations and add them together. Take each separate moment (ft-lb) X, Y, Z and add them together separately. To find the new X centroid location, divide the total X moment by the total weight. To find the new Y and Z centroid location, divide them by the total weight.

Ladder

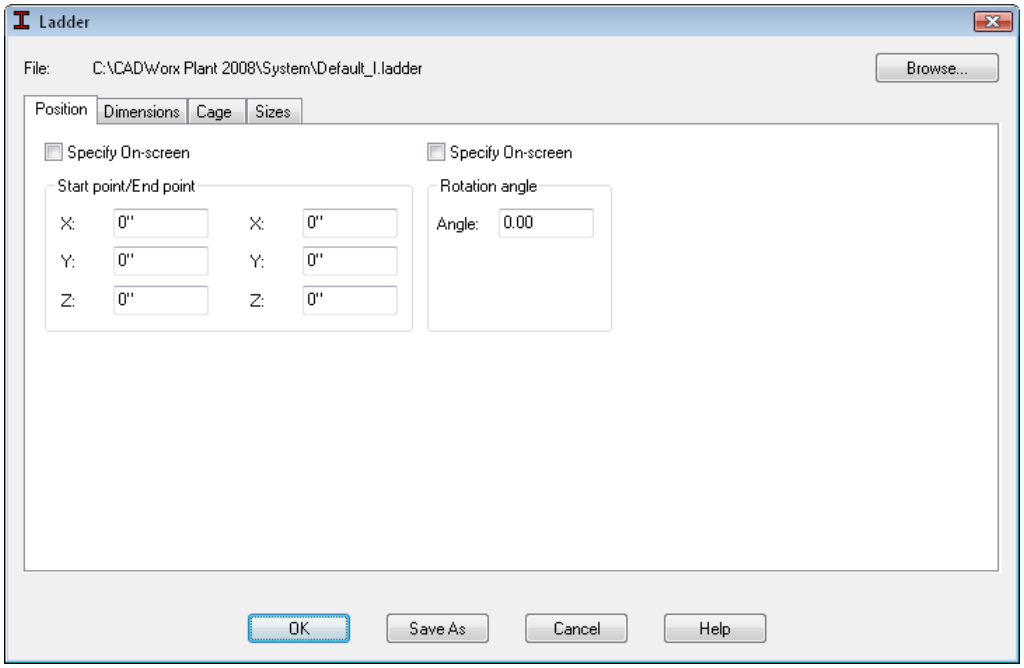
Plant menu: ▶ **Steel** ▶ **Ladder**

Command line: SLADDER

Steel toolbar:

These dialogs allow the user to create ladders. The user must specify the start and end points of the ladder, as well as the ladder's rotation angle.

Position



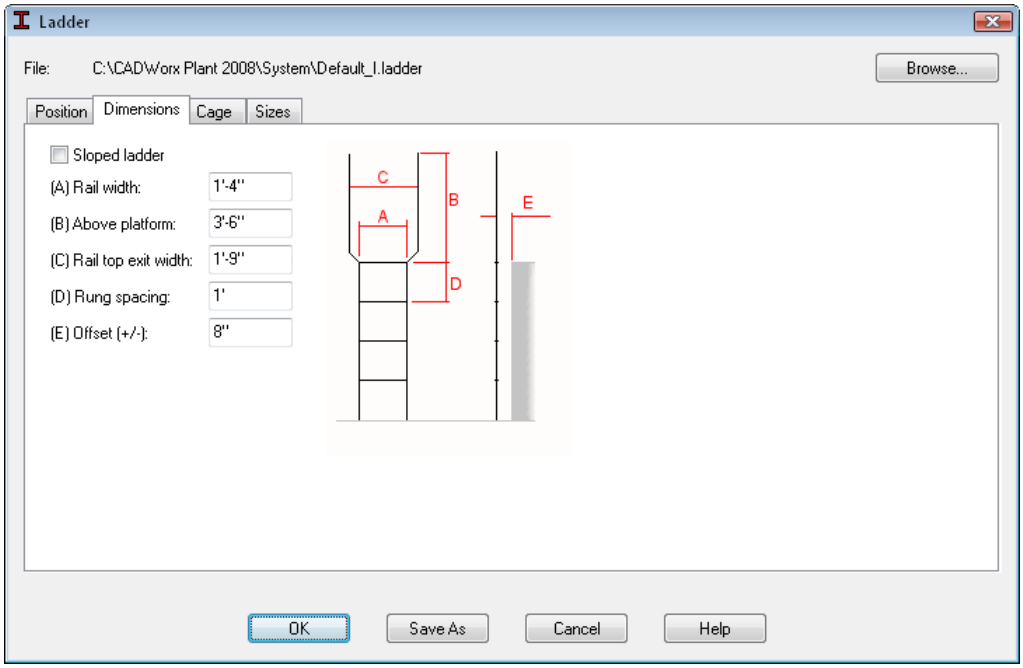
This dialog is for positioning the ladder.

Specify On-screen toggle disables the *Start/End Point* edit boxes and allows the user to pick a location on the screen after clicking the OK button.

The user can enter exact X, Y, and Z coordinates for start and end points using the edit boxes provided.

Rotation Angle allows different rotation angles to be applied to the entire ladder.

Dimension



This dialog allows dimensioning of the ladder to exact user requirements.

The *Sloped ladder* toggle allows the ladder to be sloped. The exact slope of the ladder is determined by the X or Y location of the start and end points in the previous dialog. This toggle must be ON for it to take affect.

Rail width is the inside distance from rail to rail.

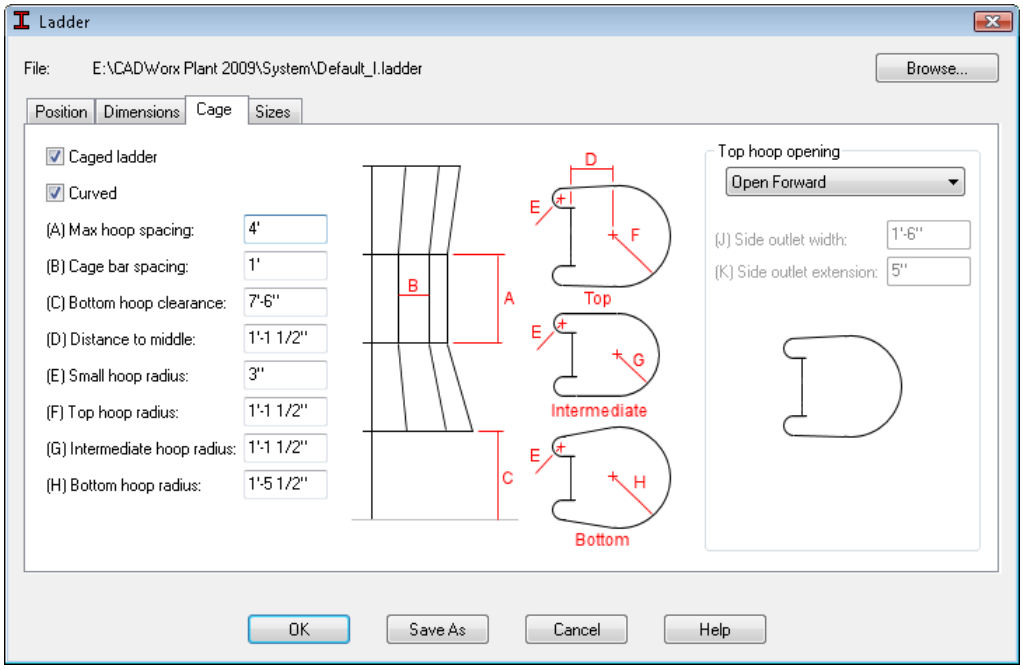
Above platform is the distance the ladder rails (offset extension of rail) are to be extended above the top rung.

Rail top exit width is the inside to inside distance of the ladder opening above the platform.

Rung spacing is the center to center distance of the ladder rungs.

Offset (+/-) is the distance the ladder is offset from the platform.

Cage


Ladder
File: E:\CADWorx Plant 2009\System\Default_I.ladder
Browse...
Position
Dimensions
Cage
Sizes
Caged ladder
Curved
(A) Max hoop spacing: 4'
(B) Cage bar spacing: 1'
(C) Bottom hoop clearance: 7'-6"
(D) Distance to middle: 1'-1 1/2"
(E) Small hoop radius: 3"
(F) Top hoop radius: 1'-1 1/2"
(G) Intermediate hoop radius: 1'-1 1/2"
(H) Bottom hoop radius: 1'-5 1/2"
A
B
C
D
E
F
G
H
Top
Intermediate
Bottom
Top hoop opening
Open Forward
(J) Side outlet width: 1'-6"
(K) Side outlet extension: 5"
OK
Save As
Cancel
Help

This dialog allows the user to adjust the cage parameters of the ladder.

The *Cage ladder* toggle places a cage on the ladder if the height of the ladder is over the minimum distance.

The *Curved* toggle allows the hoop bars to connect to the ladder using a curved connection.

Max. hoop spacing is the distance from center to center of the intermediate hoops.

Cage bar spacing is the distance from the center to center of the vertical bars on the cage.

Bottom hoop clearance is the distance from the base elevation or grade to the bottom hoop.

Distance to middle is the distance from the ladder rungs to the center of the cage.

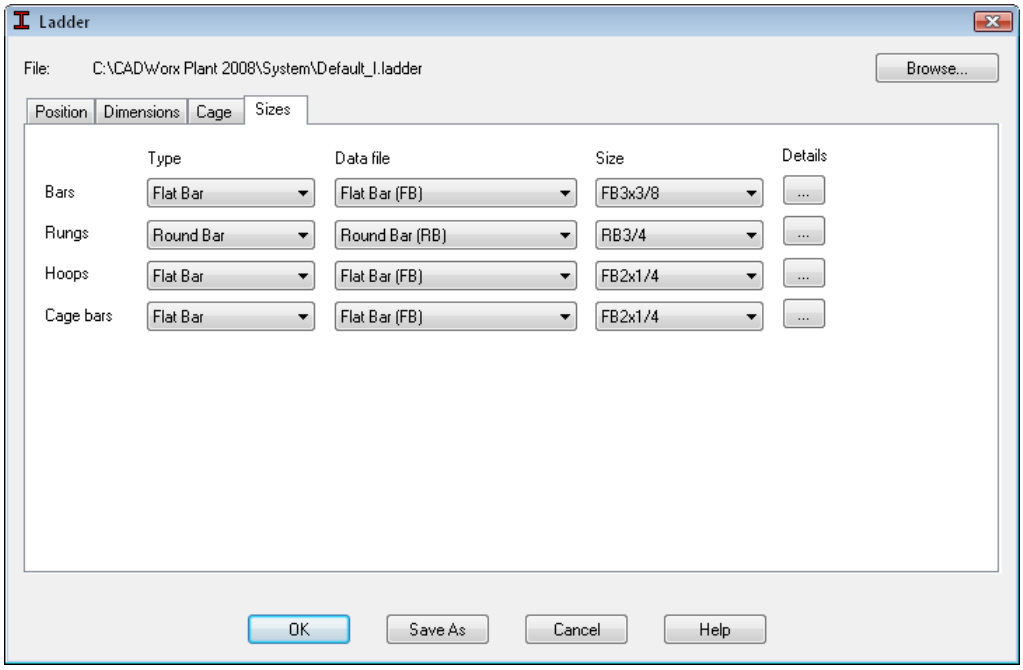
Small hoop radius is the radius of the hoop bar at the ladder rail location. If the *Curved* toggle is un-checked, this parameter becomes the *Connection distance* of the hoop bar at the ladder rail location.

Top hoop radius is the radius of the top hoop from the center of the cage.

Intermediate hoop radius is the radius of the intermediate hoop from the center of the cage.

Bottom hoop radius is the radius of the bottom hoop from the center of the cage.

Sizes

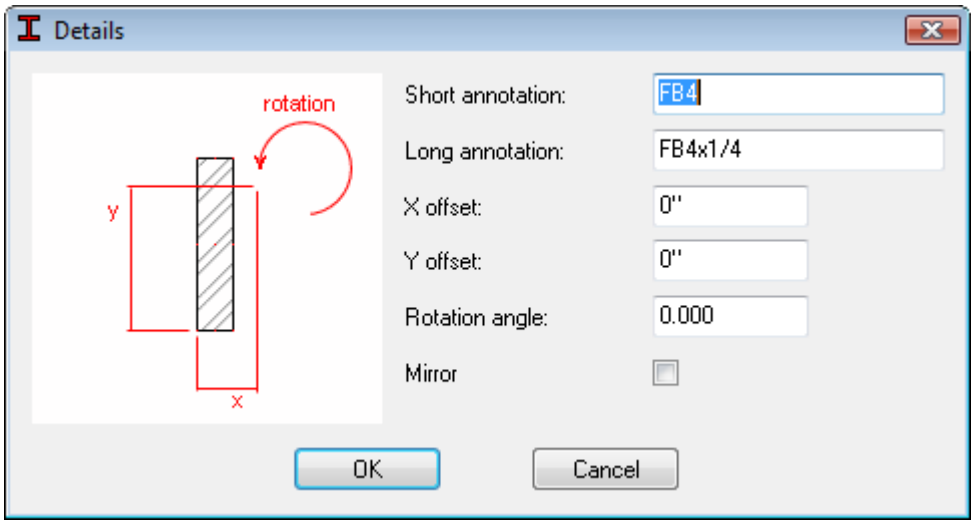

Ladder
File: C:\CADWorx Plant 2008\System\Default_I.ladder
Browse...
Position
Dimensions
Cage
Sizes
Type
Data file
Size
Details
Bars
Flat Bar
Flat Bar (FB)
FB3x3/8
Rungs
Round Bar
Round Bar (RB)
RB3/4
Hoops
Flat Bar
Flat Bar (FB)
FB2x1/4
Cage bars
Flat Bar
Flat Bar (FB)
FB2x1/4
OK
Save As
Cancel
Help

This dialog allows the user to set the members to be used in the ladder along with any offsets or rotations required.

The *Bars, Rungs, Hoops,* and *Cage bars* rows allows the selection of different components to be used with the ladder. Use the drop downs in the *Type* column to select the member type.

After the *Type* is selected, The *Data File* column allows the selection of the exact data file to be used. The data files appearing in the list are controlled by the SteelLibraryDirectory (see page 63) variable in the configuration file.

The *Size* column allows selection of a member from the data file selected.



The *Details* button (with three dots on it) displays the *Details* dialog and allows the member to be offset, rotated, or mirrored. Short and long annotation can also be changed to reflect in a bill of material.

Short annotation allows the member to be labeled as per any user requirement.

Long annotation is used the same as the Short annotation.

X offset is used to offset the member along the x axis. Please note that the base point is not always located at the centroid of the member.

Y offset is used to offset the member along the y axis.

Rotation angle can be used to rotate the member any degree.

The *Mirror* toggle simply mirrors the member about it's base point as shown in the diagram.

Stair

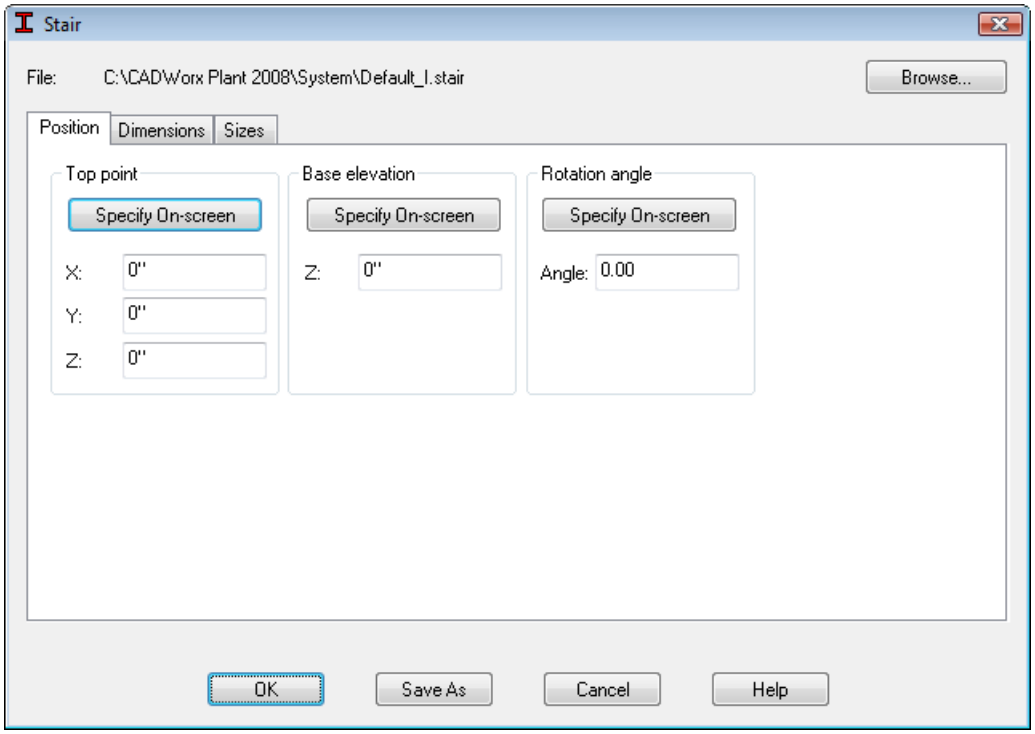
Plant menu: ▶ **Steel** ▶ **Stair**

Command line: SSTAIR

Steel toolbar:

These dialogs allow the user to create stairs. The user must specify the top point of the stair and then a base elevation. Rotation will also be required.

Position



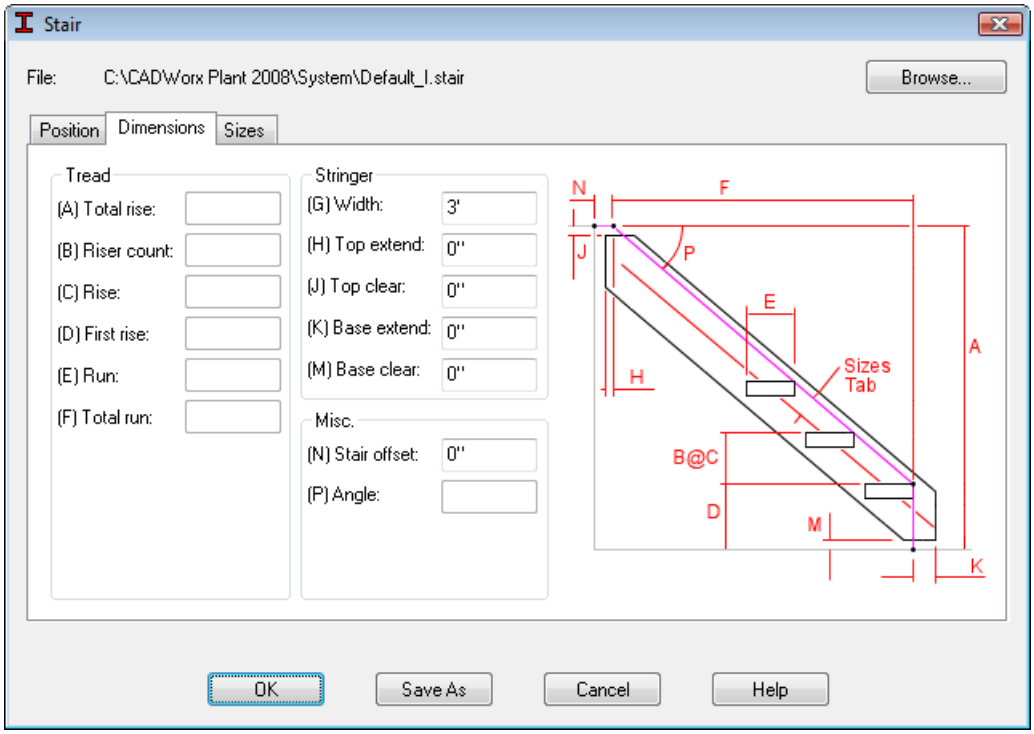
This dialog is for positioning the ladder.

The *Top Point* allows the entry of exact X, Y, or Z entries or it can be specified on the screen. When clicking on the *Specified On-screen* button, the dialog will disappear allowing a point to be picked on the screen. The dialog will then re-appear.

The *Base Elevation* also allows exact entry or can be *Specified On-screen*.

Rotation Angle is provided to allow the stair to be rotated to an exact degree.

Dimension



This dialog allows the user to change dimensions and cut backs for the ladder as shown in the diagram.

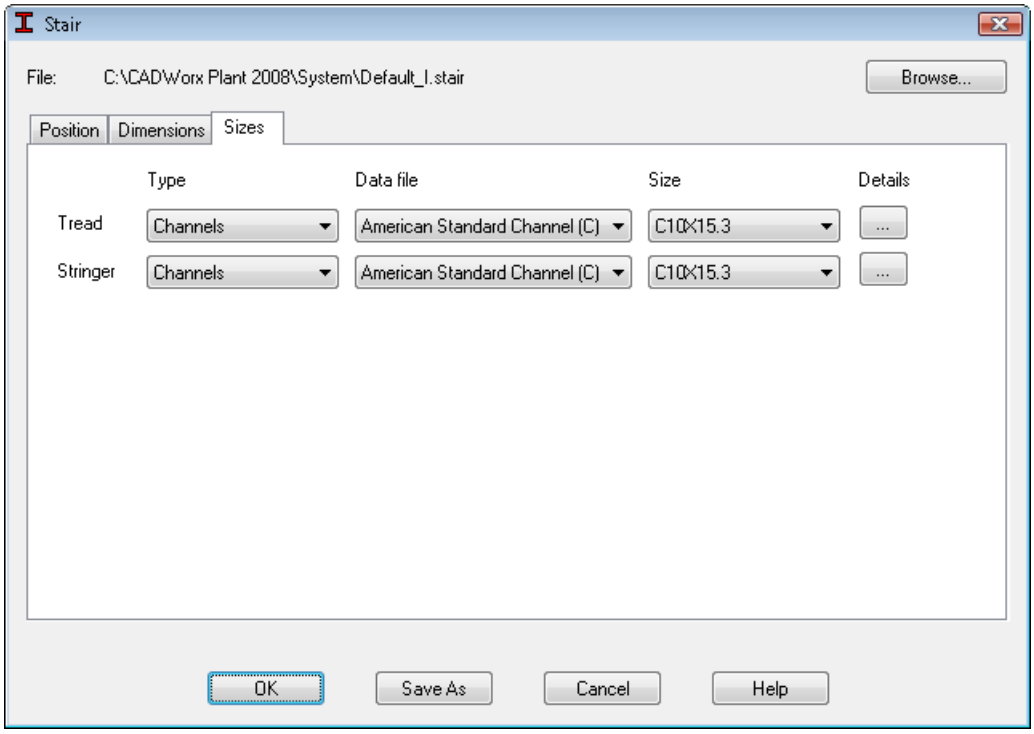
The *Tread* sections allow the *Total rise*, *Riser count*, *Rise*, *First rise*, *Run* and *Total Run* to be entered. This group will be disabled unless a start and finish elevation are entered in the *Position* tab dialog. Upon entering this information, this section will automatically populate.

Changes made to	**affects these edit boxes**
(A) Total rise	Rise
(B) Riser count	Rise, Run
(C) Rise	Total rise
(D) First rise	Rise
(E) Run	Total run
(F) Total run	Run

The *Stringer* section allows the *Width*, *Top extend*, *Top clear*, *Base extend* and *Base clear* to be changed as per the diagram.

The *Misc.* section allows the *Stair offset* to be changed and will indicated the stair *Angle* from the horizontal.

Sizes

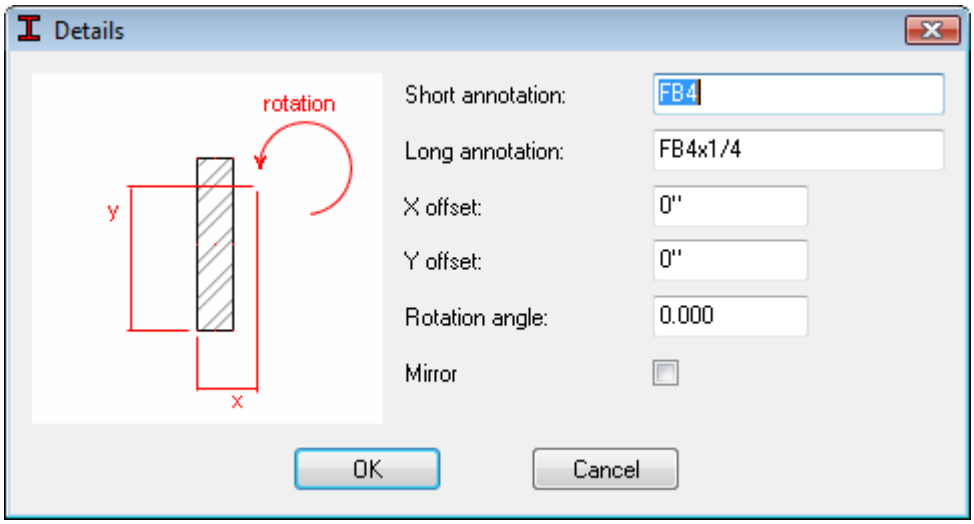

Stair
File:
C:\CADWorx Plant 2008\System\Default_I.stair
Browse...
Position
Dimensions
Sizes
Type
Data file
Size
Details
Tread
Channels
American Standard Channel (C)
C10X15.3
...
Stringer
Channels
American Standard Channel (C)
C10X15.3
...
OK
Save As
Cancel
Help

This dialog allows the user to set the members to be used in the stair along with any offsets or rotations required.

The *Treads* and *Stringers* rows allows the selection of different components to be used with the stair. Use the drop downs in the *Type* column to select the member type.

After the *Type* is selected, The *Data File* column allows the selection of the exact data file to be used.

The *Size* column allows selection of a member from the data file selected.



The *Details* button (with three dots on it) displays the *Details* dialog and allows the member to be offset, rotated, or mirrored. Short and long annotation can also be changed to reflect in a bill of material.

Short annotation allows the member to be labeled as per any user requirement.

Long annotation is used the same as the Short annotation.

X offset is used to offset the member along the x axis. Please note that the base point is not always located at the centroid of the member.

Y offset is used to offset the member along the y axis.

Rotation angle can be used to rotate the member any degree.

The *Mirror* toggle simply mirrors the member about it's base point as shown in the diagram.

Handrail

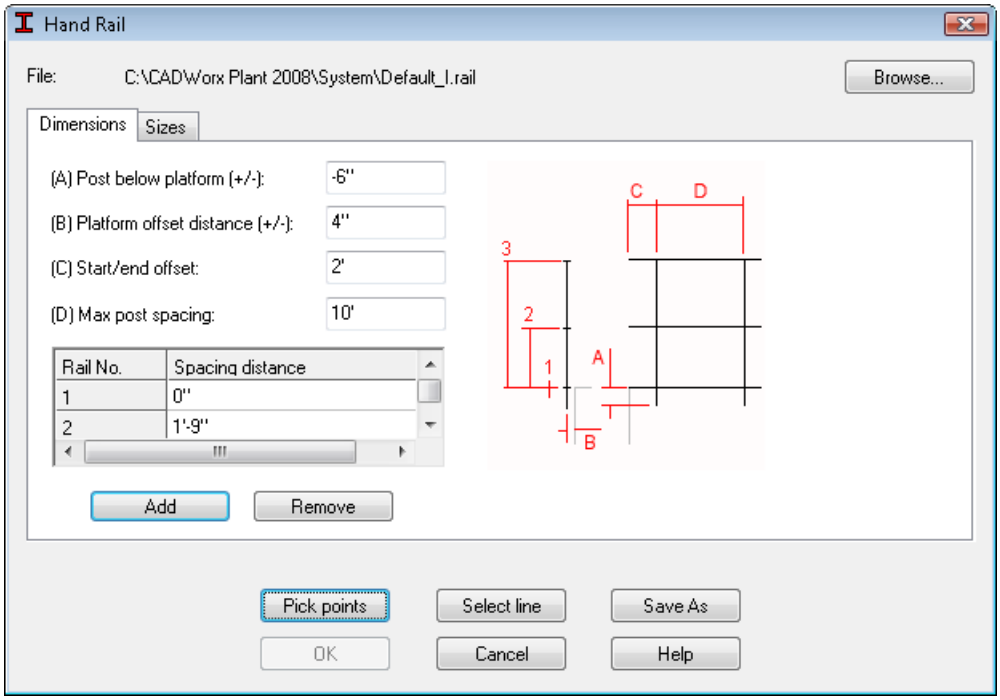
Plant menu: ➤ **Steel** ➤ **Hand Rails**

Command line: SHRAIL

Steel toolbar:

This dialog allows the user to create handrails. The user must either specify the corner points of the handrail, or must select a polyline or 3D polyline that specifies the path of the handrail. The user will be asked to specify the direction in which to offset the handrail after the points have been picked, or the polyline has been selected.

Dimensions



This dialog allows the user to set the dimensions of the hand rail.

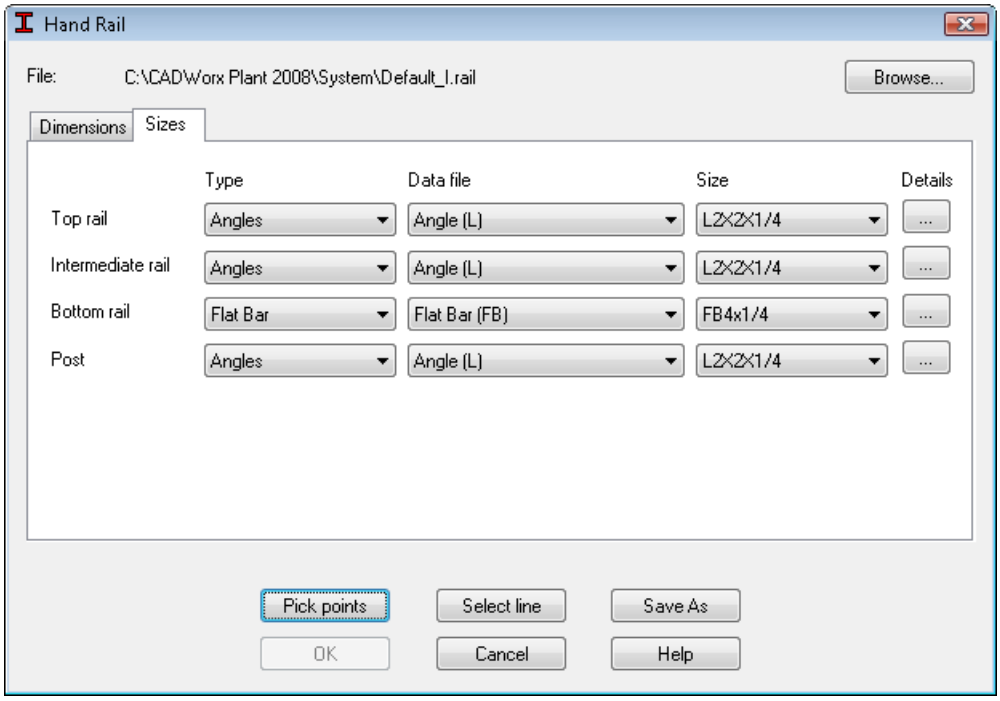
Post below platform (+/-) allows the hand rail to be placed above or below the platform to which it is attached to.

Platform offset dist (+/-) allows the hand rail to be offset from the platform. It can be offset off the platform as well as offset onto the platform.

Start/end offset places a post or stanchion a particular distance from the end of the hand rail.

Max post spacing is the distance in between the post or stanchions.

Sizes

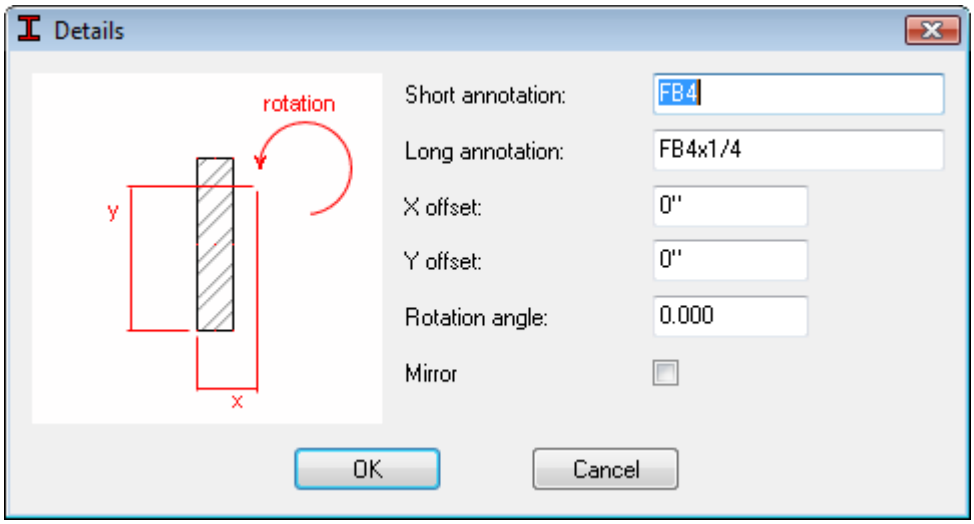


This dialog allows the user to set the members to be used in the hand rail along with any offsets or rotations required.

The *Top rail*, *Intermediate rails*, *Bottom rail*, and *Post* rows allows the selection of different components to be used with the hand rail. Use the drop downs in the *Type* column to select the member type.

After the *Type* is selected, The *Data File* column allows the selection of the exact data file to be used.

The *Size* column allows selection of a member from the data file selected.



The *Details* button (with three dots on it) displays the *Details* dialog and allows the member to be offset, rotated, or mirrored. Short and long annotation can also be changed to reflect in a bill of material.

Short annotation allows the member to be labeled as per any user requirement.

Long annotation is used the same as the Short annotation.

X offset is used to offset the member along the x axis. Please note that the base point is not always located at the centroid of the member.

Y offset is used to offset the member along the y axis.

Rotation angle can be used to rotate the member any degree.

The *Mirror* toggle simply mirrors the member about it's base point as shown in the diagram.

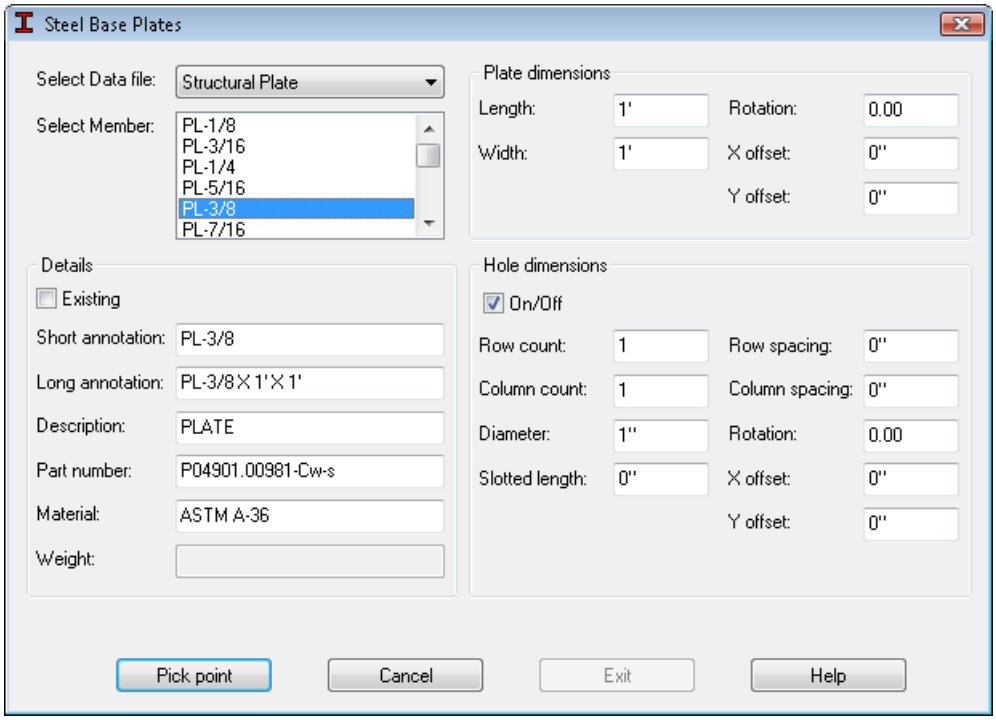
Base Plate

Plant menu: ▶ **Steel** ▶ **Base Plate**

Command line: SBPLATE

Steel toolbar:

This dialog allows the user to create base plates. Base plates can be placed in open space or attached to the end of another steel member. CADWorx Plant will automatically cope the steel member with the base plate.



Data File and Member

These list boxes allows the user to select a plate type and a size or thickness within the plate data file. Upon entering the main dialog, *Select data file:* and *Select member:* will be blank. Selecting a data file will present all the sizes within the data file in a list. The data files displayed here are based on the SteelLibraryDirectory (see page 63) variable in the configuration file.

Details

The Existing toggle allows the user to change the existing status of the member.

These edit boxes will be filled out automatically when the user selects a data file and member within the data file. All of these edit boxes can be edited to reflect any user requirements.

The weight of the base plate displays the weight of the plate minus the weight of any hole(s) that were placed on it.

Dimensions

This group of edit boxes allow the size of the plate to be entered along with a rotation and an offset if required. The offset is from the centroid of the plate.

Hole Dimensions

This group allows the hole(s) dimensions to be entered. A hole can also have a slotted length which is the distance at the center.

Auto Cope

Plant menu: ➤ **Steel** ➤ **Auto Cope**

Command line: SAUTOCOPE

Steel toolbar: 

This command allows the user to set automatic coping on or off. When the system is on and two members are coped, they will remain coped even if one of the member is changed. They will automatically re-cope to the shape of the new member.

Command: **SAUTOCOPE**
Auto cope is on...

Run the command again and it will turn the system off.

Command: **SAUTOCOPE**
Auto cope is off...

Edit

Plant menu: ▶ **Steel** ▶ **Edit**

Command line: SCEDIT

Steel toolbar:

This command allows the editing of the steel shapes and plate. Users can invoke this command from the command line, pulldown menu, or simply by double clicking on the shape. The command will display the exact same dialog that was used to place the shape. The user can change the size, the type of shape, the description, or any field in the dialog which is editable. See the Steel Shapes (see page 426) or Steel Plate (see page 432) command above for more information.

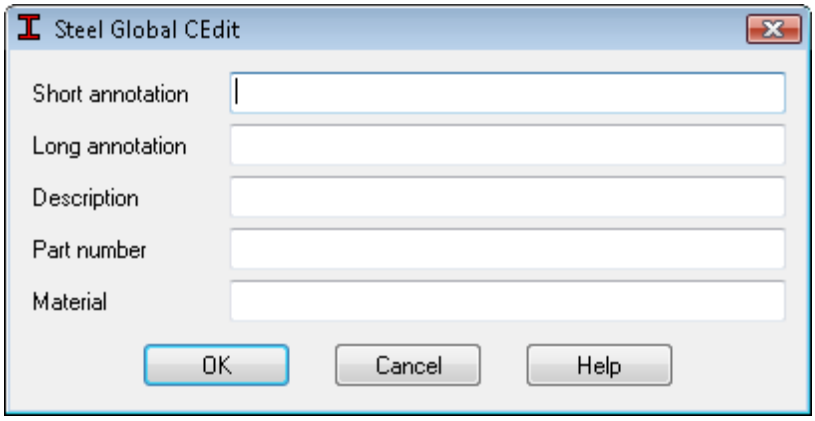
Global Edit

Plant menu: ▶ **Steel** ▶ **Global Edit**

Command line: SGCEDIT

This command allows multiple components to be modified at one time. When multiple components are selected, this command displays the following dialog. Each field within this dialog will be blank. If any field or edit box is modified within this dialog, then that field will be changed throughout all the components that were selected. The edit boxes or fields that were not modified or changed will not be updated within the components.

Command: **SGCEDIT**



When finished, a prompt will appear indicating the total number of components that were updated.

Updated 44 component(s)...

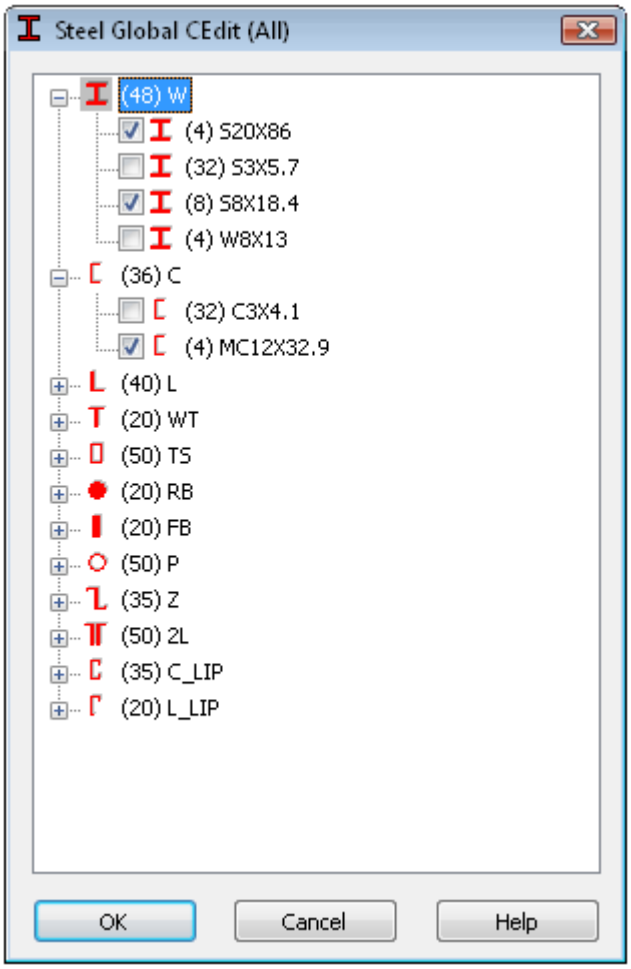
Steel - Global Edit All

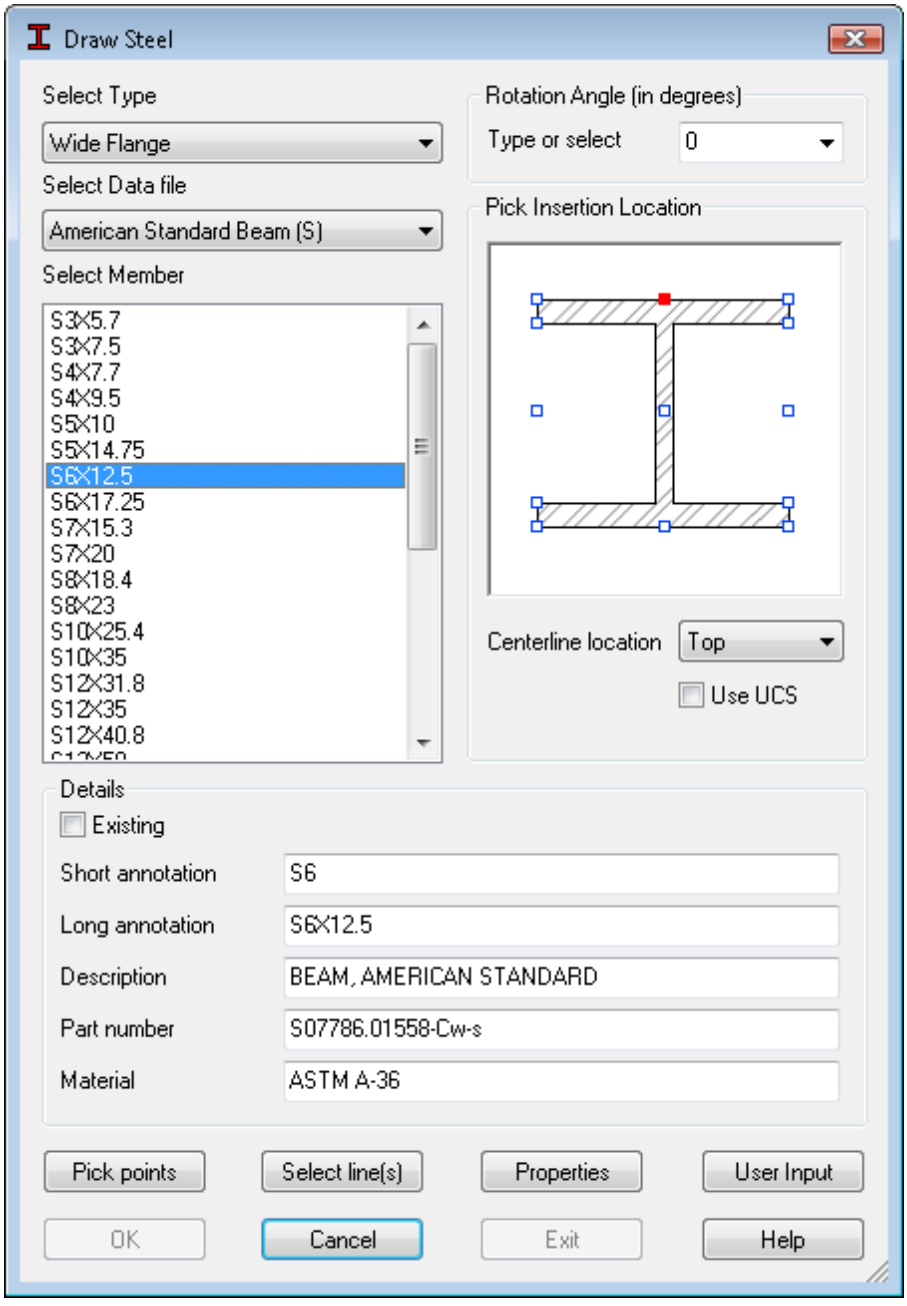
Plant menu: ▶ **Steel** ▶ **Global Edit All**

Command line: SGCEDITALL

This command allows multiple components to be modified at one time. When multiple components are selected, this command displays the following dialog. The dialog will allow you to select different types of beam by the short annotation. For each type selected, the Draw Steel dialog will appear to allow changes to the beams selected.

Command: **SGCEDITALL**





When finished, a prompt will appear indicating the total number of components that were updated.

```
Type '501': updated 4 steel element(s)
Type '501': updated 8 steel element(s)
Type '502': updated 4 steel element(s)
Finished updating 16 steel components
```

Set Component

Plant menu: ➤ **Utility** ➤ **Set Size by Component**

Command line: COMP2STEEL

Settings toolbar:

This command allows the user to set steel size by simply selecting a component.

Command: **COMP2STEEL**
Select steel component to set member size: *pick component*

The new size is then displayed.

Type: Wide Flange
Data file: American Standard Beam (S)
Member: S4X9.5
Centerline location: 0
Insertion location: 11

Generic Attachment - Steel

Command line: SGENERIC

This routine attaches intelligence to any graphics within the drawing. It later can be used in bill of material extraction, labeling, etc. for various equipment or components not provided with CADWorx Plant. After manually constructing a component, use this routine to attach xdata information.

To pinpoint the center of gravity locations use the Osnap commands. For example, to place information on a pump and have it reflect the CG correctly, draw a centerline and pick *MIDpoint* or *NEArest* to the end of the centerline toward the electrical motor or the exact known location.

Command: **SGENERIC**
Enter short annotation: *enter string (W6)*
Enter long annotation: *enter string (W6X25)*
Enter description: *enter string (Wide Flange)*
Enter part number: *enter string (123456789)*
Enter material: *enter string (ASTM A-36)*
Enter weight: *enter real*
Enter length: *enter real*
Select object: *select any object*

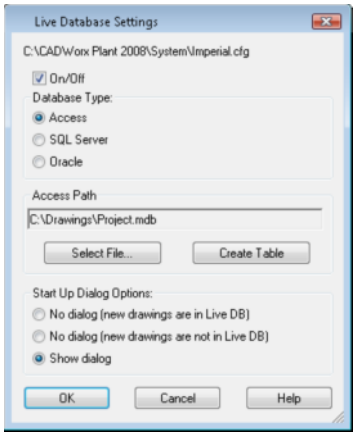
Database

Steel - Live Database Setup

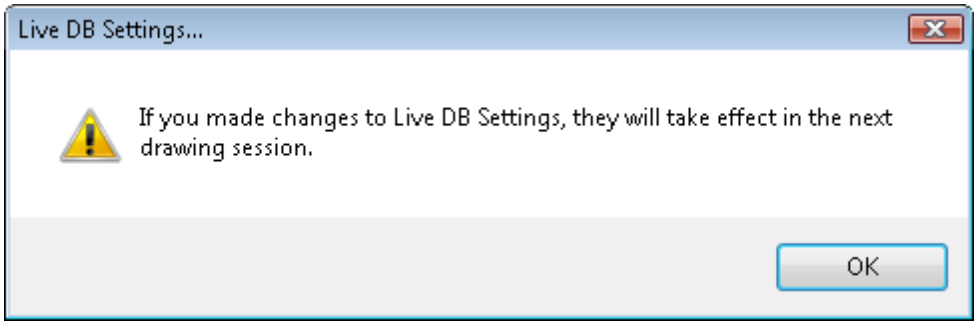
Command line: SLIVEDB

CADWorx Plant supports a fully functional live external database. The settings for this database are controlled through this dialog. This system will react according to the configuration file listed in the top of this dialog. When a drawing is started and a configuration file is located within the same directory the drawing will initialize according to that configuration file - live database or not.

Selecting this function, will present the Live Database Settings dialog shown next.



After making any changes to the live database settings, click OK to save the changes. All changes to live database settings take effect in the next drawing session as shown below.



Note: When a component is erased from the drawing, and there is a live database active, it is marked for deletion in the database. When the drawing is closed, the appropriate action will take place within the database. If the drawing is not saved the item is not deleted. If the drawing is saved, then the item is deleted.

Database Type

These options toggle the type of database to be used as the live database in CADWorx Plant. Depending on the *Database Type* chosen, the path to the database or schema is displayed below.

Please refer to Oracle.txt and SQLServer.txt located in the <INSTALL-DIR>\SYSTEM directory for more information on setting up Oracle or SQL Server respectively.

On/Off

This toggle turns the live database system on and off.

When the system is on, all components drawn are updated to the database instantaneously. Changes can be made in the drawing or in the database.

When the system is off, all CADWorx Plant component information is stored within the drawing only.

Live DB - Path

Once the database type is chosen, click on *Create Table* button to create the PIPE table in the database. This is for creating a completely new database table. If the database type is Access, the user can also select an existing database by clicking on the *Select File* button.

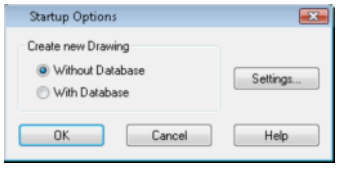
Start Up Dialog Options

There are three options for the start up dialog.

- *No dialog (new drawings are always in the live database)*. This option requires that a new drawing be given a name and saved immediately on creation.
- *No dialog (new drawings are never in the live database)*. This option does not add new drawings to the live database.
- They can be added to the live database using the command.

Note: Drawings that use a live database should be in the same directory with the appropriate configuration file.

- *Show dialog*. This options displays the following dialog every time a new drawing is created. The dialog allows the user to add drawings to the live database or not.



The *Settings* button allows the user to change settings that control whether this dialog appears on start up or not.

Note: Existing drawing can be added to the database by using the DBFGEN (see page 236) command (Pipe LiveDB) or SDBFGEN (Steel LiveDB)

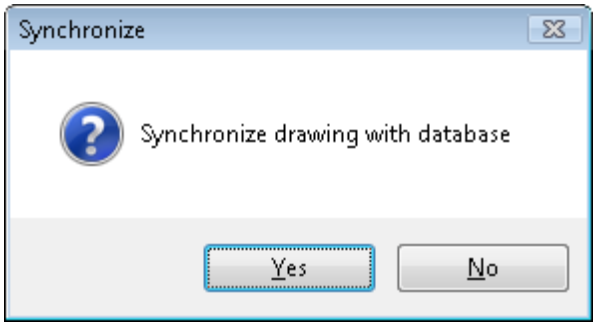
Steel - Synchronize

Command line: SSYNC

This function checks all components in the drawing and updates them according to the current value within the database tables. This function will check every component within the drawing. This command will not write new records in the database for new components that are in the model that are not in the database.

Command: **SSYNC**

The user will be prompted to confirm the synchronization of the drawing with the database as shown below.



A user would use the SYNC function to update the drawing especially if they know that someone is working in the project database.

When a drawing(s) has a live database attached, the user can enter the database and change fields located within the STEEL table. The columns values which can be modified are the SHORT_ANNOTATION, LONG_ANNOTATION, DESCRIPTION, PART_NUMBER, and MATERIAL. Changing any other column value will not have any effect on the drawing.

Export

Plant menu: ▶ **Steel** ▶ **Database** ▶ **Export**

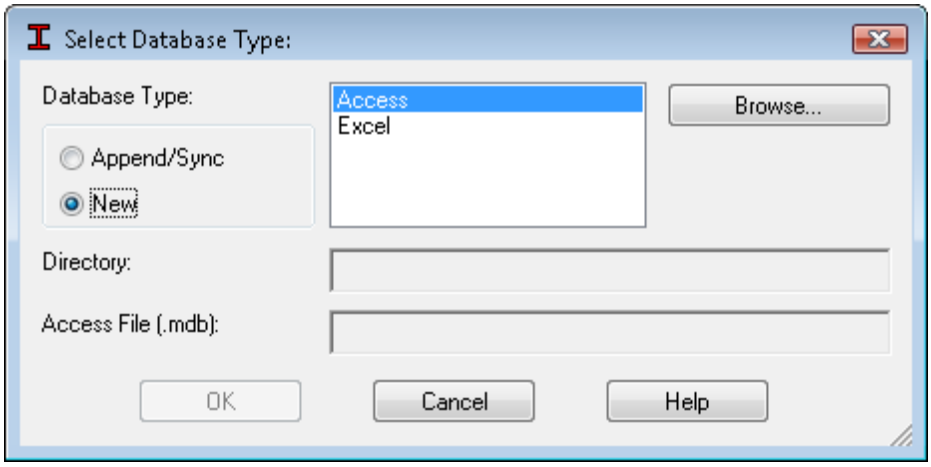
Command line: SDBFGEN

This function can export CADWorx steel information to a Microsoft Access or Microsoft Excel file. After executing the command, selection of components from the drawing is required. After selection, the following dialog will appear requesting the type of database desired.

Note: CADWorx Plant supports only Microsoft Access or Microsoft Excel when using the SDBFGEN command.

The Append/Sync function can be used to maintain a perfect database at all times. When the *Append/Sync* toggle is selected, the exported information will update the database intelligently. Items that have been deleted in the drawing(s) will be deleted from the database. Items that have been changed in the drawing(s) will be changed in the database. Any new items added to the drawing(s), will be added to the database. Use this functionality once a day or five times an hour. This toggle will always maintain a database according to the drawing(s).

Command: **SDBFGEN**



Select components to write to database:
Select objects: *select steel components*
Select objects: *return to finish selection*

The *Append/Sync* toggle appends or synchronizes the drawing and the database file. It will remove entries from the database that no longer exist in the drawing and it updates any components in the database according to the drawing. It will also perform an Database Audit at the end.

Database Type refers to either Microsoft Access, or Microsoft Excel. Select one of these types depending on the database system the user is writing or appending to.

The disabled edit boxes at the bottom of the dialog tell the user the name of database and directory in use. These edit boxes are automatically populated when the *Browse* button is used. If appending, many users can append to the same database at the same time.

The *New* toggle creates a new database based on the chosen directory and database type.

Steel - Import

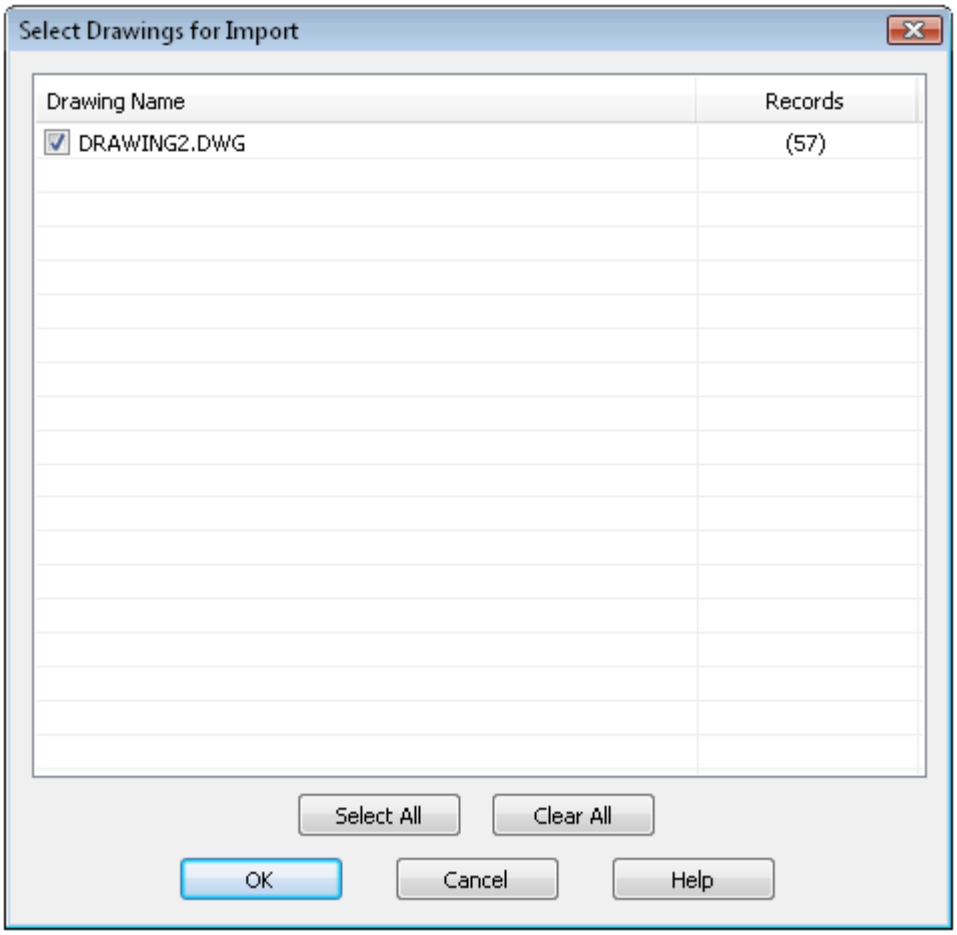
Plant menu: ▶ **Accessory** ▶ **Database** ▶ **Import**

Command line: SDBFIN

This command can import and draw CADWorx Steel components from a Microsoft Access or Microsoft Excel file that stores CADWorx component information. This command is not available when a live database is being used.

Note: CADWorx Plant supports only Microsoft Access or Microsoft Excel when using the SDBFIN command.

When the command is run, a standard file selection dialog appears requesting the user to select the database to import. Once the database is selected, the program presents a list of drawings. The user can select one or more drawings to import. CADWorx Plant then draws the imported components.



Audit

Plant menu: Steel ▶ Database ▶ Audit

Command line: SDBAUDIT

This function is provided as a maintenance solution. Whenever the drawing or the database has any problems, this function can be used to assure that there is an entry in the database for each component which has data attached in the drawing.

It also allows the user to delete problems that are evident in the drawing vs. the database. If a problem is found in the drawing, the following prompt allows the user to add the symbol to the database. Here the function found a component that was not present in the drawing. The user has the option to delete this component in the database or to simply ignore it.

```
Command:  SDBAUDIT
Checking objects in this drawing with rows in the database...
```

Here the function found a row in the database that is not in the drawing. The user has the option to delete this row in the database or to simply ignore it. No would simply ignore it.

```
A row with ID_Count of 100004 is not referenced...
Delete row from database? Yes/<No>:  Y
1 row(s) removed from the database...
```

Bill of Material - Steel

Setup

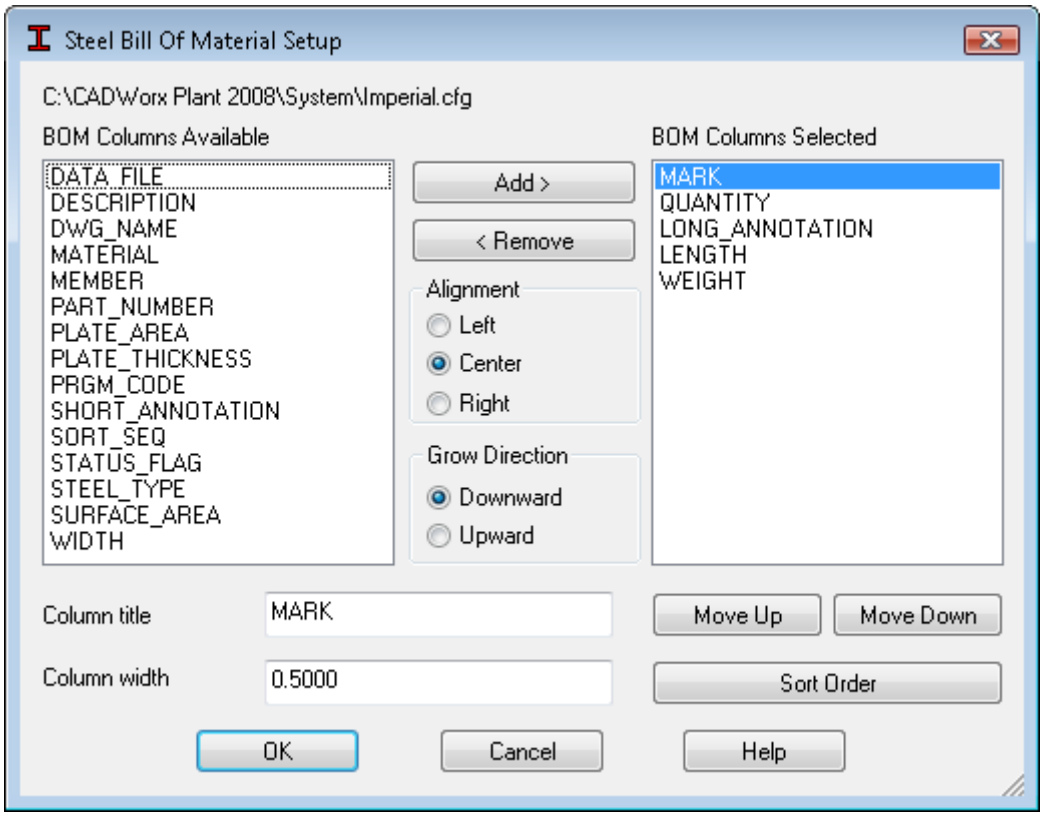
Plant menu: ➤ **Steel** ➤ **Bill of Material** ➤ **Setup**

Command line: SBOMSETUP

This dialog provides the user the ability to customize the Bill of Material schedule to any style or layout desired. This utility can specify column widths, provide the schedule direction (up or down), provide user specified headings for columns, allow alignment of columns and most of all allow the selection of over 15 different entries into the schedule. The columns can be arranged in any order. Select (click on the desired columns) an entry on the left side of the dialog and use the *Add and Remove* button in the middle to add these columns to any Bill of Material schedule.

The template for this information will be stored in the configuration file as listed at the top of the dialog.

Note: Bill of material lengths are based on DIMLFAC, DIMDEC, and DIMLUNIT.



BOM Columns Available

The *Database Columns Available* list box contains all the pieces of information inside of a steel member. These columns can be added to any schedule in any order.

BOM Columns Selected

The *BOM Columns Selected* list box shows all the columns that have been selected and their order of appearance in the schedule from left (top of list box) to right (bottom of list box).

The *Sort Order* (see page 468) button should be used for setting the order by which these columns are sorted.

Add & Remove

The *Add* and *Remove* buttons in the middle of the list boxes copy columns from the columns available list to the column selected list box. These buttons allow individual columns to be placed or the user can select all to be added to the right list box. The facility to remove the columns is also available.

The user can also add a column by double-clicking on the Columns Available list. In addition, they can remove a column by double-clicking on the Columns Selected list.

Alignment & Grow Direction

The *Alignment* toggle specifies alignment for a particular highlighted column. Highlight the column and then select its alignment.

The *Grow Direction* toggle allows the schedule to start at the top and work its way downward or start at the bottom and work its way upward.

Move Up & Move Down

The *Move Up* and *Move Down* buttons allow the user to order the columns highlighted in the right list box. Highlight a column in the right list box and then click the Move Up button (to place it further left) or click on the Move Down button (to place it further right).

Note: Move Up and Move Down do not set the sort order. Click on Sort Order (see page 468) for setting the sort order.

Column Width and Title

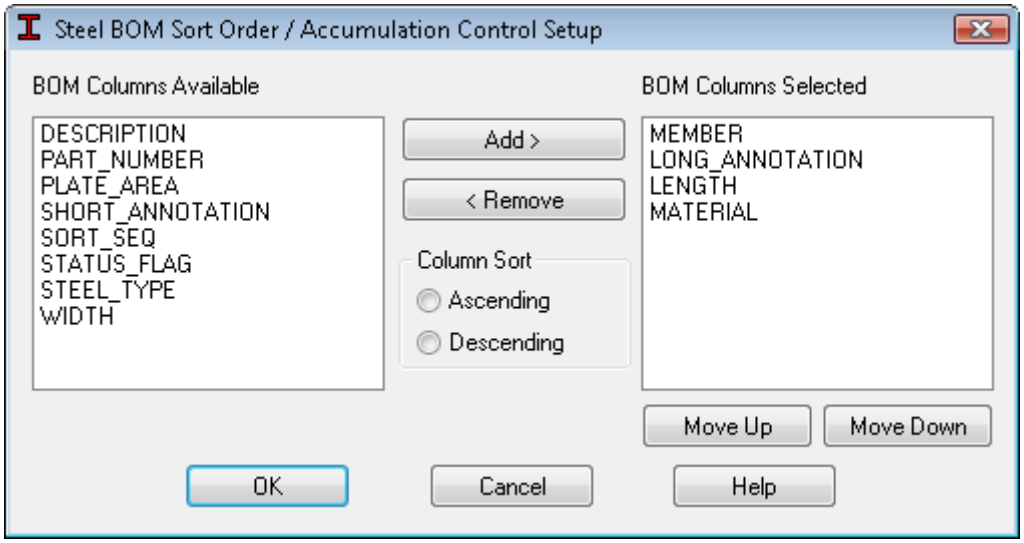
The *Column Title* edit box allows the user to specify a column heading for the highlighted column in the *Column Selected* list box.

The *Column Width* edit box allows the user to specify a column width for the highlighted column in the *Column Selected* list box.

BOM Sort Order

Note: The BOM Sort Order also controls **accumulation** of steel components.

For example: If two members have different Short Annotation and you do not want these to accumulate, Short Annotation would have to be added to the BOM Sort Order.



The list box on the right hand side of the dialog lists all the columns that are selected. The user can select one or more columns to sort by. They can also use the *Move Up* and *Move Down* buttons to change the order in which the BOM is sorted. The Column Sort allows additional sorting on each column. Ascending means sorting from small numbers to larger numbers (Ex: 1 to 100) and from A to Z.

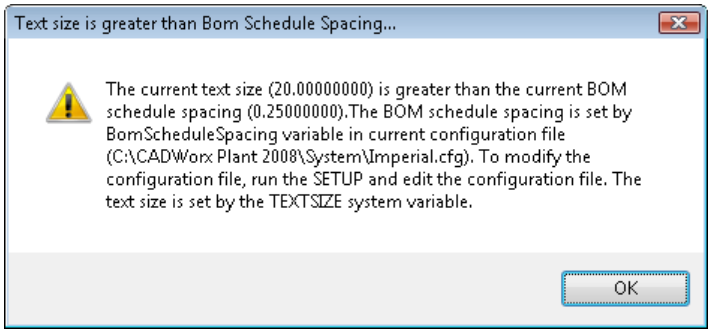
Run

Plant menu: ▶ **Steel** ▶ **Bill of Material** ▶ **Run**

Command line: SBOMCUT, SBOMTOTAL, **or** SBOMSINGLE

This is the main Bill of Material generator. The Total option provides one tag for a total length of pipe in each size. The Cut option provides each piece of pipe with a length and a tag. There are various options that will enhance the Bill of Material generator that will be discussed after each prompt. By selecting Cut or Total from the pulldown, the following prompts will appear. If no components are detected or selected, the generator will halt. The component could have been converted with the Existing option.

The schedule's text size is based on the AutoCAD setvar TEXTSIZE. If this text size is larger than the schedule spacing, the following message will appear and the Bill of Material generation will halt. The schedule spacing is controlled by the BomScheduleSpacing (see page 52) variable in the Configuration file.



Note: A common mistake in AutoCAD is to apply a text height while setting up a text style. If the height is specified in the text style dialog, then there is no other way of controlling it. It will be set to the size in the dialog regardless of what the TEXTSIZE variable is.

CADWorx Plant performs special optimization for automatic tag placement while in paper space only. This optimization will place tags in non-overlapping positions. This optimization is controlled by the BomTagOptimize variable in the current configuration file. For more information regarding this functionality and its controls, please refer to BomTagOptimize (see page 53) variable.

This function is also available as **BOMTOTAL** and **BOMSINGLE**.

Command: **BOMCUT**
Enter tag option [**Counter**/**Manual placement**/**Automatic placement**] <Automatic>:

Counter

This option allows the user to start the tag counter at any positive nonzero integer. Once set, the command resumes requesting the user pick Manual or Automatic placement of tags.

Manual placement

This option inserts tags manually. It prompts the user for the location of the tag and there will be a rubber band line attached to the component extending to the desired tag location. The process will continue until the last component's tag has been placed.

Enter an option [Line number/Select components] <Select components>: *select components or line number*

If the BOM is being generated from paper space, the following prompt would ask whether to turn the snap option on and to change the cursor to an isometric flat view. This works well for placing text, tags, etc. in paper space for isometric models.

Pspace isometric and snap <Yes>: *enter or type* **N** *for no*

This option requires the user to pick a point on the drawing that will be the upper right corner of the users Bill of Material's heading and schedule.

Select upper right corner of BOM heading:

At this point the BOM generator will step through each component selected and prompt for a tag location.

(PIPE, S/40 SMLS, ASTM A-106 GR B) Pick tag location or [Skip]:
(PIPE, S/40 SMLS, ASTM A-106 GR B) Pick tag location or [Skip]:
Etc...

Automatic placement

This option inserts tags automatically. In model space, it places the tag in the direction (tag to component) provided in the next prompt. In paper space, this prompt will not appear if the tag optimization discussed earlier is set on. Please refer to BomTagOptimize (see page 53) for more information.

Enter tag placement angle <45>: *enter angle or enter*

If the user is in paper space, it will switch to the last Mview for component selection. If the components are not in the last Mview, just pick the desired Mview with the arrow indicator that is provided in paper space. Selection can be achieved by using the crossing, window, or individual selection methods.

Select upper right corner of BOM heading: *pick a point*
Enter an option [Line number/Select components] <Select components>:

The user can select components by normal selection methods or select one or more line numbers using the line number option.

Note: Bill of material lengths are based on DIMLFAC, DIMDEC, and DIMLUNIT.

Export

Plant menu: ▶ **Steel** ▶ **Bill of Material** ▶ **Export**

Command line: SBOMEXPORT

This utility exports the steel bill of material to various file formats. This facility will export the steel bill of material to a file that will be based on the columns that have been chosen in SBOMSETUP (see page 466). It will only use the column names. CADWorx Plant can export the bill of material to the following file formats:

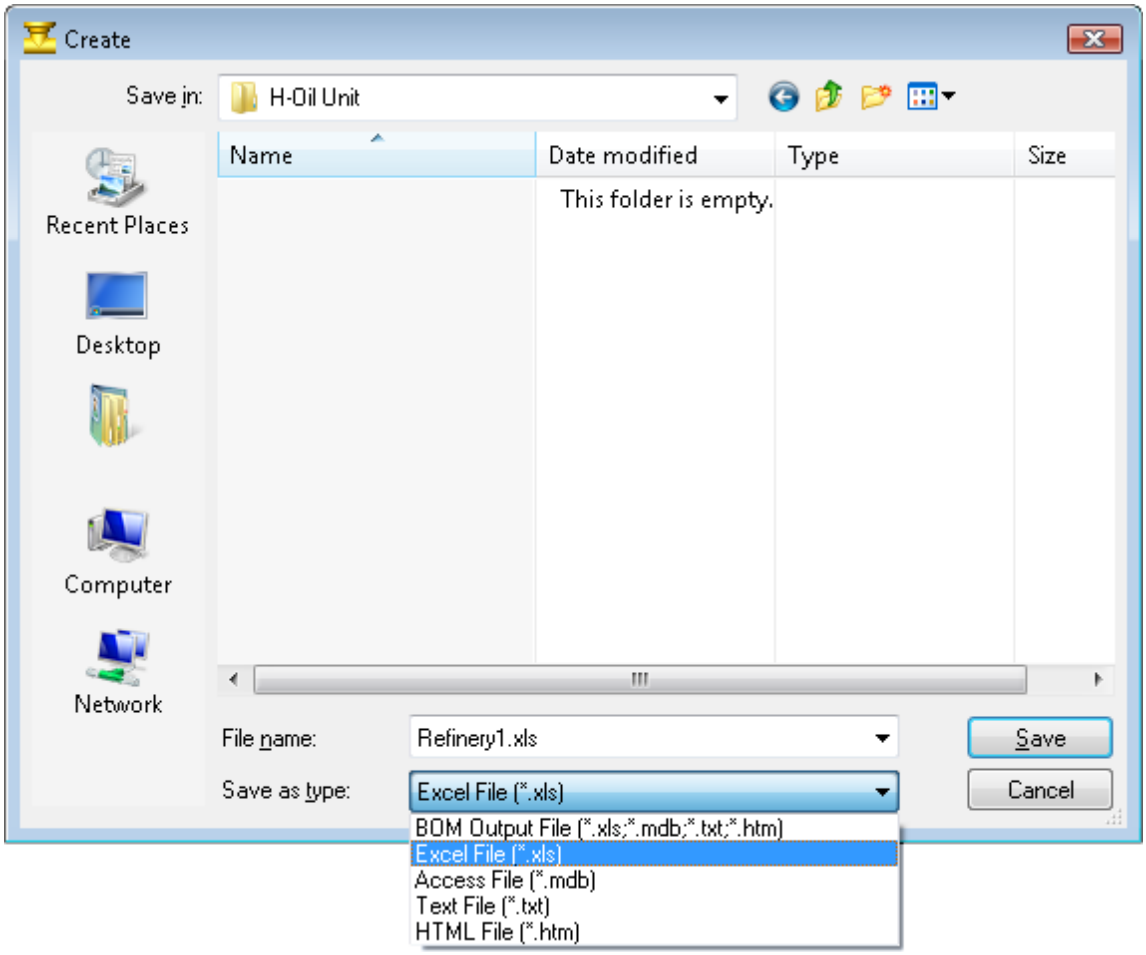
- Access file (*.mdb)
- Excel file (*.xls)
- Text file (*.txt)
- HTML file (*.htm)

Command: **SBOMEXPORT**
Enter an option **[Single item/Total length/Cut length]** <Cut>:

These options are the same as the run options in the Bill of Material generator (see page 469).

Enter an option **[Line number/Select components]** <Select components>: *select component or pick line number*

After the components or line numbers have been selected, the program displays the file selection dialog that allows the name to be specified. At the bottom of the dialog, in the *Save as* type list box, notice the different file types. One of these types should be selected and then a file name is needed.



A final prompt indicates that the file was exported.

BOM output file created: E:\CADWorx Pipe\Refinery1.mdb

Note: When exporting data to Excel, numeric data will be exported as text and will be preceded by a single quote. This single quote will prevent you from totaling the column in Excel. To correct this in Excel, first select the column that needs to be updated. Then, from the Data pulldown menu in Excel, select Text To Columns option. Go through the wizard selecting the "Delimited" radio button and "Tab" as the Delimiter check box. This will cause Excel to change the data from text to numbers. In Imperial/Inch settings, length is exported in feet and inches which cannot be totaled in Excel. To export length in decimal format, change the current AutoCAD Dimension Style settings "Primary Units" format to decimal.

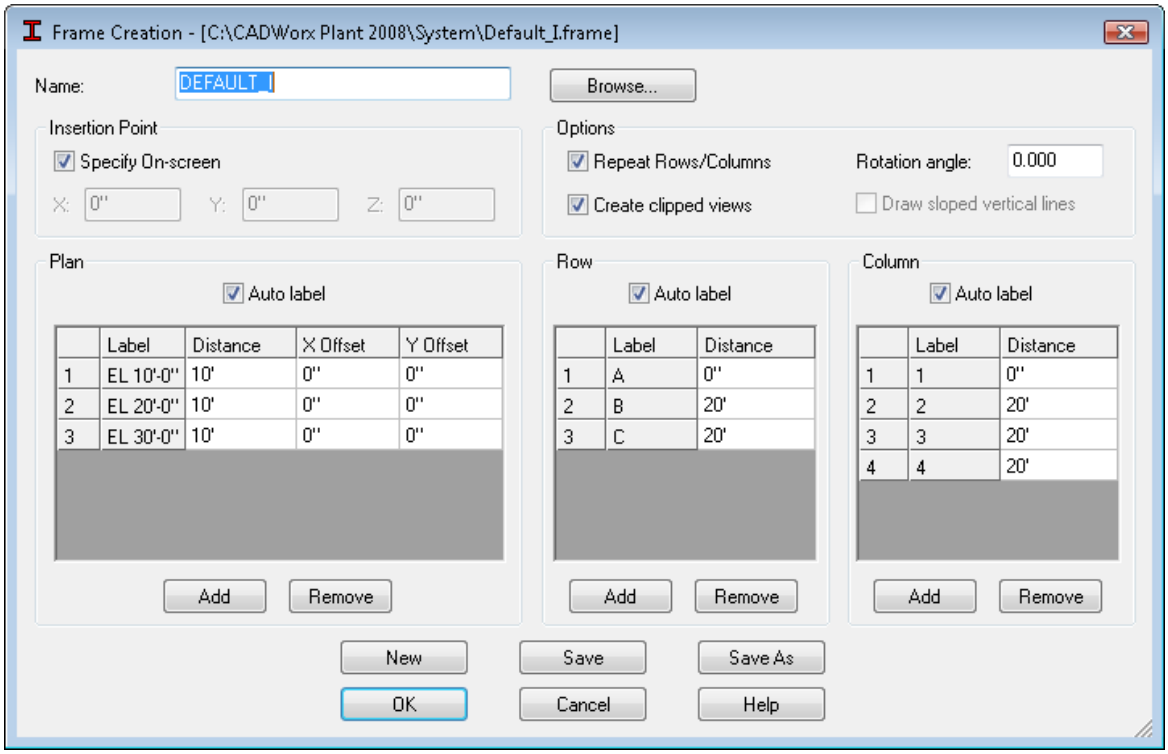
Frame

Frame Creation

Plant menu: ➤ **Steel** ➤ **Frame Creation**

Command line: SFRAME

Steel toolbar:



The frame creation dialog allows the user to create intelligent frames that they can use to build structural steel models. The following sections describe in detail each section of the dialog.

Name

The Name edit box allows the user to specify a name for the current frame. The frame name is required only if the user intends to create more than one frame in the same drawing. If no name is specified and the frame configuration is saved, the program will use the file name as the frame name.

Insertion Point

The Specify On-screen Point toggle allows the user to specify the insertion point of the frame in the dialog or to pick the point on screen. If the toggle is off, the program uses the values from the X, Y and Z edit boxes as the insertion location of the frame. If the toggle is turned on, then the user will be prompted to pick an insertion point to before drawing the frame.

Options

There are four options that the user can specify for each frame. These options are saved with the frame configuration.

Repeat Rows/Columns

This toggle allows the user to repeat the same rows and columns as the first plan for all subsequent plans. This option provides a quick way to create large frames without repeatedly entering the same information.

Create Clipped Views

CADWorx Plant can create clipped views for each plan, row and column. These views can be accessed using the VIEW command in AutoCAD. The views are clipped in the front and back making them excellent to use in layouts for annotation etc. This option will controls whether the views are created or not.

Note: The user will have to check the Update Layer toggle within the View command dialog for the view to be set correctly.

Rotation Angle

This option allows the user to specify a rotation angle for the entire frame. The entire frame will be rotated by the angle specified around the insertion point specified.

Draw Sloped Vertical Lines

This option will make the vertical lines sloped according to each plans X and Y offsets. It is useful while creating platform frames.

Note: This option requires that each plan have the same number of rows and columns.

Plans

This section allows the user to create different plans for the current frame. The user can specify the distance from one plan to the next in the distance column for each plan. They can also specify X and Y offset values. These offset values are included to the insertion point before the row and column lines for this plan are created.

By default, CADWorx Plant automatically creates labels for each plan based on the distance specified. The elevation label can be changed from the default setting by editing the <INSTALL_DIR>\System\FrameSetup.tbl file and changing the PLAN_LABEL_START setting. The user can turn off this feature using the Auto label toggle. Turning the system off will allow the user type in any text they desire as the label for the plan. This label is saved with the frame once it is drawn and will be available for the frame annotation command.

The Add button will add a new plan to the current frame. The Remove button will remove the selected plan(s) from the current plan. If the Repeat Rows/Columns toggle is off, the remove button will remove all rows and columns associated with the plan that is being removed.

Rows

This section allows the user to create different rows for the current frame. The user can specify the distance from one row to the next in the distance column for each row. If the Repeat Rows/Columns toggle is turned ON, the rows are for the currently selected plan.

By default, CADWorx Plant automatically creates labels for each row. The row label can be changed from the default setting by editing the <INSTALL_DIR>\System\FrameSetup.tbl file. The [ROW] section contains the characters used in making the label. If there are more rows and available characters, they will be concatenated. For example, A, B, C.....X, Y, Z, AA, AB, AC and so on. The user can turn off this feature using the Auto label toggle. Turning the system off will allow the user type in any text they desire as the label for the row. This label is saved with the frame once it is drawn and will be available for the frame annotation command.

Columns

This section allows the user to create different columns for the current frame. The user can specify the distance from one column to the next in the distance column for each column If the Repeat Rows/Columns toggle is turned ON, the columns are for the currently selected plan.

By default, CADWorx Plant automatically creates labels for each column. The column label can be changed from the default setting by editing the <INSTALL_DIR>\System\FrameSetup.tbl file. The [COLUMN] section contains the characters used in making the label. If there are more columns than available characters, they will be concatenated. For example, A, B, C.....X, Y, Z, AA, AB, AC and so on. The user can turn off this feature using the Auto label toggle. Turning the system off will allow the user type in any text they desire as the label for the column. This label is saved with the frame once it is drawn and will be available for the frame annotation command.

New

The new button allows the user to create a new frame. If changes are detected to the current frame, the program will prompt to use to save changes.

Save

The save button allows the user to save the current frame to a file. If no name is specified, the user will be prompted to enter a name and specify the location to save the frame configuration.

Save As

The save as button allows the user to save the current frame under a different file name. A standard file dialog is displayed for the user to enter a file name and specify the location to save the file.

OK

The OK button allows the user to draw the frame in the current drawing. If changes are detected to the current frame, the user will be prompted to save the changes to the frame configuration file.

Frame Annotation

Plant menu: ▶ **Steel** ▶ **Frame Annotation**

Command line: SFRAMEANOT

Steel toolbar:

This command allows the user to annotate and dimension the frame lines.

Command: **SFRAMANOT**
Select frame line to dimension/annotate:
Enter an option **[Dimensions only/Call outs only/Both]** <Both>:

Dimensions only

This option draws only the dimension for the selected frame line. The user will be prompted to pick the dimension side. The distance the dimensions are offset from the frame line is controlled by the AutoCAD setvar DIMDLI. This option respects the current DIMSCALE setting.

Pick dimension side: *pick a direction*

Call outs only

This option creates the call out and places the label of the selected frame line in a block with attribute. The M5 block located in the <INSTALL_DIR>\Support directory is used for the call out. The user will be prompted to pick the call out location.

Pick call out location: *pick a location*

Both

This option creates both dimensions and call outs.

This command can also create the dimensions in call outs in paper space.

Existing

Command line: SSETEXIST / SREMOVEEXIST

These commands change the existing flag for steel members and allow for layer changes to be made to the steel solid and center line.

Command line: SLAYERCHANGE

This command allows for layer changes to be made to the steel solid and center line.

Running these command, will present the user with the following options:

Select components [**All/Select**] <Select>:

Select

This option allows the user to manually select steel members.

All

This option will automatically select all steel members.

After either of the two options above, the user will have the option to change the solid's layer.

Change the solid layer [**Yes/No**] <No>:

No

This option will not change the solid layer of the selected components.

Yes

This option will invoke a dialog from which the user can select a desired solid layer.

After either of the two options above, the user will have the option to change the centerline's layer.

Change the centerline layer [**Yes/No**] <No>:

No

This option will not change the centerline layer of the selected components.

Yes

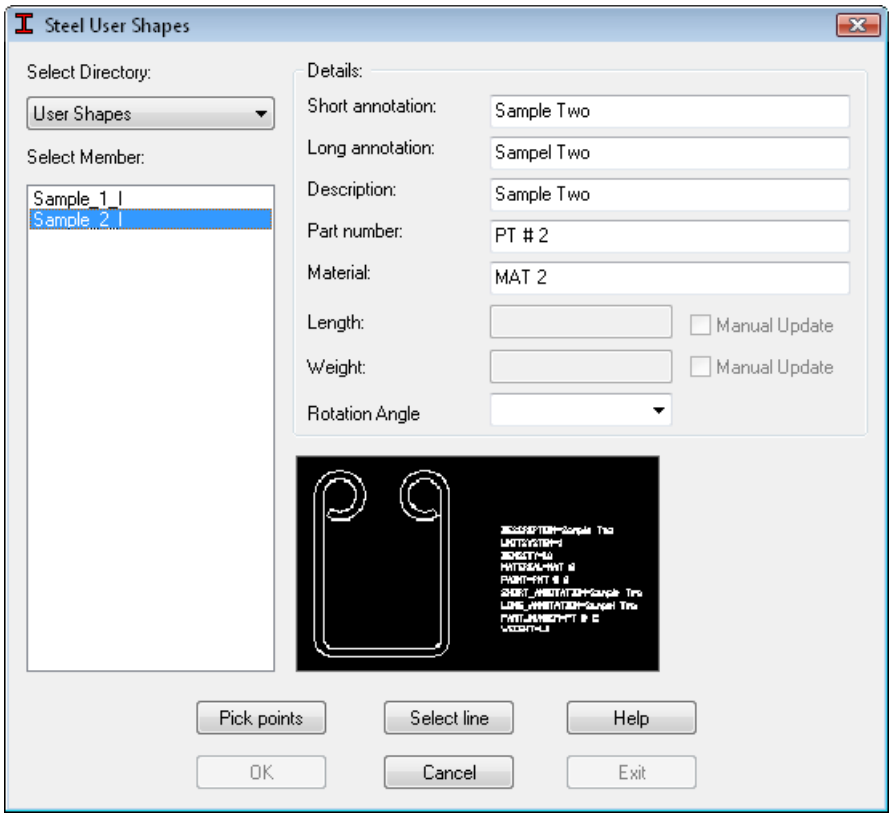
This option will invoke a dialog from which the user can select a desired centerline layer.

Note: Steel plates/baseplates will automatically have their centerline's layer changed to whichever layer was chosen for their corresponding solid.

User

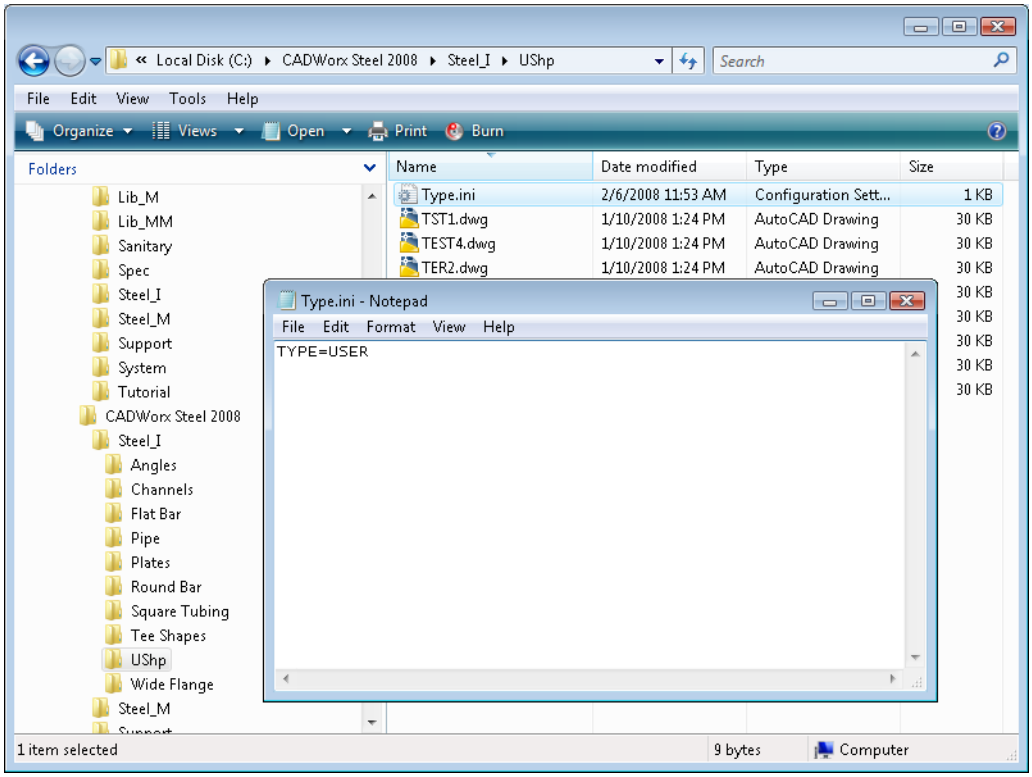
Command line: SUSER

This command allows the user to insert a custom user shape using the dialog below.

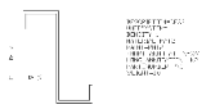


In this dialog the user can select the component to draw based on the user shape DWG file that they created. The extrusion direction and distance is based on points picked or line selected. Rotation angle may be selected from the pull-down, and annotations may also be entered within this dialog.

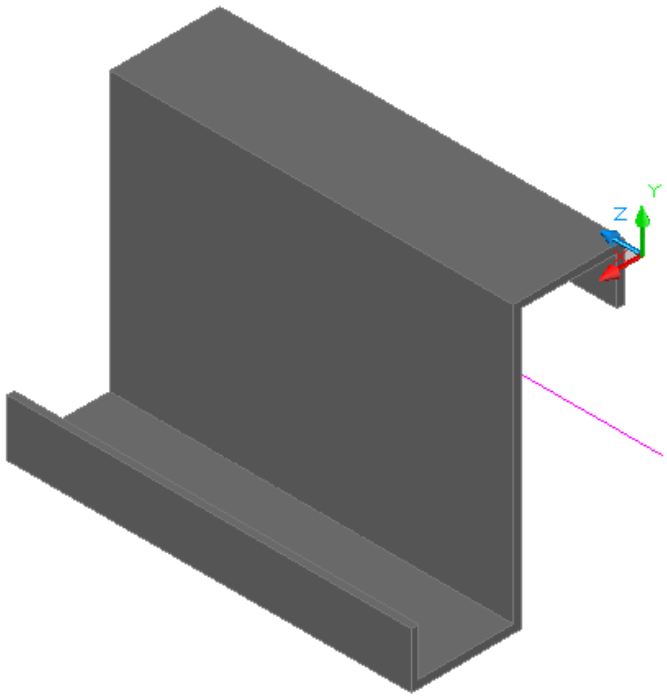
For user shapes to work the Type.ini file must have TYPE=USER. In the example below, the directory (...CADWorx Steel\Steel_I\UShp) was used to store the user shape drawing files.



Shown below are the parameters stored in a user created DWG file that can be used to create a roof purlin user shape.



After selecting the shape from the dialog, the user may pick points, or select a line to determine the extrusion length. See figure below.



Loft

Command line: SLOFT

This commands allows the user to create lofted steel components.

Command: **SLOFT**
Specify density: *enter real*
Select cross sections in lofting order: *select cross sections*

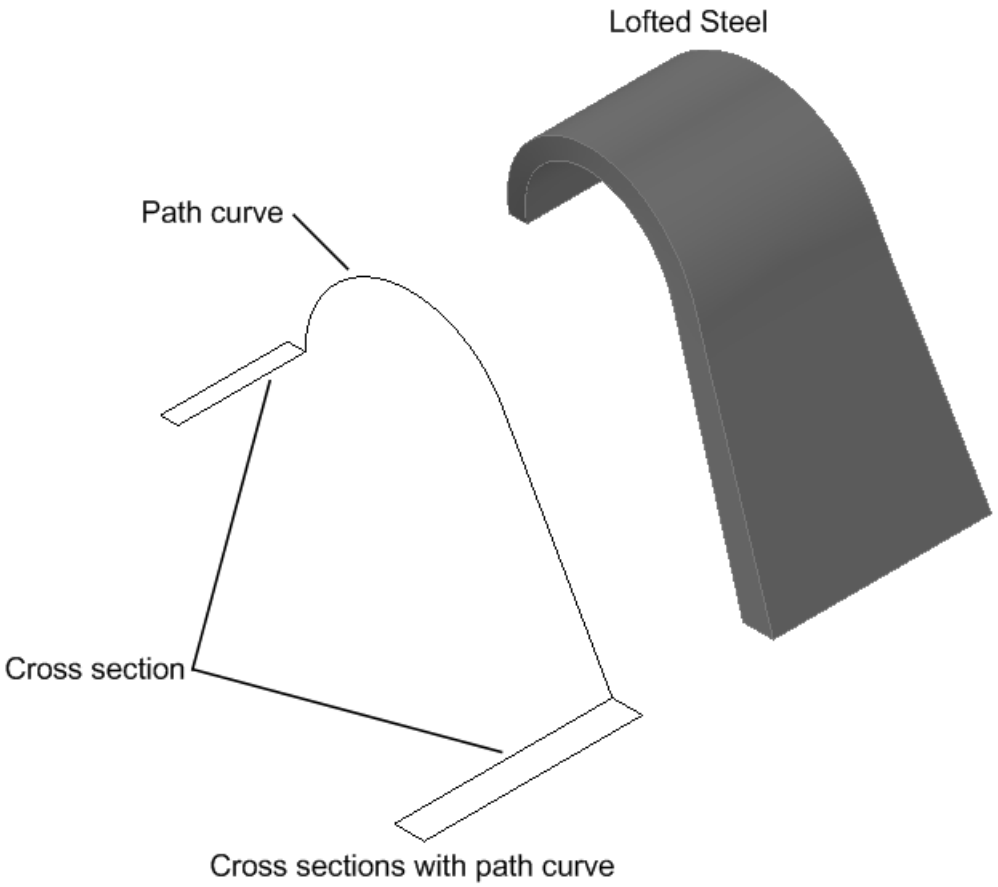
The cross sections define the shape of the resulting steel component. You must specify at least two cross sections.

Select path curve: *select a path curve*

The path curve must intersect all planes of the cross sections.

Enter short annotation: *enter string*
Enter long annotation: *enter string*
Enter description: *enter string*
Enter part number: *enter string*
Enter material: *enter string*

Any of these entries can be left blank if the user presses Escape or Enter at their prompt.



Steel - Export / Import

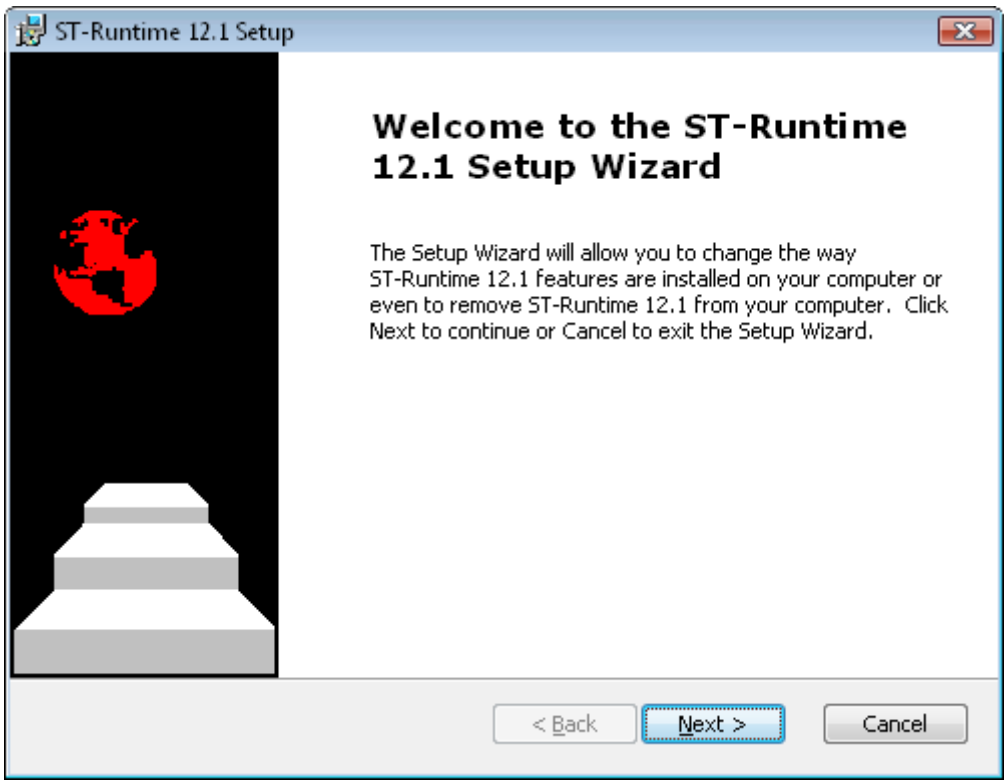
Plant menu: ▶ **Steel** ▶ **Export or Import**

Command line: SEXPORT or SIMPORT

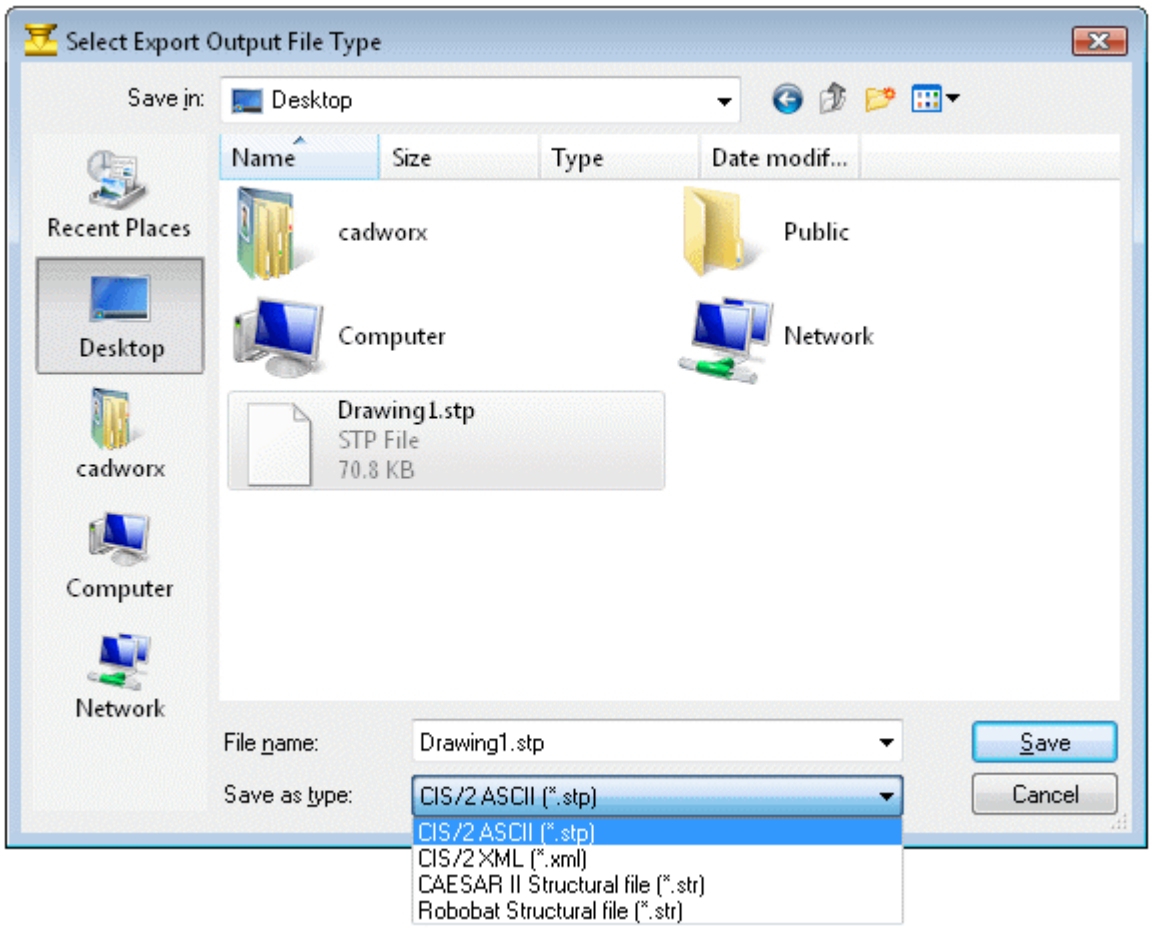
The SEXPORT command can export the CADWorx Steel components to three data format: CIS/2, CAESAR II, and RoboBat

The SIMPORT command can import from a CIS/2 data file and create CADWorx Steel components. The import requires that the CADWorx Steel parametric data file be available for the components that are being imported from the CIS/2 file.

The export or import to CIS/2 requires installing the "ST-Runtime". The install for this utility is located at: ..\CADWorx\System\struntime.msi

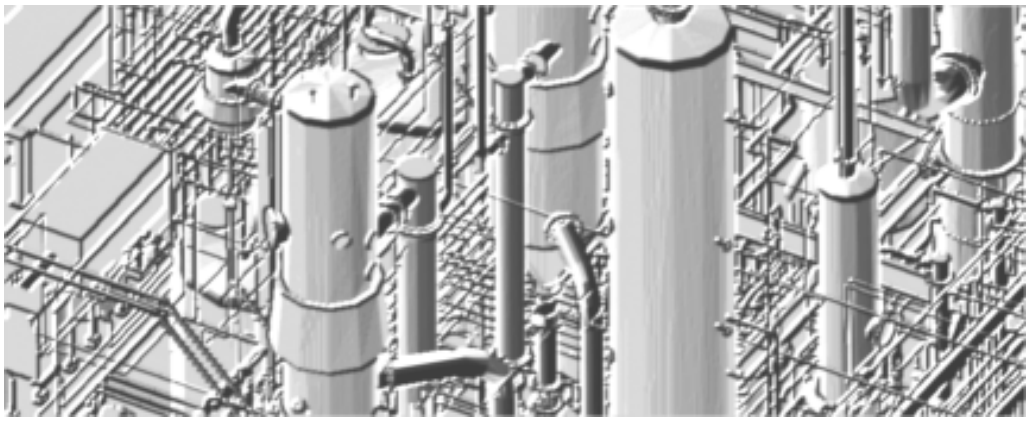


To change the type of file exported with the SEXPORT command, change the "Save as type" at the bottom of the dialog.



CHAPTER 10

HVAC/Cable Tray



In This Chapter

HVAC 488
Edit 490
Bill of Material - HVAC 491

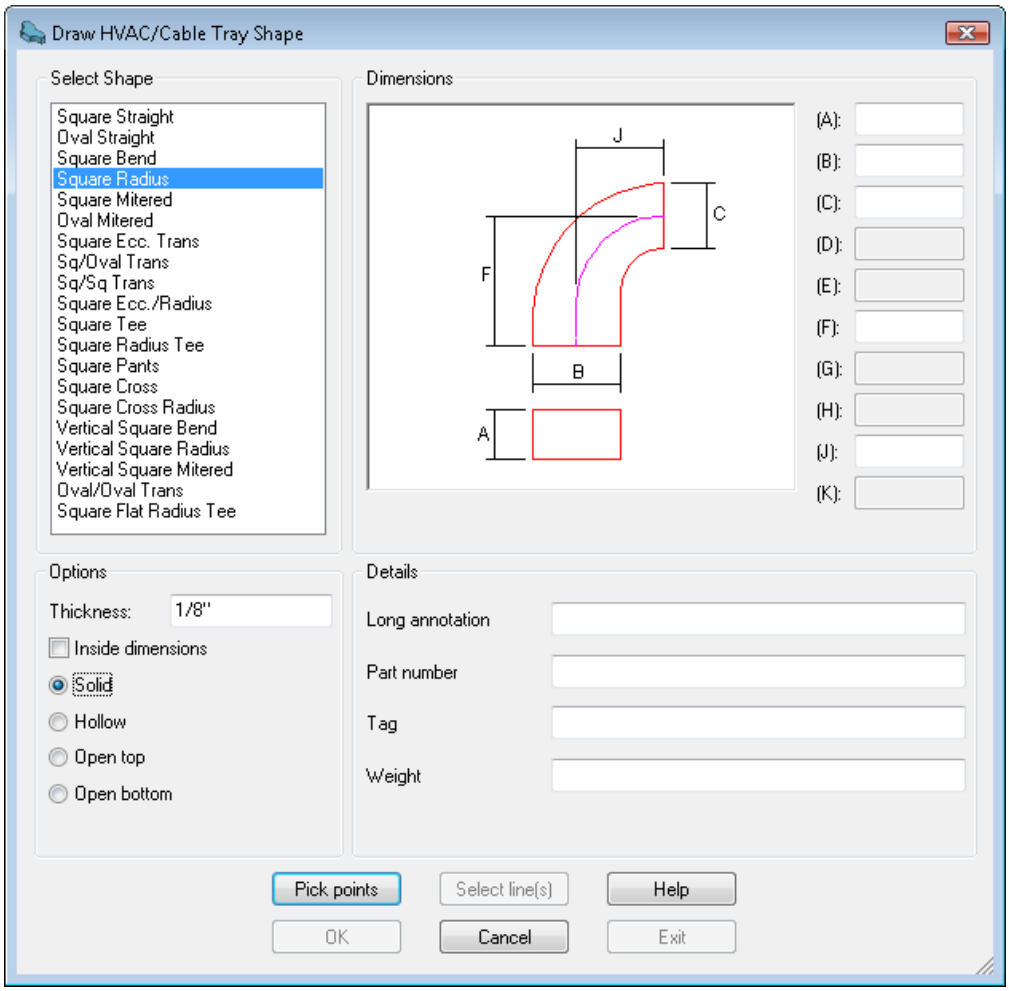
HVAC

Plant menu: ▶ **HVAC**

Command line: HVAC

This dialog allows the user to call up different HVAC and cable tray shapes from a data file library.

Double clicking on an HVAC shape in the drawing will also bring up this dialog and allow modification of the options, details and parameters.



Select Shape

The list displays all available HVAC shapes. Selecting one activates the correct dimension boxes on the right side of the dialog.

Dimensions

The preview of the shape shows a list of all the required dimensions to draw the shape selected. The user can enter dimensions in the current units they set for the drawing.

Details

Enter long annotation, part number, tag number, and weight for the shape being drawn. There is no automatic procedure for creating any of these entries such as steel shape or plates.

Options

The Options group is provided to assist the user in further modifying the shape.

The check box *Inside dimensions* is used to specify the exact dimensions to which the HVAC shape will be drawn. If this box has been checked, then the shape will be drawn with the thickness added to all the necessary dimensions that the user entered in the edit boxes. If the box is left unchecked then the dimensions of the shape drawn will not include the value of the thickness.

Thickness value is required when the Inside dimensions box is checked, or if the HVAC shape is to be drawn as a Hollow or Open top.

Solid, a radio button that when selected draws the HVAC shape as a Solid shape. This is the default setting.

Hollow, a radio button that when selected draws the HVAC shape as a Hollow shape.

Open top, a radio button that when selected draws the HVAC shape as a Open top shape. None of the Oval shapes can be drawn as Open top.

Open bottom, a radio button that when selected draws the HVAC shape as a Open bottom shape. None of the Oval shapes can be drawn as Open bottom.

Note: Creating a large number of HVAC shapes with the hollow or open top options in a single drawing can significantly increase the drawing size.

Edit

Plant menu: ▶ **HVAC** ▶ **Edit**

Command line: HCEDIT

This command allows the editing of the HVAC or cable tray shapes. This command can be invoked from the command line, pulldown menu, or simply by double clicking on the shape. The command will display the exact same dialog that was used to place the shape. The available shapes will be grayed out on the left side. Changing any dimension is easily accomplished by simply changing the value on the right side. See HVAC/Cable Tray Shapes command above for more information.

Bill of Material - HVAC

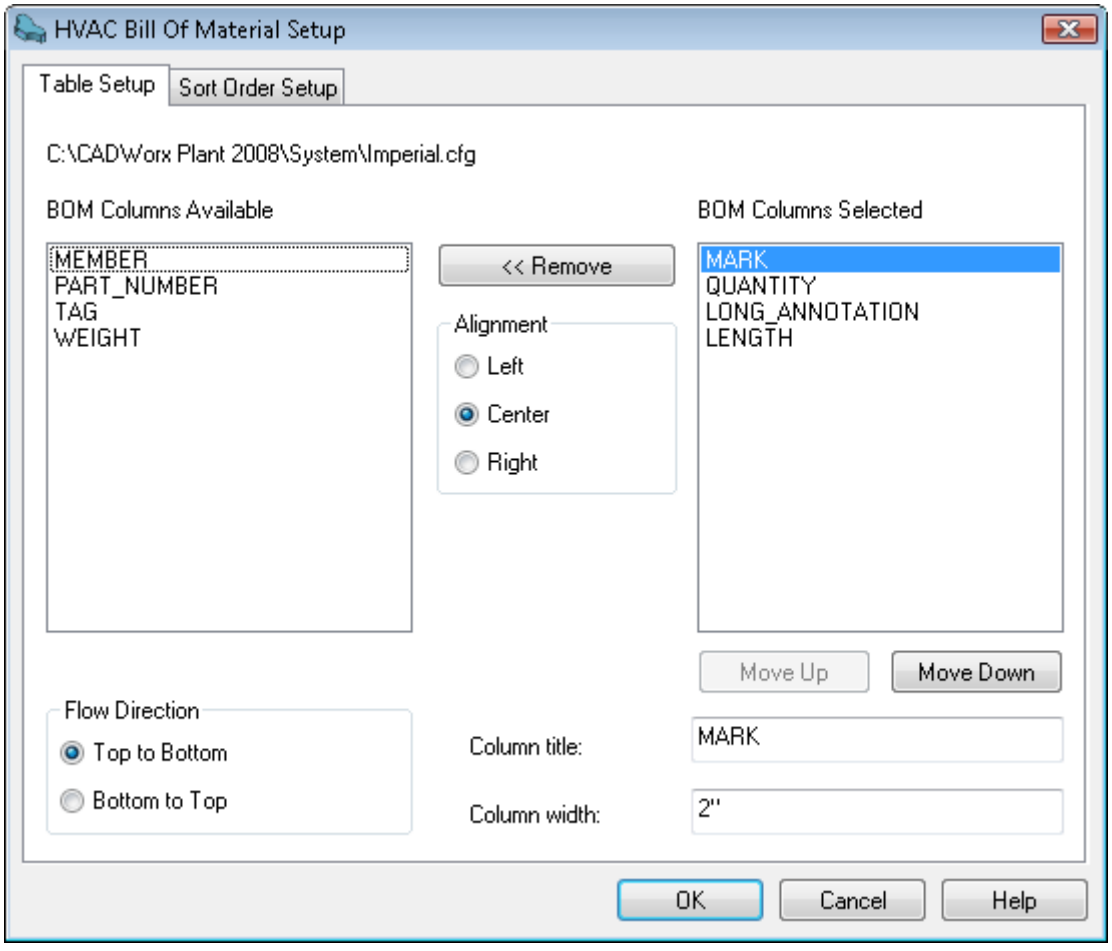
Setup

Plant menu: ▶ **HVAC** ▶ **Bill of Material** ▶ **Setup**

Command line: HBOMSETUP

This dialog provides the user with the ability to customize the Bill of Material schedule to any style or layout desired. This utility can specify column widths, provide the schedule flow direction (top to bottom or bottom to top), provide user specified headings for columns, and allow the user to specify the alignment of column data. The columns can be arranged in any order. By selecting (click on the desired columns) an entry and using the *Add / Remove* button, the user can decide which columns will be displayed in the Bill of Material schedule.

The template for this information will be stored in the configuration file as specified at the top of the dialog.



HVAC - BOM Columns Available

The *Columns Available* list box contains column data available for generating a bill of material that has not yet been selected by the user to become part of the bill of material.

HVAC - BOM Columns Selected

The *Columns Selected* list box shows all the columns that have been selected and their order of appearance in the schedule from left (top of list box) to right (bottom of list box).

The dialog's *Sort Order* tab should be used for setting the order by which these columns are sorted.

HVAC - BOM Add / Remove

Upon the selection of an entry in the *Columns Available* list box, the *Add* button is activated. By clicking this button individual columns can be moved to the *Columns Selected* list box.

Upon the selection of an entry in the *Columns Selected* list box, the *Remove* button is activated. By clicking this button individual columns can be moved to the *Columns Available* list box.

HVAC - BOM Alignment & Flow Direction

The *Alignment* toggle specifies the data alignment for a particular selected column. Simply select the column and then select its alignment.

The *Flow Direction* toggle allows the schedule to start at the top and work its way downward or start at the bottom and work its way upward.

HVAC - BOM Move Up & Move Down

The *Move Up* and *Move Down* buttons allow the user to order the columns selected in the *Columns Available* list box. Simply select a column in the *Columns Available* list box and then click the Move Up button (to place it further left) or click on the Move Down button (to place it further right).

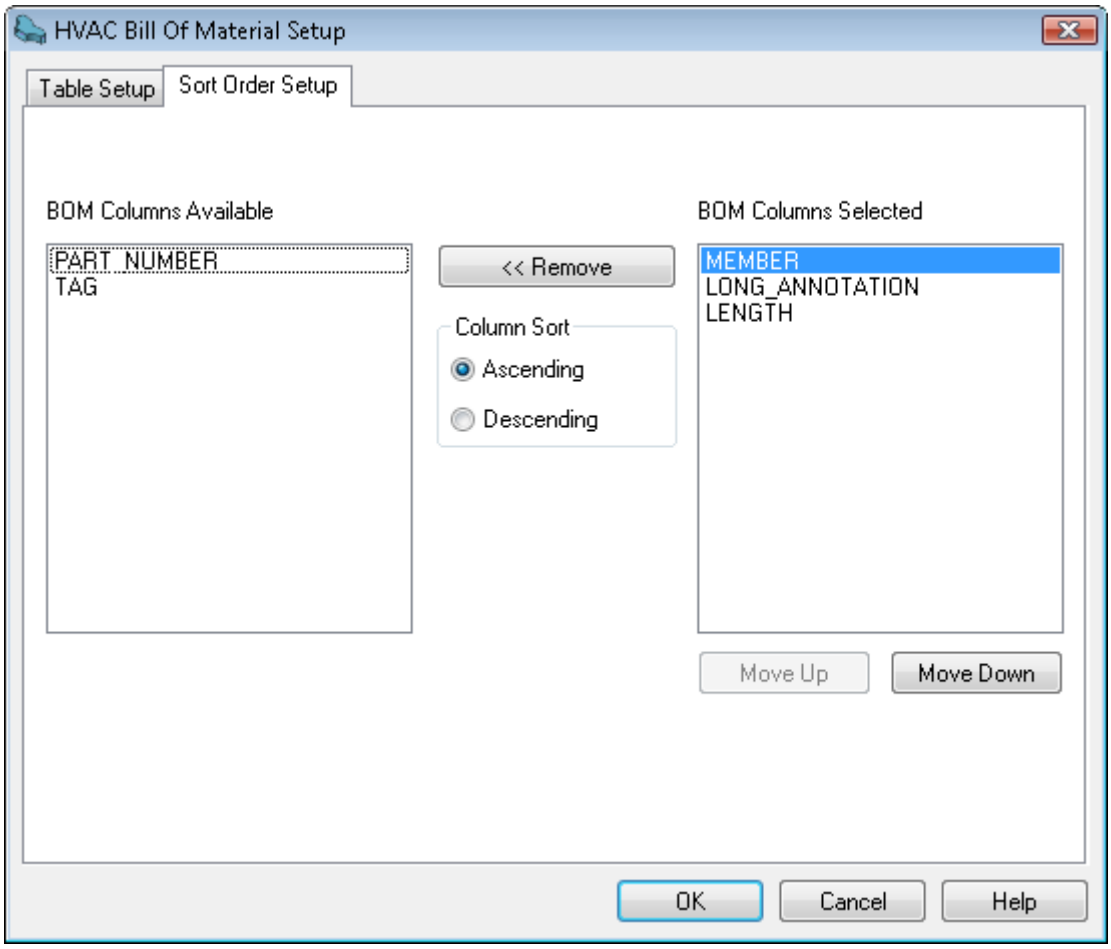
Note: Move Up and Move Down operations do not affect the sort order. To specify the sort order, click on the *Sort Order* tab.

HVAC - BOM Column Width and Title

The *Column title* edit box allows the user to specify a column heading for the selected column in the *Columns Selected* list box.

The *Column width* edit box allows the user to specify a column width for the selected column in the *Columns Selected* list box.

HVAC - BOM Sort Order



The list box on the right hand side of the dialog lists all the columns that are selected. The user can select one or more columns to sort by. They can also use the *Move Up* and *Move Down* buttons to change the order in which the BOM is sorted. The Column Sort allows additional sorting on each column. Ascending means sorting from small numbers to larger numbers (Ex: 1 to 100) and from A to Z.

HVAC Accumulation:

The HVAC BOM Sort Order does NOT control **accumulation** of components. Accumulation is determined based on the following HVAC shape parameters:

Shape Name: Square straight, Oval straight, etc.

A through K parametric parameters: User specified.

Inside dimensions: On or Off.

Thickness: User specified.

Draw mode: Solid, Hollow, Open Top, Open Bottom.

Long annotation: User specified.

Part number: User specified.

Tag: User specified.

Weight: Only applicable for a single or cut bill of material generation.

Length: Only applicable for a single or cut bill of material generation.

Run

Plant menu: ▶ **HVAC** ▶ **Bill of Material** ▶ **Run**

Command line: HBOMCUT, HBOMTOTAL, or HBOMSINGLE

This is the main Bill of Material generator.The Total option provides one tag for a total length of straight HVAC components in each size. The Cut option provides each HVAC component with a length and a tag. There are various options that will modify the output of the Bill of Material generator that will be discussed after each prompt. If no HVAC components are detected or selected, the generator will halt.

This following functionality is also available for **HBOMTOTAL** and **HBOMSINGLE**.

Command: **HBOMCUT**
Enter tag option [**Counter**/**Manual placement**/**Automatic placement**] <Automatic>:

Counter

This option allows the user to start the tag counter from any positive non-zero integer. Once set, the command resumes requesting the user pick Manual or Automatic placement of tags.

Manual placement

This option inserts tags manually. It prompts the user for the location of the tag and there will be a rubber band line attached to the component extending to the desired tag location. The process will continue until the last component's tag has been placed.

The following option requires the user to pick a point on the drawing that will be the upper right corner of the users Bill of Material's heading and schedule.

Select upper right corner of BOM heading:

Automatic placement

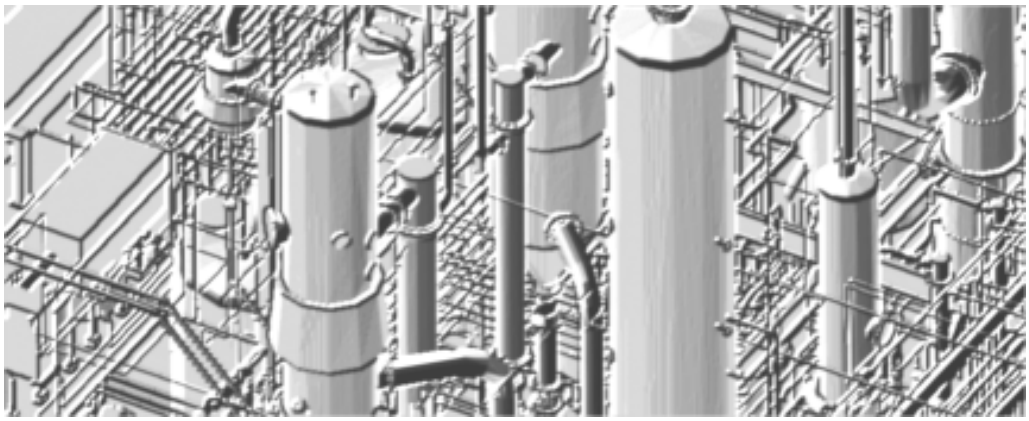
The following option inserts tags automatically. In model space, it places the tag in the direction (tag to component) provided in the next prompt. In paper space, this prompt will not appear.

Enter tag placement angle <45>: *enter angle or enter*

If the user is in paper space, the Bill of Material generator will switch to the last Mview for component selection.

CHAPTER 11

Text



In This Chapter

Annotate 497
ISO Text 499
Height 500

Annotate

These functions are used to annotate the short description, long description, or the elevation of any CADWorx Plant component.

Elevation

Plant menu: ▶ **Text** ▶ **Annotate** ▶ **Elevation**

Command line: ANOTELEV

Command: **ANOTELEV**
Select component to annotate elevation: *select component*

Enter an option [**BOP/TOP/Centerline**] <Centerline>:

BOP

Use this option to label the bottom of the pipe elevation.

TOP

Use this option to label the top of the pipe elevation.

Centerline

Use this option to label the centerline elevation.

This routine provides a rubber band to the component selected for annotation. It will not provide a leader line from the label to the component.

Start point or [**Leader/Justify**]:

Justify

This option allows five different justifications similar to the AutoCAD standards. For more information please refer to the AutoCAD documentation.

Leader

This option provides the user the ability to add a leader to the text being placed.

Start point

Pick starting point for text.

The final prompt will ask for a rotation. This can be entered from the keyboard or by picking two points.

Rotation angle <0>: *enter angle, pick two points*

Component

Plant menu: ▶ Text ▶ Annotate ▶ Component

Command line: ANOTCOMP

This is similar to the annotation function above.

Command: **ANOTCOMP**
Annotation type [**Short/Long/Tag**] <Short>:

Tag

This option will annotate the component with its tag.

Long

This option will annotate the component with its long description and append the size to the front.

Short

This option will annotate the component with its short description and append the size to the front.

The user is next prompted to select the component to annotate the description.

Select component to annotate description:

For both options the following prompts will follow.

Start point or [Leader/Justify]: *see component elevation described earlier for details*

The final prompt will ask for a rotation. This can be entered from the keyboard or by picking two points.

Rotation angle <0>: *enter angle, pick two points*

ISO Text

Plant menu: ▶ Text ▶ ISO Text

These functions are provided for use in paper space when working with isometrics. It provides for 30 and 330 oblique angled text. These functions work similar to the AutoCAD DTEXT command.

Height

Plant menu: ▶ **Text** ▶ **Height**

Command line: TEXTHEIGHT

This function changes the height of selected text by matching it with other text. This is useful when the user is zoomed to an unknown factor in paper space.

Command: **TEXTHEIGHT**
Change text height in [**Entire drawing**/**Layer**/**Selection**]:

Entire drawing

This option automatically chooses all text on the drawing.

Layer

This allows the user to enter a layer for selection. CADWorx Plant's default text layer is "TEXT".

Enter name of layer: *enter layer name*

Selection

This option allows the user to select individual text.

Select text to change height...
Select objects: *select text*

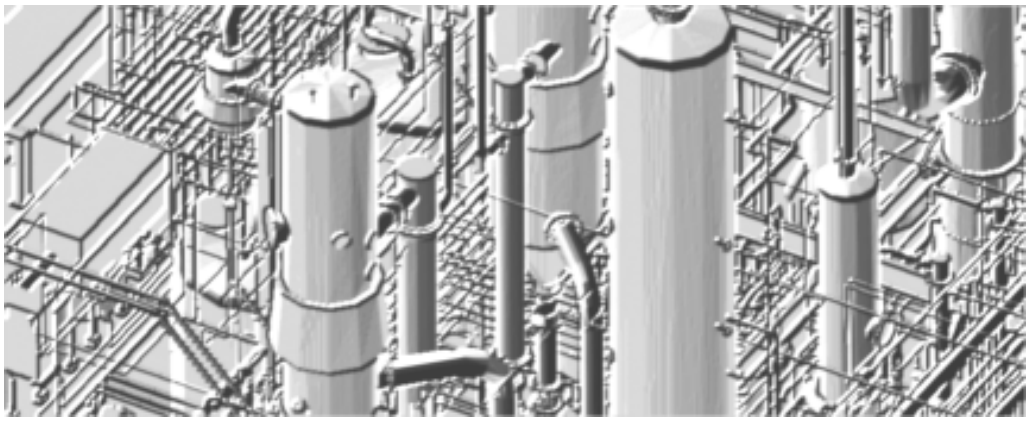
Select objects: *enter to complete selection set*

After each option above, the user will be asked to select the text which is going to determine the desired text size.

Select text determine height: *pick the text with correct height*

CHAPTER 12

Dimension



In This Chapter

Coordinates 502
Automatic 503
Multiple 504
Tailed 504

Coordinates

Plant menu: ▶ **Dimension** ▶ **Coordinates**

Command line: DIMCOORD

This routine places coordinates on the user's drawing in either a user coordinate system or a world coordinate system. To end this sequence hit enter or space bar. TEXTSIZE is controlled by DIMSCALE and DIMTXT. This routine allows the user to place the text at the coordinate with or without a leader.

Command: **DIMCOORD**
Change user coordinate system to world [Yes/No] <Yes>: *hit enter or type* **Y** *to change to world*

Enter to end or pick extension line point: *pick point*
Enter an option [**Leader/Other point/Same point/Centerline**] <Leader>:

Leader

This option allows for placement of a leader.

To point: *pick intermediate point of leader*
Direction for centered text: *pick direction*

Other point

This option allows for an alternate location for the text to be located without a leader.

Pick other point: *pick alternate location*
Direction for centered text: *pick direction*

Same point

This option places the dimension text at the same location which is picked without a leader.

Direction for centered text: *pick direction*

Centerline

This option places the dimension text either with a north or east designation. It will label a vertical dimension as east and a horizontal dimension as north.

Direction for centered text: *pick direction*

Enter to end or pick extension line point: *pick point*

Note: The following AutoCAD setvars affect the dimensioning routines; DIMALT, DIMALTD, DIMALTF, DIMASZ, DIMBLK, DIMEXE, DIMEXO, DIMLFAC, DIMTXT, DIMZIN, DIMSCALE, LUNITS, LUPREC and UNITMODE.

Automatic

Plant menu: ▶ **Dimension** ▶ **Automatic**

Command line: AUTODIM

This function provides dimensions automatically for a selected set of components. The options to set the offset distance and whether to include flanges are also provided.

Command: **AUTODIM**
Enter an option [**Flange**/**Offset**/**Selection**] <Selection>:

Flange

This option provides flange face dimensioning.

Flange dimensioning [**On/Off**] <Off>:

On

Set on to dimension to the flange faces in the selection set.

Off

Set off to avoid dimensioning to the flange faces.

Offset

This option specifies the distance from the dimension line to the component.

Enter offset distance: *enter a real*

Selection

This option starts the selection process.

Select objects: *enter when finished*

The next prompt asks for the centroid point of dimensions. An example would be two pieces of pipe perfectly parallel with elbows attached to each end. Picking between the two pieces of pipe would place the dimension lines to the outside.

Pick centroid point of dimensions: *pick point*
Auto Dimensioning...finished

If the above was tried on just two pieces of pipe parallel with no elbows, then an error code would appear. Combined components must have at least one turn.

Multiple

Plant menu: ▶ Dimension ▶ Multiple

Command line: DIMH **(**DIMV **and** DIMR**)**

This routine allows the user to pick multiple origin points to be dimensioned. The user will be prompted for the first, second, third, etc., point to be dimensioned. To end the sequence the user will need to type a "D" for dimension line as shown below.

Command: **DIMH** (**DIMV** and **DIMR**)
First extension line origin: *pick point*
Second extension line origin: *pick point*
[Dimension location]/<3th extension line origin>: **D** *(to end sequence)*
Dimension line location: *pick point*
Dimension text <12'-1 1/2">: *hit enter or enter new value for the 1st two points*
Dimension text <10'-5">: *hit enter or enter new value for the last two points*

If Auto Rotated was chosen, one additional prompt would have appeared in the beginning.

Dimension line angle <0.00>: *pick to points or enter degree (ex: 30)*

Tailed

Plant menu: ▶ Dimension ▶ Tailed

Command line: DIMTAILH **(**DIMTAILV**)**

This routine allows the user to place tailed dimensions on objects from a user selected reference point. To end this sequence hit enter or space bar.

Command: **DIMTAILH** (**DIMTAILV**)
Pick reference point: *pick point, this will be asked the first time this routine is executed*
Enter to end or pick extension line point: *pick origin point or hit enter to end*
Dimension text <1'-8 3/4">: *hit enter or enter new value*

Tailed dimension location: *pick point*
Enter to end or pick extension line point:

Enter to end or pick extension line point: *hit enter to end*

By executing the command a second time, the user will be given the option to re-select the reference point.

Reference point [New/Previous] <Previous>:

New

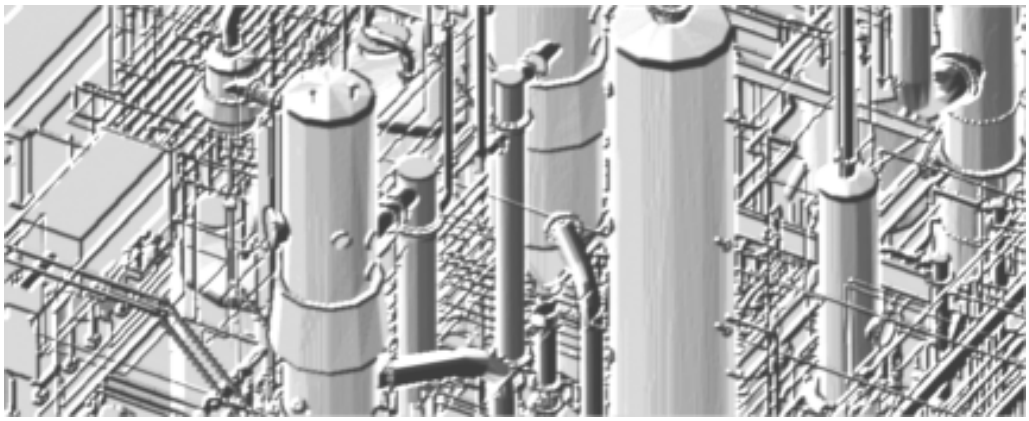
This function will allow the user to re-select the reference point.

Select reference point: *pick new point*

Previous ***hit enter to use previous reference point***

CHAPTER 13

UCS



In This Chapter

Point and Shoot UCS 506
UCS Next 509
UCS Object 509
Compass 510

Point and Shoot UCS

Plant menu: ▶ **UCS**

Command line: VIEWFLAT **(also** VIEWNORTH**,** VIEWSOUTH**,** VIEWEAST**, and** VIEWWEST**)**

UCS toolbar: , , , , **and**

One of the reasons AutoCAD is such a powerful program is that it allows the user to manipulate the X, Y, Z, coordinate system, UCS (user coordinate system). This is the heart of CADWorx Plant's system for drawing in 3D. To utilize this program to its' potential the user needs to be familiar and comfortable with the Point and Shoot system of manipulating the UCS. When the user becomes accustomed to it they will find it quite simple.

UCSICON options needs to be On (visible) and set to Origin. This allows the user to see exactly where the icon is at all times. Although it isn't necessary, it is highly recommended that it be somewhere on the centerline of the components the user is drawing. This will keep everything in the same plane. If the user has to turn into another plane, use UCS NEXT or choose the appropriate plane (North, South, East, West, or Flat) and place the icon on the centerline of the new plane. If the user cannot see the rubber band effect, or if it doesn't appear correct, they are out of plane. The should pay attention to the icon as it shows where they are. If the broken pencil icon appears, and there are strange lines on the screen instead of pipe and components, the user's current plane is perpendicular to the viewpoint. Try an Undo or change the viewpoint. When dimensioning or annotating an isometric or 3D model, the user should place the UCS icon on a node or a centerline on the run they are working on. Text will have the same orientation as the icon.

The five planes CADWorx Plant has provided (Flat, North, South, East, and West) can be accessed from the pulldown under UCS, or from the keyboard by simply typing VIEWFLAT (alias VF), VIEWNORTH (alias VN), etc. and hitting enter or the space bar.

Some people find the UCS icon distracting and keep it off, however, when drawing in 3D it is recommended that the user turn it on - at least until they become accustomed to using UCS.

Command: **VIEWFLAT** (or **VIEWNORTH**, **VIEWSOUTH**, **VIEWEAST**, and **VIEWWEST**)
Xtext/Ztext/Elevation/Rotate/COordinate/CLip/<Pick point, or enter>:

Xtext

When viewing a model from the bottom side, using Vpoint with a negative angle from the X-Y plane, text and dimension placement will be upside down and backwards. This option allows the rotation of the X axis of the UCS to remedy this. Text or dimensions will then be placed in the proper orientation for viewing and plotting. If changing from a negative to positive angle from the X-Y Vpoint, the text will have to be replaced for the proper orientation. This option will rotate the UCS 180 degrees along the X-axis.

Ztext

When viewing a model from the back side, using Vpoint with a 0 to 180 degree angle in the X-Y plane from the X axis, text and dimension placement will be upside down and backwards. This option allows the rotation of the Z axis of the UCS to remedy this. Text or dimensions will be placed in the proper orientation for viewing and plotting. If changing to a Vpoint rear view position (180 - 360 degree angle in the X-Y plane from the X axis), the text will have to be replaced for the proper orientation. This option will rotate the UCS 180 degrees along the Z axis.

Elevation

This option is for setting the elevation to present UCS. This command does not work for setting depth when in a North, South, East or West plane, instead, by setting an elevation of 10' (when the present elevation is 0') will raise the UCS up to 10' regardless of the position.

Enter desired elevation <*current elevation*>: *enter elevation*

Rotate

This option is for rotating the present UCS around the X, Y, and Z axis. For example: set to 45 degrees after inserting a 45 degree elbow allows placement of other components in line with the 45 degree elbow.

X/Y/Z rotation angle: *enter angle or option*

All these options allow the UCS to be rotated around whichever axis is selected here.

COordinate

This option allows entry of an exact X,Y,Z coordinate in relation to the WCS (world coordinate system).

Enter desired coordinate: *enter coordinate (x,y,z)*

CLip

This option allows view clipping to be performed at the present plane set by the UCS. This plane can be established with the Point and Shoot UCS options or any three point UCS position. The plane can be skewed at any angle.

Enter an option [**Front/Back/Off/On**] <On>: *choose one of the option or enter*

On

This option turns view clipping on with clipping distances as set by the Back and Front options below.

Off

This option turns view clipping off.

Back

This option sets the distance to the back of the clipping area. It must be a negative number. If this number is positive, it could clip past the front clipping area, therefore leaving nothing in the view.

Enter negative distance to back clipping plane <-5'-0">:

Front

This option sets the distance to the front of the clipping area. It must be a positive number. If this number is negative, it could clip past the back clipping area, therefore leaving nothing in the view.

Enter positive distance to front clipping plane <5'-0">:

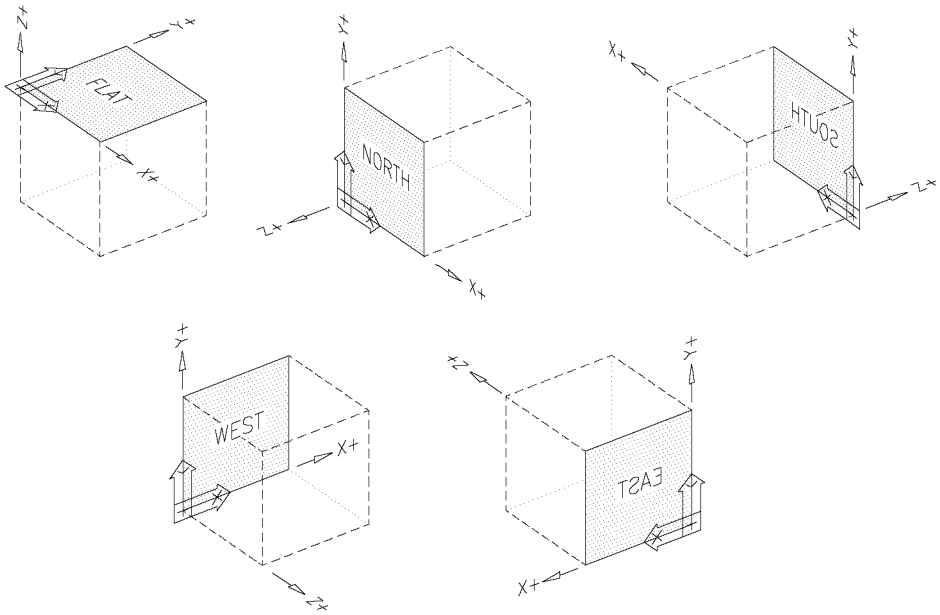
Note: This option *CLip* is also available as the command VIEWCLIP.

Pick point

By picking a point, the UCS will move the 0,0,0 user base to the picked location. By hitting enter, the UCS will remain at the same origin and switch to the new plane. For example: Hit enter to change from north to a west plane.

Other UCS positions required by the user not supplied with CADWorx Plant's point and shoot routine will have to be entered with AutoCAD's UCS command. Two extremely useful options in this command are the OBject and 3 Point. We advise leaving the UCSICON on and set to origin when using any method of re-arranging the UCS. This is accomplished with the UCSICON command.

Note: If a point is picked close to the screens edge and the UCS icon is on and set to origin, the icon might not remain at this point because AutoCAD requires that the whole icon be visible. It will place the icon in the lower left corner of the screen. If this happens, zoom out a little or pan toward the new UCS position.



UCS Next

Plant menu: ▶ **UCS** ▶ **UCS Next**

Command line: UCSNEXT

UCS toolbar: 

This command toggles through each of the five Point and Shoot UCS positions provided above.

UCS Object

Plant menu: ▶ **UCS** ▶ **UCS Object**

Command line: UCS

UCS toolbar: 

This command provides the Object option of the UCS command. Please refer to the AutoCAD documentation on the UCS (User Coordinate System) for more detail. The command allows the user to select an object in the drawing to orient the UCS.

Command: **UCS**
Enter an option [New/Move/orthoGraphic/Prev/Restore/Save/Del/Apply/?/World] <World>: **OB**
Select object to align UCS: *select an object or a dimension*

This command is very useful for rearranging dimensions in the model space (MSPACE) of a CADWorx Plant isometric. Run this command and then select the dimension. It places the UCS according to the position of the dimension allowing it to be stretch and moved.

Compass

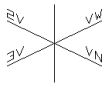
Plant menu: ▶ **UCS** ▶ **Compass**

Command line: COMPASS

UCS toolbar:

This command provides a compass type preview of the desired Point and Shoot orientation. This is not really a true compass but does provide the user with the exact plane with which to choose from. A view such as VIEWNORTH or VIEWSOUTH will be oriented in the East - West direction. This indicates a view north elevation (or plane) as looking in a north direction.

By selecting this command, it allows the selection of a point and then shows a compass with different planes available within the Point and Shoot system. The preview will show 4 different planes with which to choose from. The VN will indicate a view north. Opposite to this view will be the VIEWSOUTH indicator. The VIEWWEST and VIEWEASE will also be available. The VIEWFLAT is also available but is not shown.



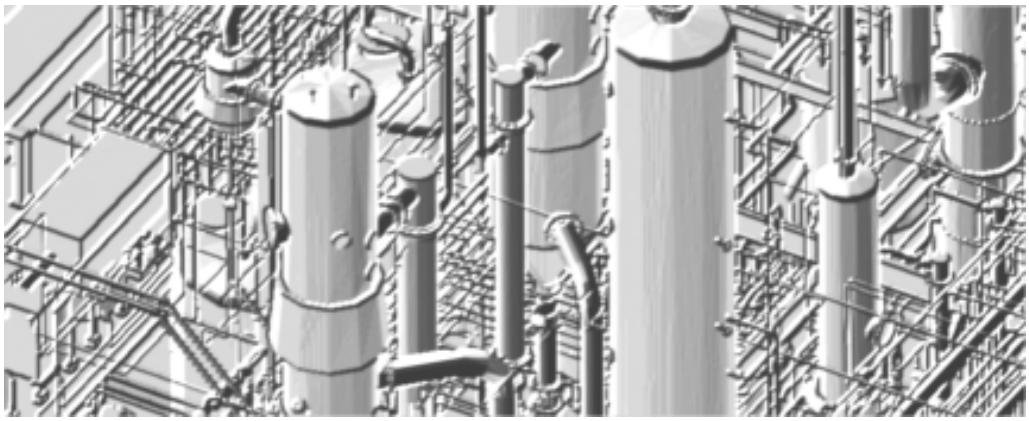
The two obvious Point and Shoot orientations shown above would be VIEWNORTH and VIEWWEST. This allows proper placement of text and other objects. It also provides the user with a positive X direction as normal instead of it being backwards. As the display's orientation is changed, the compass changes accordingly.

Command: **COMPASS**
Origin point <0,0,0>: *Pick a desired point for the point and shoot UCS*
Select view...
<Flat>/North/South/East/West: *Select N, S, E, W or enter for F*

Note: The alias VF, VN, VS, VE, and VW are also available options of the command which are not shown.

CHAPTER 14

Tutorial



In This Chapter

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Tutorial Notes

In AutoCAD and CADWorx Plant, help is provided on-line by pressing F1 while the menu item is highlighted. Also, holding the pick arrow over the desired toolbar button will show the function name or purpose and provide a descriptive sentence on the lower status bar. If the AutoCAD setvar TOOLTIPS is set to 1, CADWorx Plant will display a descriptive tool tip box over a component while the cursor is located there.

Throughout this tutorial, it is assumed that the working directory will be the <INSTALL-DIR>\TUTORIAL. If the CAESAR II importing and exporting exercise is performed, then it would be useful to direct or operate CAESAR II in this directory as well.

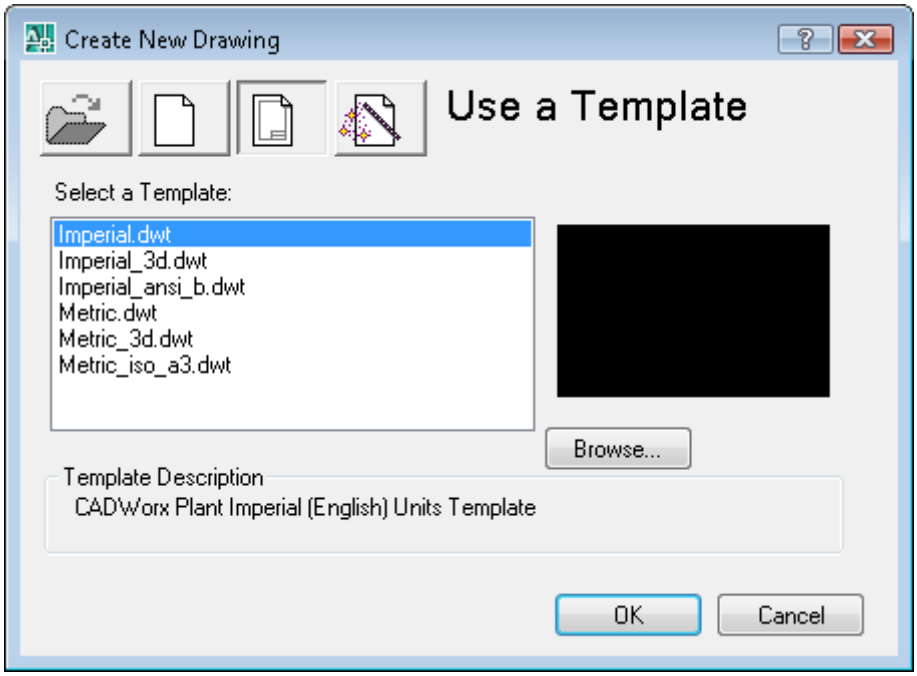
Please make backup copies of the tutorial drawings. They will be used several times within different sections of the tutorial.

This tutorial can be performed in either metric or English units. All commands have the English responses with metric in parenthesis. For imperial, a user would enter “4’ ” and for metric the entry would be “1220”. For example:

Length: **4' (1220)**

All references to the CADWorx Plant pulldown menus are like Plant>Setup using the greater than symbol. Shown with all references to the pulldowns are command line entries throughout this tutorial. These commands are just for reference. Aliases for all these commands are located in the file <INSTALL-DIR>\SUPPORT\Pipe_alt.pgp. The install will copy these aliases into the ACAD.PGP located in the AutoCAD support directory. It is the responsibility of the user to make sure there is no duplication of existing aliases in the ACAD.PGP. Use any standard text editor such as Notepad to modify any of the aliases in the ACAD.PGP.

When starting a new drawing, it is very important to use a template. CADWorx Plant supplies six different templates which have been specifically designed to fit the user needs. Unless you are a very experience AutoCAD user, and you know exactly what template AutoCAD is going to use, we would suggest that you configured AutoCAD to use the tranditional start up dialog. This can be done with the OPTIONS command using the *System* tab under *General options*. This dialog is shown below.



A good working knowledge of AutoCAD is required to perform this tutorial. This tutorial will only address CADWorx Plant commands. For help on AutoCAD commands, please use the AutoCAD on-line help and on-line tutorials.

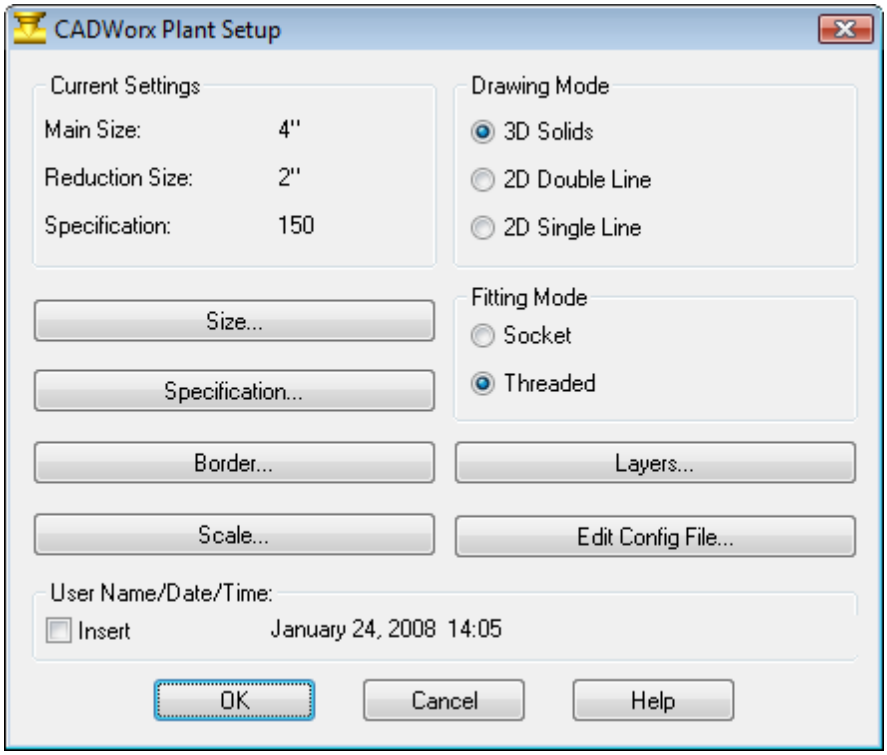
We recommend 3 lines at the command prompt for working with this tutorial.

Setup

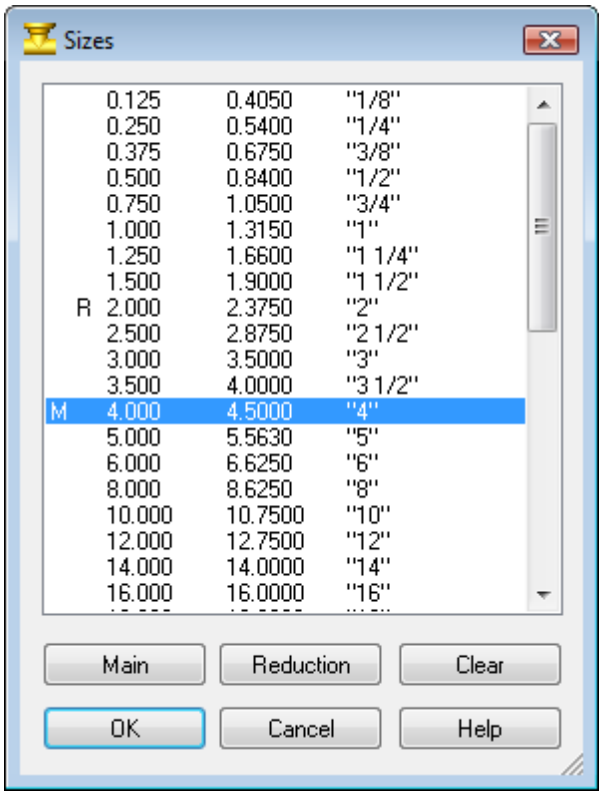
The first time CADWorx Plant is started, the user is required to provide startup defaults that will be used each time a new drawing is started. These defaults are outlined under Configuration (see page 48). The dialog box provides many user customizable settings and layer controls for working with CADWorx Plant.

This tutorial explains how to start CADWorx Plant and draw some sample piping. From the pulldown menu, select Plant>Setup to display the following dialog. The user can also type the command at the AutoCAD prompt as shown.

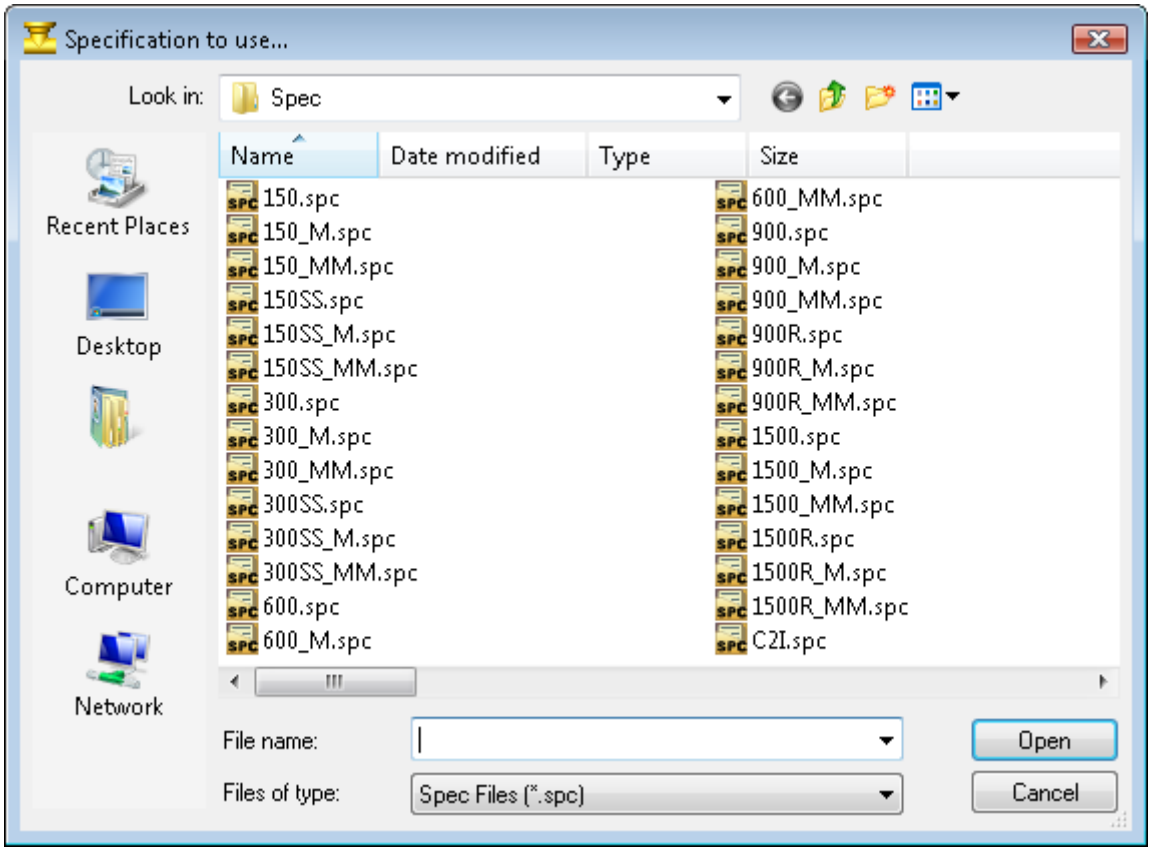
Command: **SETUP**



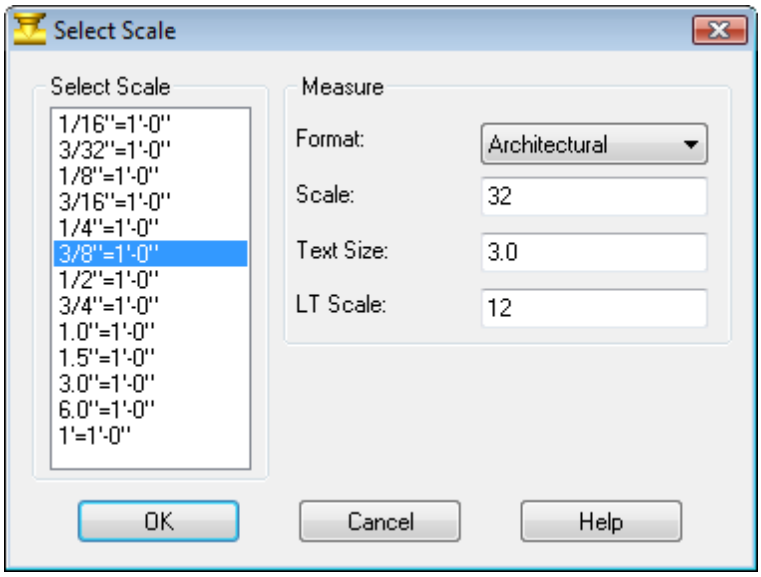
Here the pipe sizes can be set to 4” and 2” and the specification can be set to 150 (Use 150_M for metric measure). Setting these sizes can be accomplished by picking the *Size* button below the *Current Settings* box. When this button is chosen, a list is presented allowing the size to be selected. Pick the 4” for the main size and 2” for the reduction size. Double click for setting the main size and then double clicking on another size will make it the reducing. The buttons at the bottom can also be used after highlighting a size.



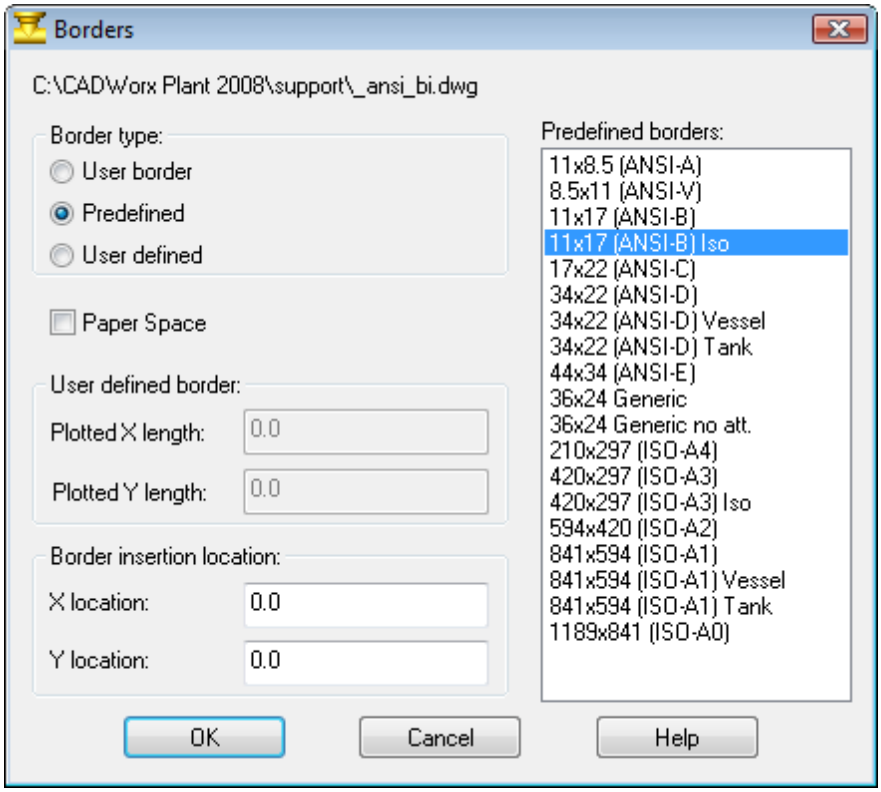
The *Specification* button presents a standard dialog for selecting a file (specification). Double click 150.SPC (150_M.SPC) or highlight it and then pick *Open*.



The following dialog box can be accessed from the main Setup dialog box using the *Scale* button. The user can select preset scales from the list box on the left or enter different text sizes, scales, etc. on the right. Setting the scale is the same as setting DIMSCALE in AutoCAD. This functionality is primarily used for model space development only. If working in paper space, DIMSCALE should be left at 0.



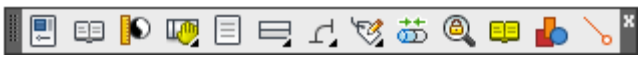
Using the *Borders* button on the main setup dialog displays this dialog. Pick the *Predefined* button in the upper left. The borders in the dialog are located in the MVSetup.dfs in the <INSTALL-DIR>\SUPPORT directory and are easily changed by the user with any text editor such as Notepad. Pick the 11x17 ANSI_B (420x297 ISO_A3) and use the *Paper Space* toggle. Select OK on all remaining dialogs. The scales listing in the Scales dialog above would then be disabled. This is due to using paper space instead of the conventional model space. If the border was placed in model space, the scales would not be disabled and would require that a scale was chosen.



Upon exiting all dialogs, the border will present an attributes dialog (only if the border has attributes associated with it (CADWorx Plant borders contain attributes). Enter titles for the border (if desired – use DDATTE to edit later) and select OK on this dialog. There should now be a border with a triangular icon in the lower left corner. Later, this border will be used with a simple isometric that will be drawn in the following sections of this tutorial.

Note: Setting the drawing up in the model space environment with scales could have also been accomplished. This would have provided a traditional model space environment with the proper AutoCAD setvar DIMCALE set. This would not be useful with a CADWorx Plant isometric. Isometrics in CADWorx Plant require paper space.

From the pulldown menu, select Plant>Toolbars>Settings. This toolbar is more often used to set the size and specification. The specification is the 2nd button and the size is the 3rd button from the left.



All of these commands could have been entered at the command line. As shown above SETUP (see page 18) is available from the command line. The main size could have been set also at the command line

```
Command: MAIN
Nominal main size <>: 6
```

The reduction size could have been set the same way.

```
Command: RED
Nominal reduction size <>: 2
```

The specification is set the same way.

```
Command: SPEC
Specification <>: 150  enter the spec or enter ~ which will show the user the dialogs above


Prepared <INSTALL-DIR>\SPEC\150.spc specification...
```

The tilde “~” could have been used at all of these prompts. This would have presented the dialogs as if Setup was running.

Point and Shoot UCS

CADWorx Plant provides enhanced UCS (see page 506) (user coordinate system) control. By typing VF (flat), VN (north),VS (south), VE (east), or VW (west), the UCS icon (if UCSICON is on and at its' origin) will change to the appropriate plane. Other areas of input for this routine are located from the pulldown under Plant>UCS. For example, start a new drawing and at the command prompt type VPOINT as demonstrated below.

Command: **VPOINT**
*** Switching to the WCS ***
Current view direction: VIEWDIR=1.0000,-1.0000,1.0000
Specify a view point or [Rotate] <display compass and tripod>: 1,-1,1
*** Returning to the UCS ***

The user can also pick the DDVPOINT command and enter 315 from X Axis and 35.3 from XY Plane.

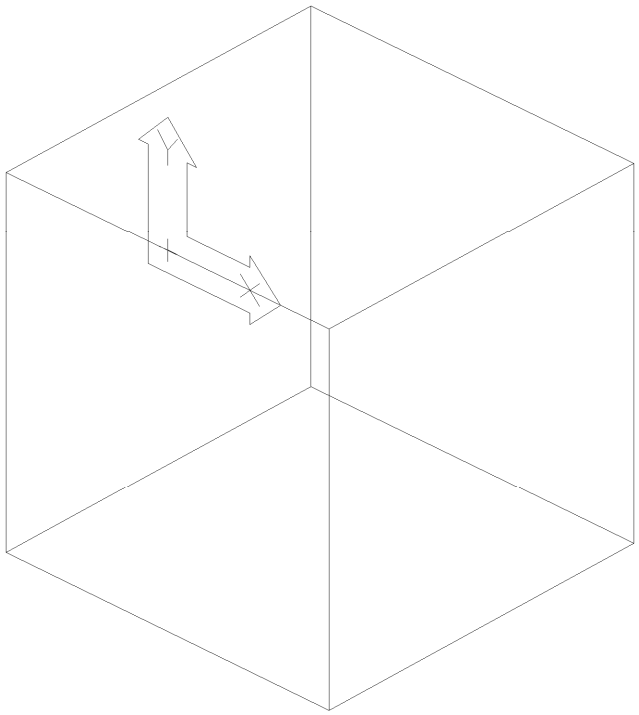
Type "AI_BOX" at the AutoCAD Command Line to start the next step:

Command: **AI_BOX**
Initializing... 3D Objects loaded.
Specify corner of box: **2,2 (50,50)**
Specify length of box: **4' (1220)**
Specify width of box or [Cube]: **5' (1524)**
Specify height of box: **5' (1524)**
Specify rotation angle of box about the Z axis or [Reference]: **0**

Command: **ZOOM**
Specify corner of window, enter a scale factor (nX or nXP), or
[All/Center/Dynamic/Extents/Previous/Scale/Window] <real time>: **E**

Enter the alias for the CADWorx Plant's VIEWNORTH command.

Command: **VN**
[Xtext/Ztext/Elevation/Rotate/COordinate/CLip] <Pick point, or enter>: *MIDpoint of the box as shown*



This would place the UCS in a north working position. Now, select Plant>UCS>UCS Next from the pulldown, and hit enter several times. This places the UCS in the most common working planes for drawing any type of piping system. This icon and position can be changed by typing the letter representing the desired position and just pointing.

Command: **VIEWFLAT** (or **VIEWNORTH**, **VIEWSOUTH**, **VIEWEAST**, and **VIEWWEST**)
Xtext/Ztext/Elevation/Rotate/COordinate/CLip/<Pick point, or enter>:

Xtext

When viewing a model from the bottom side, using Vpoint with a negative angle from the X-Y plane, text and dimension placement will be upside down and backwards. This option allows the rotation of the X axis of the UCS to remedy this. Text or dimensions will then be placed in the proper orientation for viewing and plotting. If changing from a negative to positive angle from the X-Y Vpoint, the text will have to be replaced for the proper orientation. This option will rotate the UCS 180 degrees along the X-axis.

Ztext

When viewing a model from the back side, using Vpoint with a 0 to 180 degree angle in the X-Y plane from the X axis, text and dimension placement will be upside down and backwards. This option allows the rotation of the Z axis of the UCS to remedy this. Text or dimensions will be placed in the proper orientation for viewing and plotting. If changing to a Vpoint rear view position (180 - 360 degree angle in the X-Y plane from the X axis), the text will have to be replaced for the proper orientation. This option will rotate the UCS 180 degrees along the Z axis.

Elevation

This option is for setting the elevation to present UCS. This command does not work for setting depth when in a North, South, East or West plane, instead, by setting an elevation of 10' (when the present elevation is 0') will raise the UCS up to 10' regardless of the position.

Enter desired elevation <*current elevation*>: *enter elevation*

Rotate

This option is for rotating the present UCS around the X, Y, and Z axis. For example: set to 45 degrees after inserting a 45 degree elbow allows placement of other components in line with the 45 degree elbow.

X/Y/Z rotation angle: *enter angle or option*

All these options allow the UCS to be rotated around whichever axis is selected here.

COordinate

This option allows entry of an exact X,Y,Z coordinate in relation to the WCS (world coordinate system).

Enter desired coordinate: *enter coordinate (x,y,z)*

CLip

This option allows view clipping to be performed at the present plane set by the UCS. This plane can be established with the Point and Shoot UCS options or any three point UCS position. The plane can be skewed at any angle.

Enter an option [**Front/Back/Off/On**] <On>: *choose one of the option or enter*

On

This option turns view clipping on with clipping distances as set by the Back and Front options below.

Off

This option turns view clipping off.

Back

This option sets the distance to the back of the clipping area. It must be a negative number. If this number is positive, it could clip past the front clipping area, therefore leaving nothing in the view.

Enter negative distance to back clipping plane <-5'-0">:

Front

This option sets the distance to the front of the clipping area. It must be a positive number. If this number is negative, it could clip past the back clipping area, therefore leaving nothing in the view.

Enter positive distance to front clipping plane <5'-0">:

Note: This option *CLip* is also available as the command VIEWCLIP.

Pick point

By picking a point, the UCS will move the 0,0,0 user base to the picked location. By hitting enter, the UCS will remain at the same origin and switch to the new plane. For example: Hit enter to change from north to a west plane.

Other UCS positions required by the user not supplied with CADWorx Plant's point and shoot routine will have to be entered with AutoCAD's UCS command. Two extremely useful options in this command are the OBject and 3 Point. We advise leaving the UCSICON on and set to origin when using any method of re-arranging the UCS. This is accomplished with the UCSICON command.

Note: If a point is picked close to the screens edge and the UCS icon is on and set to origin, the icon might not remain at this point because AutoCAD requires that the whole icon be visible. It will place the icon in the lower left corner of the screen. If this happens, zoom out a little or pan toward the new UCS position.

As with all CADWorx Plant commands, aliases are available for each command. See Appendix E: Alias and Command Names (see page 641) for a complete list of the commands and aliases.

Isometric Modeling

CADWorx Plant's various component routines sometimes require that an option be selected (type the first letter of the option). Suppose a tee starting at the branch location was required. The following example demonstrates the required input after selecting the component from either the pulldown menu or the toolbar.

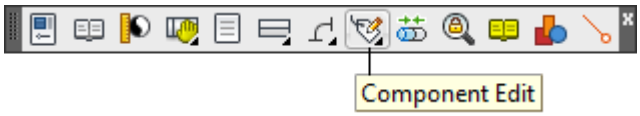
Command: **TESW**
4" TEE, STR. S/STD, ASTM A-234 GR WPB
Pick start point or [Branch/Center] <last point>: **B** *(type B)*
Pick branch end point <last point>: *pick point or enter to connect to the last placed component*
Pick center direction: *pick direction*
Pick main end direction: *pick direction*

The available options were C (center), B (branch) or picking a point for the main buttweld end. By typing "B" at the first line of the options, the program will prompt "Pick branch end point". Another feature of CADWorx Plant's piping component routines, is the ability to remember the last point. If a buttweld pipe was to be placed on the end of the tee drawn above, hitting enter is all that is required.

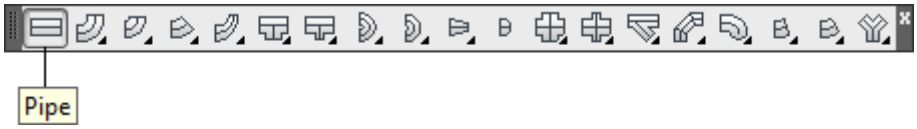
Command: **PIPW**
4" PIPE, S/STD SMLS, ASTM A-106 GR B
Pick start point or [TOP/BOP] <last point>: *enter for last point at the end of the tee*
Pick end point: *pick point*

Since the tee has two main buttweld points to select from, the last point option might not give the end desired. However, this option is extremely useful for placement of components down a single pipe run. If a component is inserted, then deleted or undone, the last point option will provide a point on the last placed component that still exists in the drawing or prompt for the last point. All components can have nodes attached to the centerline at the ends. This is controlled by the NodePlacement (see page 60) variable controlled from the configuration. The component's centerline has extended data (xdata) attached that contains all the vital information about the component. This information is used by other CADWorx Plant routines. As long as AutoCAD's setvar PICKSTYLE is set to 1 (on), the components will be grouped and will act as one piece for moving, deleting and editing.

Editing the component's xdata is accomplished by typing CEDIT at the command line or by picking the button shown below that can be located from the pulldown under Plant>Toolbars>Setting.

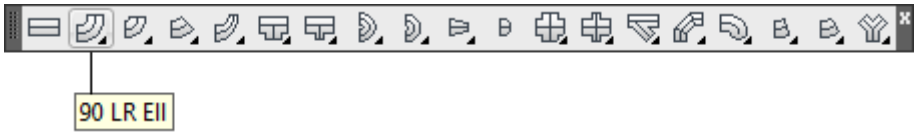


For this section, use the drawing created in the Setup section of this tutorial or open TUTOR1.DWG (TUTOR1M.DWG). Set an appropriate view as demonstrated below. From AutoCAD's pulldown menus, chose the View>3D Views>SE Isometric. This provides an excellent view for the purpose of the tutorial that follows. From the Buttweld toolbar (located from the pulldown under Plant/Toolbars/Buttweld) select the buttweld pipe as shown.



Command: **PIPW**
4" PIPE, S/STD SMLS, ASTM A-106 GR B
Pick start point or [TOP/BOP] <last point>: **B** *bottom of pipe*
Pick start point: *0,0 or pick point*
Pick end point: *@48,0 (@1219,0) or rubber band the direction and enter 48*

Select the *90 LR Ell* from the toolbar below.



Command: **90LR**
4" ELL, 90%%D LR S/STD, ASTM A-234 GR WPB
Pick start point or [Corner] <last point>: *hit enter to connect to last point*
Pick corner direction: *@12,0 (@305,0) or pick direction*
Pick other direction: *@0,12 (@0,305) or pick direction*

Use the AutoCAD ZOOM command to better position the view.

Command: **ZOOM**
Specify corner of window, enter a scale factor (nX or nXP), or [All/Center/Dynamic/Extents/Previous/Scale/Window] <real time>: *enter to enable real-time zoom or E for extents*

Notice that the first end of the pipe does not appear to start from 0,0 as directed. This is caused by the selection of the *BOP* option (bottom of pipe) which tells CADWorx Plant to start the pipe half the outside diameter higher (2 1/4" in this example) than the current UCS. This is useful for setting bottom of pipe to top of steel elevations. The X and Y coordinates of the pipe end are at exactly 0,0.

Next, select the *Weld Neck* button from the toolbar below. This can be located from the pulldown under Plant>Toolbars>Flanges.



Command: **FLGW**
4" FLG, RFWN 150LB S/STD BORE, ASTM A-105
Pick start point or [Face end] <last point>: *pick point or enter for last*
Pick direction: **54,12 (1372,305)** *or pick direction*
4" GASKET, 1/8" THK, 150LB
Automatically placed...

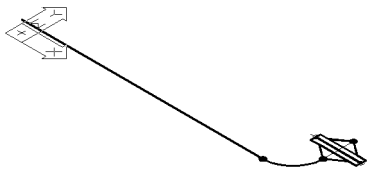
Notice the gasket was automatically placed. This function can be turned on or off from the pulldown under Plant/Utility/Drawing Control/Auto Gasket. In this tutorial, the AutoGasket function is assumed to be on.

Select Weld Neck flange again.

Command: **FLGW**
4" FLG, RFWN 150LB S/STD BORE, ASTM A-105
Pick start point or [Buttweld end] <last point>: *pick point or enter for last*
Pick direction: **54,36 (1372,914)** *or pick direction*

Notice that the options for the first weld neck flange was [Face] and its starting point was the buttweld end. The second weld neck flange option was [Buttweld] and it started at the face. The flange routines will always default to the buttweld point, as the majority of the other routines, unless a gasket is inserted. The gasket causes the flange routine to reverse the starting points.

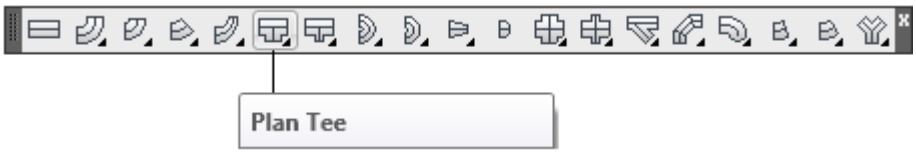
If drawn correctly, the drawing should appear as below. Zooming may be required to produce the exact view as shown below.



From the pulldown, located under Plant>Utility>Centerline, use the OFFSETCL (see page 325) command and offset the pipe 48" (1219mm), then draw the other components. By using the normal AutoCAD OFFSET command, the xdata information within the piping component will also be offset. Exploding the offset component can remove this information. For this example, use the CADWorx Plant's OFFSETCL command which will not copy the xdata.

Command: **OFFSETCL**
Specify offset distance or [Through] <4'>: **48 (1219)**
Select object to offset: *pick the pipe to offset*
Specify point on side to offset: *pick a direction to offset*
Offset to component BOP elevation [Yes/No] <Yes>: **N**
Select object to offset: *enter to finish*

When drawing the other components, in the Ortho mode, notice that the rubber banding of the points are not exactly at the crosshairs. This is caused by drawing out of plane. We recommend always working in plane, but, for this example, it will not cause a problem. When the crosshairs and the rubber-banding effect seems erratic (not being able to pick the point desired), it is usually caused by working out of plane. Use the Point and Shoot (see page 506) or the UCS command to re-align the UCS position. The objects drawn are at a 2-1/4" elevation and the current UCS elevation is at 0".



From the toolbar above, click on the TEE as shown

Command: **TESW**
4" TEE, STR. S/STD, ASTM A-234 GR WPB
Pick start point or [Branch/Center] <last point>: *enter for last point*
Pick main end direction: *pick direction in line with the flanges*
Pick branch end direction: *pick direction parallel with the offset line*

Command: **PIPW**
4" PIPE, S/STD SMLS, ASTM A-106 GR B
Pick start point or [TOP/BOP] <last point>: *enter to connect to the tee*
Pick end point: **PER** to *the offset centerline*

Command: **PIPW**
4" PIPE, S/STD SMLS, ASTM A-106 GR B
Pick start point or [TOP/BOP] <last point>: *enter to connect to the last placed pipe*
Pick end point: **ENDP** of *the offset line*

This leaves two pieces of pipe with a square corner which is not correct but an elbow can be inserted into the two pieces automatically which will be demonstrated next. From the same toolbar above, pick the 90LR button.

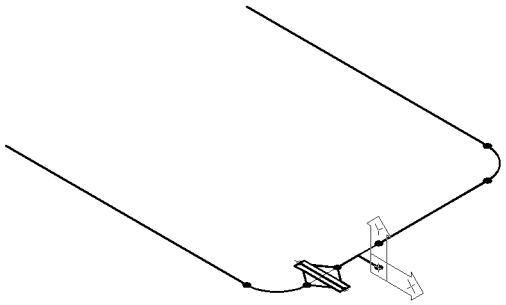
Command: **90LR**
4" ELL, 90%%D LR S/STD, ASTM A-234 GR WPB
Pick start point or [Corner] <last point>: **C**
Pick corner point: **INT** of *the corner of the two pieces of pipe*
Pick first direction: **NEA** to *either piece of pipe*
Pick second direction: **NEA** to *the other piece of pipe*

It should automatically break the pipe components back the distance of the 90 degree elbow in both directions.

For 2D and/or isometric development, breaking a pipe over or under another pipe can be accomplished with the Plant>Utility>Break>Manual-Auto (see page 332) command. For more congested areas, use the Plant>Utility>Centerline>Convert (see page 325) command on the centerline and then use the normal AutoCAD commands (BREAK or TRIM) to achieve the desired results.

Next, changing the working plane will be required to add components in the vertical or in the north plane. Type VIEWNORTH (or alias VN) and point to the branch end of the tee as shown below. The positive Z axis is now pointing to the lower left of the screen.

Command: **VN**
[Xtext/Ztext/Elevation/Rotate/COordinate/CLip] <Pick point, or enter>: *pick* **ENDP** of *the tee branch*



Command: **90LR**
4" ELL, 90%%D LR S/STD, ASTM A-234 GR WPB
Pick start point or [Corner] <last point>: **ENDP** of *the branch connection on the tee*
Pick corner direction: *pick direction parallel with branch outlet of tee*
Pick other direction: *pick direction in the positive Y (or upward)*

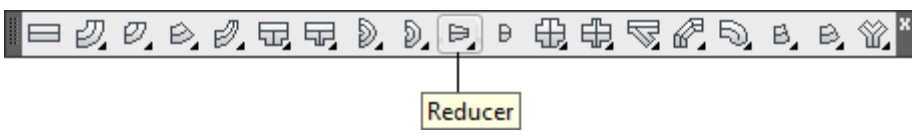
Command: **PIPW**
4" PIPE, S/STD SMLS, ASTM A-106 GR B
Pick start point or [TOP/BOP] <last point>: *enter to connect to the last point on the elbow*
Pick end point: *@0,12 (@0,305)*

Note: There is another way to enter pipe lengths within CADWorx Plant and AutoCAD. Dragging or rubberbanding the cursor in a direction and typing in 12 (305), then hitting enter, will result in drawing a pipe in the rubber band direction 12" (305).

Command: **TESW**
4" TEE, STR. S/STD, ASTM A-234 GR WPB
Pick start point or [Branch/Center] <last point>: *enter to connect to the pipe*
Pick main end direction: *pick direction in positive Y*
Pick branch end direction: *pick direction in negative X*

When inserting a reducing component, such as the concentric reducer, the main size and the reduction size will default large to small, providing the proper connection at the reducer. The option to start at the small end will be allowed in the insertion of the component as demonstrated below. After inserting a reducing component, the main size and the reduction size will reverse allowing other components down the pipe run to be placed without having to reset the sizes.

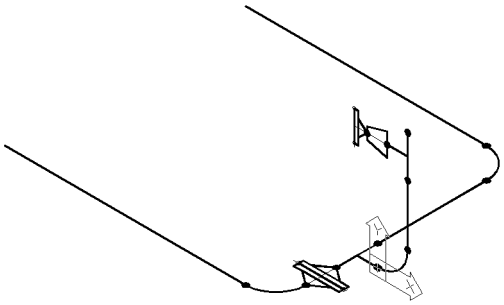
Set the reduction size to 2" and continue to draw the other components as shown below.



Command: **CONC**
4"x2" REDUCER, CONC S/STD, ASTM A-234 GR WPB
Pick large end point or [Small end] <last point>: **ENDP** of *the branch connection on the tee*
Pick small end direction: *pick direction in the negative X*

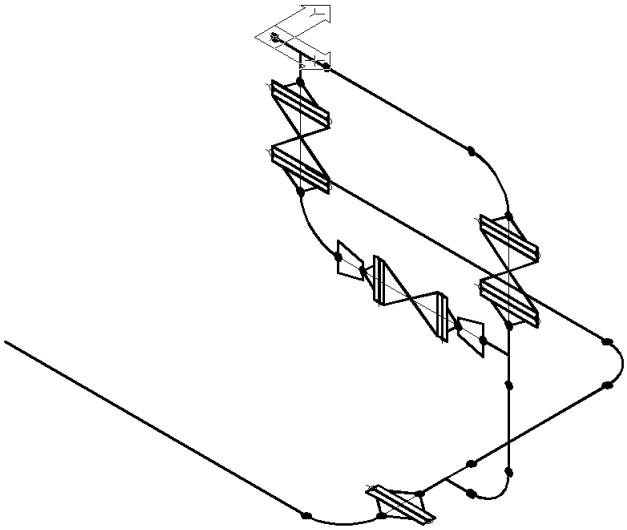


Command: **FLGW**
2" FLG, RFWN 150LB S/XS BORE, ASTM A-105
Pick start point or [Face end] <last point>: *pick point or enter for last*
Pick direction: *pick direction in the negative X*
2" GASKET, 1/8" THK, 150LB
Automatically placed...

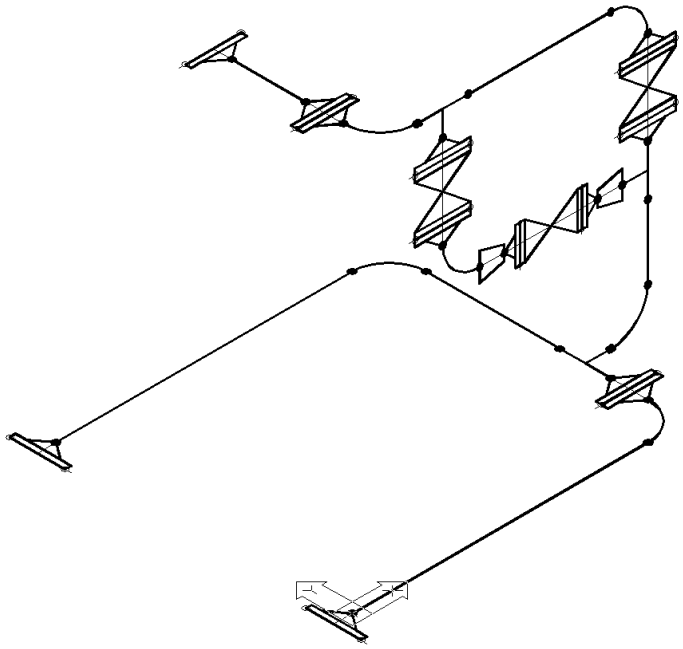


Note: The main size is now set to 2" and the reduction is set to 4".

Continue to draw the remaining components as shown. Add the gate valves, elbows, and additional piping to the drawing as shown below.



To accomplish the new flat UCS position, type VIEWFLAT (alias VF) and pick the end of the tee. Route the rest of the pipe in the new plane as shown. To improve the view of the drawing, give it another viewpoint. From AutoCAD's pulldown menus, chose the View>3D Views>SW Isometric for a better viewpoint. Type VF, point to the end of the first piece of pipe, then add the other two flanges at the end.



An isometric piping system in 3D space has been created. The components in the upper level of the isometric were originally drawn in a flat plane. To adjust these to a vertical position, select an east (type VE) or west (type VW) position then convert (from the pulldown, select Plant>Accessory>Mode Convert>2D Single line) the components to the new plane. Mode Convert (see page 295) can be used for any conversion (2D to 3D, Single to Double, or Single to Single, etc.). In this case leave the mode set to 2D Single line and convert single line to single line.

Explanations for converting this isometric into 2D orthographics and a 3D model will be presented in the next section. There are many other options which were not discussed in this topic that will be explained in later sections.

Mode Conversion

The Mode Convert (see page 295) routines (from the pulldown, Plant>Accessory>Mode Convert) can convert to any other mode or the same mode presently set. Converting to the same mode can be useful for taking a 2D double line plan and converting it to a 2D double line elevation. Mode Convert can restore a component to its original state after modification has been made to the component(s). In the previous isometric exercise, Mode Convert was used on single line components in a flat plane to convert them to a vertical (north or west) plane. Mode Convert will convert any component to any of the five pre-defined positions offered in the Point and Shoot UCS (see page 506) routine (flat, north, south, east, or west). Components not in one of the five pre-defined planes will not convert in the 2D modes.

For drawing in any mode, use the SETUP command earlier or pick the toolbar as shown below (pulldown Plant>Toolbar>Settings).



Command: **CONVERTSOLID**
Select objects: *enter to just set drawing mode*

Use the isometric drawing from the Isometric Modeling topic or open TUTOR2.DWG (TUTOR2M.DWG) to create 3D solids of the isometric model. From the pulldown, select Plant>Accessory>Mode Convert>3D Solids

Command: **CONVERTSOLID**
Pick piping components for 3D face conversion:
Select objects: ALL
Select objects: enter, to end selection

Using Solids

With solids, the view shown below can be created. Solids will also provide interference checking. Please refer to the appropriate Autodesk manuals for details on this procedure.

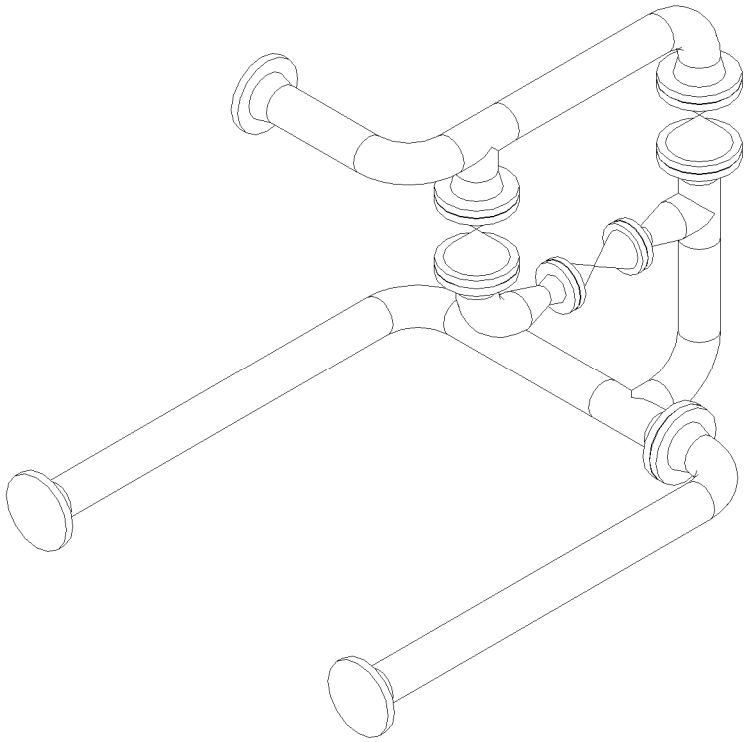
SOLPROFILE (an AutoCAD command) can be used to generate profiles in an Mview with paper space activated (tilemode = 0). This can be accomplished by issuing the SOLPROFILE command while in the mspace of an Mview. Another way to show hidden properties requires plotting the solids hidden. This process is much quicker. Use the HIDEPLOT option of the MVIEW command to hide the plot instead of the hide lines toggle in the plot dialog. If paper space is not desired, a profile can be generated and the block which is created in model space can be maintained and altered there instead.

Two setvar commands that are absolutely necessary while using solids are DISPSILH (display silhouettes) and ISOLINES (controls number of faces on an object). These must be set to 1and 0 respectfully. This will remove any faces normally associated with the solid. Other setvars which are helpful in working with solids are HIDEPRECISION, HALOGAP, OBSCUREDLTYPE, and OBSCUREDCOLOR. Please refer to the appropriate AutoCAD documentation for more information on these setvars.

Use the isometric model from the Isometric Modeling topic or open TUTOR2.DWG (TUTOR2M.DWG) to create a 3D solid model of the isometric. From the pulldown, select Plant>Accessory>Mode Convert>3D solids.

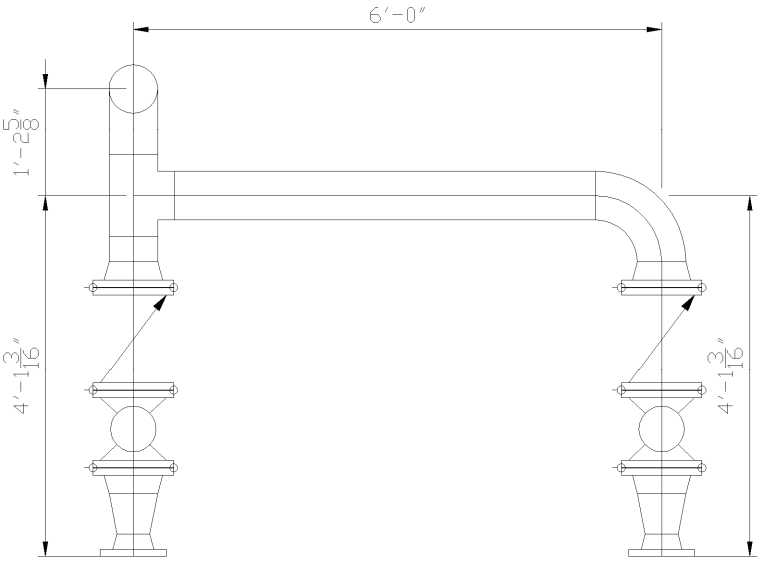
Command: **CONVERTSOLID**
Pick piping components for solid modeled conversion:
Select objects: **ALL**
Select objects: *enter, to end selection*

Command: **HIDE**
Hiding lines 100% done



Orthographic Piping

This section involves developing some discharge piping for two pumps. Open up drawing TUTOR3.DWG (TUTOR3M.DWG). Set the main size to 4” and the reduction size to 6”. Reducing components will use both of these sizes. Reducing components automatically sort the sizes in order to provide the desired graphical representation. Set the specification to 150 (150_M). See the first section on Setup for further instructions on accomplishing these procedures.



In a plan view, draw the pump discharge piping as shown above. First, place a 4” flange utilizing the Face option.

Note: This drawing already has a 3 dimensional pipe rack outline drawn. The discharge piping in this exercise will be placed on a set of pumps which will be drawn later. The piping can actually be drawn anywhere, although it is suggested to draw it within the view provided in the TUTOR3.DWG (TUTOR3M.DWG). This will accommodate easy placement later.

Command: **FLGW**
4" FLG, RFWN 150LB S/STD BORE, ASTM A-105
Pick start point or [Face end] <last point>: **F**
Pick start point <last point>: *pick a point*
Pick direction: *pick direction upward*

Then place a concentric reducer utilizing the *Small end* option.

Command: **CONC**
6"x4" REDUCER, CONC S/STD, ASTM A-234 GR WPB
Pick large end point or [Small end] <last point>: **S**
Pick small end point <last point>: *enter to connect to the flange*
Pick large end direction: *pick direction upward*

This will automatically flip the main and reduction sizes allowing placement of the next flange without changing the pipe sizes (main size equals 6” and reduction size equals 4”). Anytime a reducing component is encountered, the last size that was drawn will automatically be set to the main setting (4” small end to 6” large end, 6” large end drawn last). Continue to draw the 6” flange.

Command: **FLGW**
6" FLG, RFWN 150LB S/40 BORE, ASTM A-105
Pick start point or [Face end] <last point>: *enter*
Pick direction: *pick direction upward*
6" GASKET, 1/8" THK, 150LB
Automatically placed...

Note: Automatic last point connections. By hitting enter, connection to the last point is automatic. CADWorx Plant creates a list of components and points that it automatically selects from. This maintained list is approximately 30 components in length. When a component is deleted, this component remains in the list but will not be used. As components are added to this list, they are removed from the beginning and added to the end. Erasing components renders these locations void and the next valid component location in the list will be used. Simply hit enter and it will snap to the last valid or last drawn component.

Continue to draw the 6” ball and check valves. After drawing the 6” flange onto the check valve, copy the existing piping with the Copy command at a distance of 6’-0” (1829mm).

Command: **BALF**
6" BALL VALVE, 150LB FLG
Pick start point or [Center] <last point>: *enter*
Pick direction: *pick a direction upwards*
6" GASKET, 1/8" THK, 150LB
Automatically placed...

Command: **CHKF**

6" CHECK VALVE, 150LB FLG
Pick start point or [Center] <last point>: *enter*
Pick direction or [Reverse]: *pick a direction upwards*
6" GASKET, 1/8" THK, 150LB
Automatically placed...

Command: **FLGW**
6" FLG, RFWN 150LB S/STD BORE, ASTM A-105
Pick start point or [Buttweld end] <last point>: *enter*
Pick direction: *pick a direction upwards*

Command: **COPY**
Select objects: *pick window around components*
Specify opposite corner: 24 found, 9 groups
Select objects: *enter*
Specify base point or displacement, or [Multiple]: *pick any point*
Specify second point of displacement or <use first point as displacement>: *drag cursor to right and enter 6'*

Draw an elbow on the right side.

Command: **90LR**
6" ELL, 90%%D LR S/STD, ASTM A-234 GR WPB
Pick start point or [Corner] <last point>: **ENDP** of *the flange on the right*
Pick corner direction: *pick upward*
Pick other direction: *pick to the left*

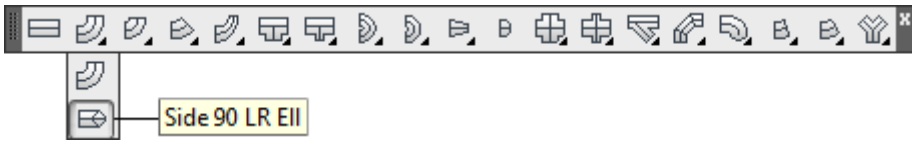
Either draw construction lines or use X, Y filters to locate the tee.

Command: **TESW**
6" TEE, STR. S/STD, ASTM A-234 GR WPB
Pick start point or [Branch/Center] <last point>: **C**
Pick center point: **.Y** of **ENDP** *of - pick the open buttweld of the elbow*
(need XZ): **ENDP** of *- pick the buttweld of the upper flange on the left*
Pick main end direction: *pick downward*
Pick branch end direction: *pick to the right*

Command: **PIPW**
6" PIPE, S/STD SMLS, ASTM A-106 GR B
Pick start point or [TOP/BOP] <last point>: *enter*
Pick end point: *pick the buttweld of the flange*

Command: **PIPW**
6" PIPE, S/STD SMLS, ASTM A-106 GR B
Pick start point or [TOP/BOP] <last point>: *pick branch point of tee*
Pick end point: *pick the open buttweld of the elbow on the right*

The remaining elbow is drawn using the toolbar flyout as shown below.



Command: **H90LR**
6" ELL, 90%%D LR S/STD, ASTM A-234 GR WPB
Pick start point or [Corner] <last point>: *pick the top of the tee*
Pick corner direction: *pick upward*
Enter direction [Up/Down] <Up>: *enter*

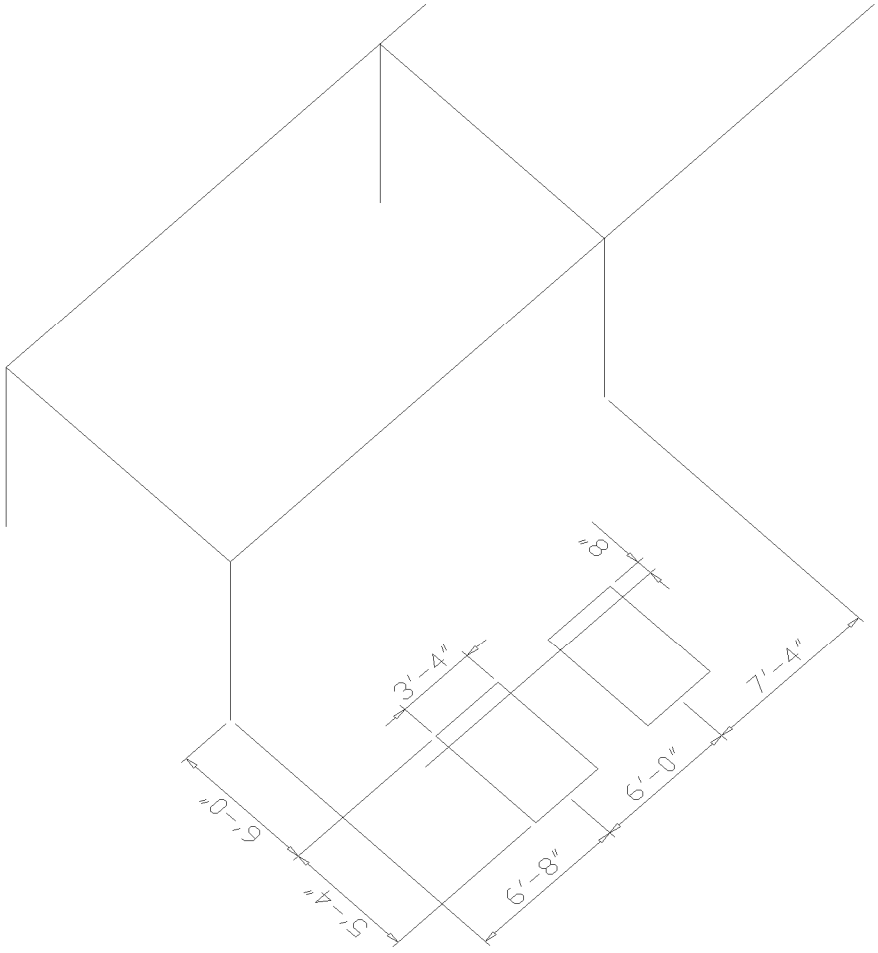
This drawing can be used in the next section for the 3D model development.

3D Model Development

In this section, two pumps and some discharge piping will be created. The discharge piping was created in an orthographic mode in the previous section of this tutorial. If the last section was not completed, open TUTOR4.DWG (TUTOR4M.DWG).

With the UCS set to world, draw the required construction lines for creating the pumps and pedestals as shown below (the orthographic piping has been omitted for clarity). Caution should be observed when drawing construction lines. Verify that they are being placed at a 0' elevation. It is very confusing when construction lines are at different elevations (intersections are hard to select, etc.). One way to insure this is by using an XY filter, then picking a known Z. Another way is to adjust the view slightly from the AutoCAD pulldown menu, select Views>3D Views>Northeast. This provides a method of picking the bottom end points of the pipe rack columns as shown below.

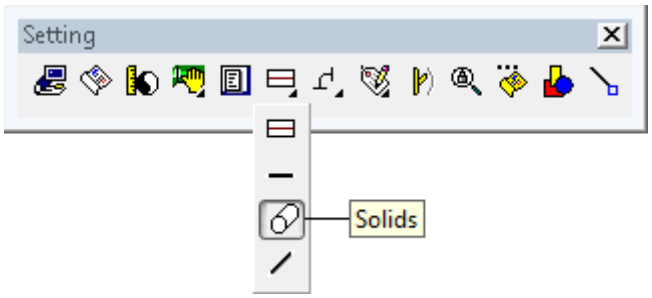
Draw the required construction lines as shown in the figure below (the edge of the pedestals are 6’-0” (1829mm) from the pipe rack; the pedestal is 5’-4” (1626mm) long by 3’-4” (1016mm) wide; the first pedestal centerline is 6’-8” (2032mm) from the end of the rack; the second pedestal centerline is 12’-8” (3861mm) from the end of the rack; the pump bases are set back on the pedestal 8” (203mm) from the front edge. After drawing the construction lines, the pump pedestal foundation must be drawn with a pline.



After drawing the pline, extrude it 2'-0" (610mm) with 0 degree taper. This will provide a pump foundation utilizing solids.

Command: **EXTRUDE**
Select objects: *select pump foundation pline*
Select objects: *enter to finish selection*
Path/<Height of Extrusion>: **24 (610)**
Extrusion taper angle <0>: *enter*

For the equipment, place the drawing mode in Solids. Pick the solid button from the Settings toolbar. Located here are double line, single line, solids, and isometric modes conversions and settings. These modes will govern how each component or piece of equipment is drawn.

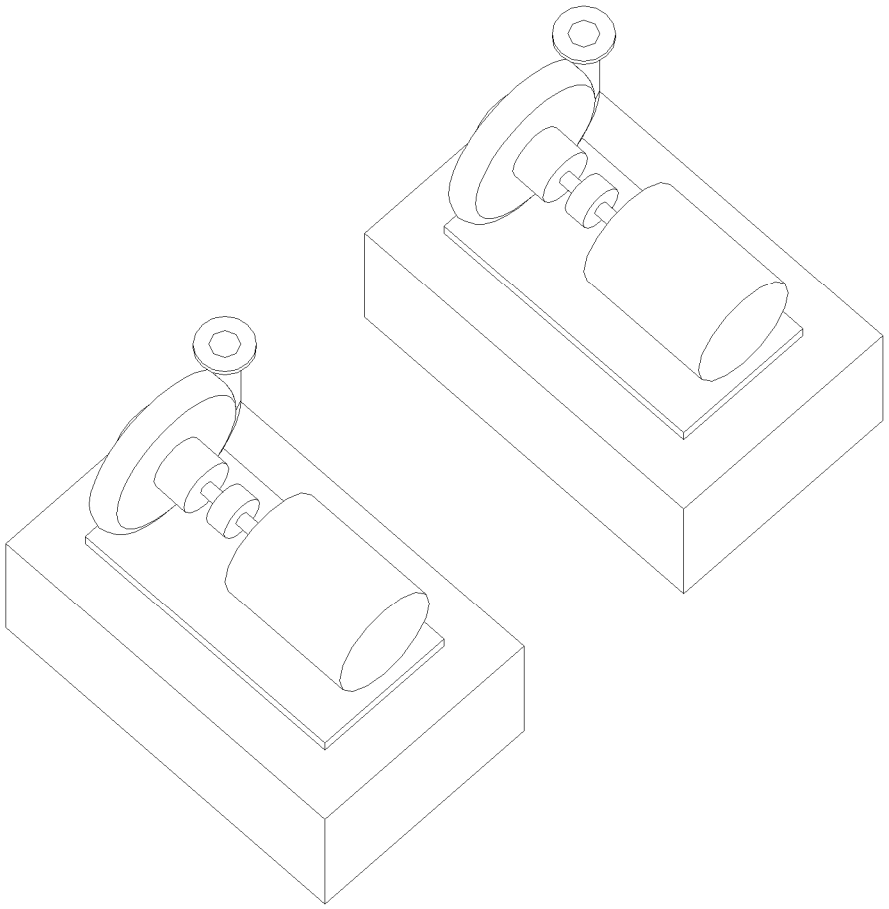


Select the PUMP command from the pulldown Plant>Equipment>Pump and draw the pump with the following dimensions.

Command: **PUMP**
3D-pump, View slide for prompts <Yes>: *enter, this provides a vslide as reference*
Enter nominal suction size: **6**
Enter nominal discharge size: **4**
Enter flange rating file (i.e. 150, 300m, 300mm or NP100) <150>: *enter for 150 (150M) lb rating*
Electric motor frame number or enter for list: **364T**
Discharge direction [Horizontal/Vertical] <Vertical>: *enter*
Discharge position [Right/Center/Left] <Left>: *enter*
<1> Enter distance, b.o.b. to centerline suction: **16 (400)**
<2> Enter distance, centerline suction to f.o.f. discharge: **16 (400)**
<3> Enter distance, centerline suction to centerline discharge: **12 (300)**
<5> Enter distance, f.o.f. suction to centerline discharge: **5 (125)**
<6> Enter distance, suction f.o.f. to front edge of base (- or +): **1 (25)**
<7> Enter base thickness: **2 (50)**
<8> Enter base width: **24 (600)**
<9> Enter base length: **48 (1200)**

The next step requires either the use of XY filters, moving the construction lines to 2'-0" (610mm) elevation, or drawing new construction lines from the top of the pump pedestals. In this exercise the construction filters will be used.

Pick front end of base start point: **.XY** of
INT of *construction lines that indicates start point of pump, centerline of pedestal and 8" offset*
(need Z): **24 (610)**
Pick direction to back end of pump: *pick direction*



This should provide one of the pumps as shown above. Copy this pump to the other location by using the AutoCAD COPY command.

Command: **COPY**
Select objects: *select by crossing* Other corner: 10 found
Select objects: *to complete selection*
<Base point or displacement>/Multiple: *pick point at pump base*
Second point of displacement: **@-72,0 (@-1839,0)**

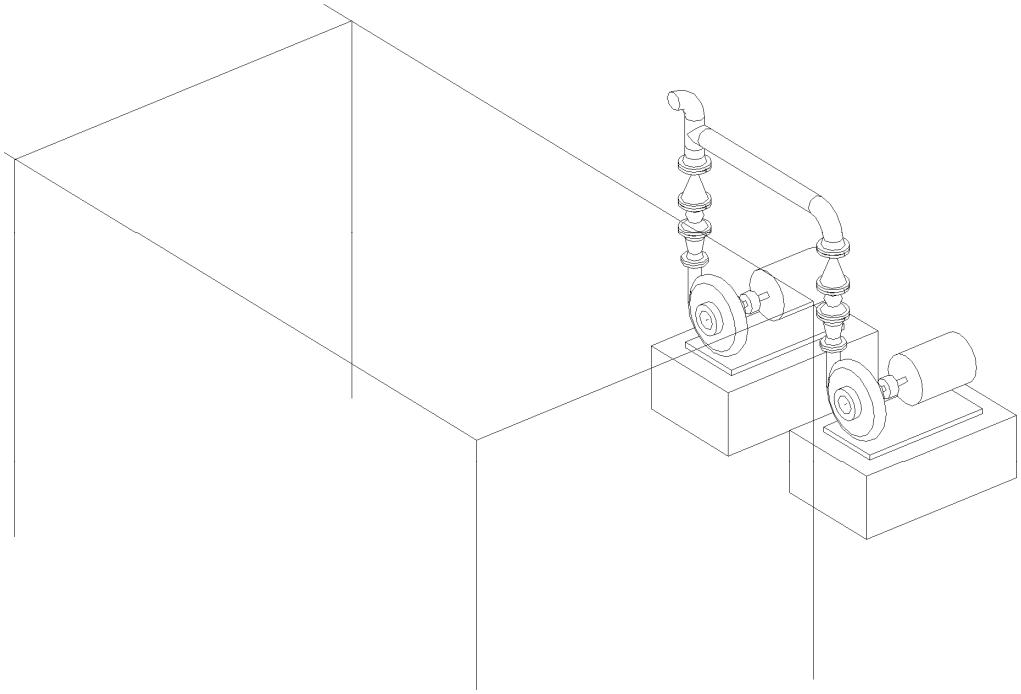
The next step to perform will be placing the discharge piping on the pumps. Re-position the view to better the placement of the piping.

Command: **PLAN**
<Current UCS>/Ucs/World: *enter for current*
Regenerating drawing.

Zoom in on the pumps and the discharge piping to better accommodate the placement of the discharge piping on the pumps. Placing the discharge piping on the pumps will be performed with the Elevation to Plan function. From the pulldown Plant>Accessory>Elevation to Plan, select this function to place the discharge piping on the pumps.

Command: **ELEV2PLAN**
Select objects: *select by crossing or window*
Specify opposite corner: 61 found
Select objects: *hit enter to finish selection set*
Pick reference point: **ENDP** of *lower 4" flange*
Component information [ON/Off] <Off>: **ON**
Draw pipe end baseballs [Yes/No] <No>: *enter*
Delete selection for future replacement [Yes/No] <No>: **Y**
Pick base point and rotation: **NODE** of *pick the discharge at the right pump (there is a node here) - rotate to left*

From the pulldown Plant>Accessory>Mode Convert>3D Solids convert all the components into solids. See the section on Using Solids (see page 533) for more information on using this function. Select a new view from the AutoCAD pulldown menu View>3D View>Southeast. It should now appear like the drawing below.



Automatic Routing

If the last section was not completed, open TUTOR5.DWG (TUTOR5M.DWG). In the tutorial drawing the T.O.S. (top of steel) of the pipe rack is at 12'-0" (3658mm). From the last 6" elbow on the discharge piping, a routing line will be required to apply the 6" discharge piping. Placement of the 6" pipe will be 1'-0" (305mm) from the edge of the pipe rack. Open up TUTOR5.DWG (TUTOR5M.DWG) and start the Router function from the pulldown Pipe>Accessory>Auto Route>Router. Next, construct the routing line.

Command: **ROUTER**
Specify start point or [Position/Reference]: **ENDP** of *the elbow*
Specify next point or [Slope/Position/Elevation/Undo/Reference]: **R**
Pick reference point: **INT** of *top of the pipe rack column closest and to the left of the pumps (see figure below)*
Use last point Z elevation <119.8125>: *enter to accept*
Enter relative/polar coordinates (with @): @6'4,-1' (@1930,-305) (*make sure the y is negative*)

This provides a point for the routing line 1'-0" (305mm) from the edge as discussed above. The next option in this function allows drawing the routing line 3 5/16" (84mm) above the 12'-0" (3658mm) T.O.S. (one half the pipe diameter). Utilizing the *Position* option in the router, the centerline routing can be moved up by half the pipe diameter. Not only can the position be changed, but the size can be verified also.

Specify next point or [Slope/Position/Elevation/Undo/Reference]: **P**
Enter an option [Nominal/Size/BOP/TOP/Centerline] <Centerline>: **B**
Nominal main size <6">: *hit enter here*

Now that the position has been relocated, entering the T.O.S. elevation is all that is required. First, select *Elev* option, and then select the *World* option to elevate the line to its proper location.

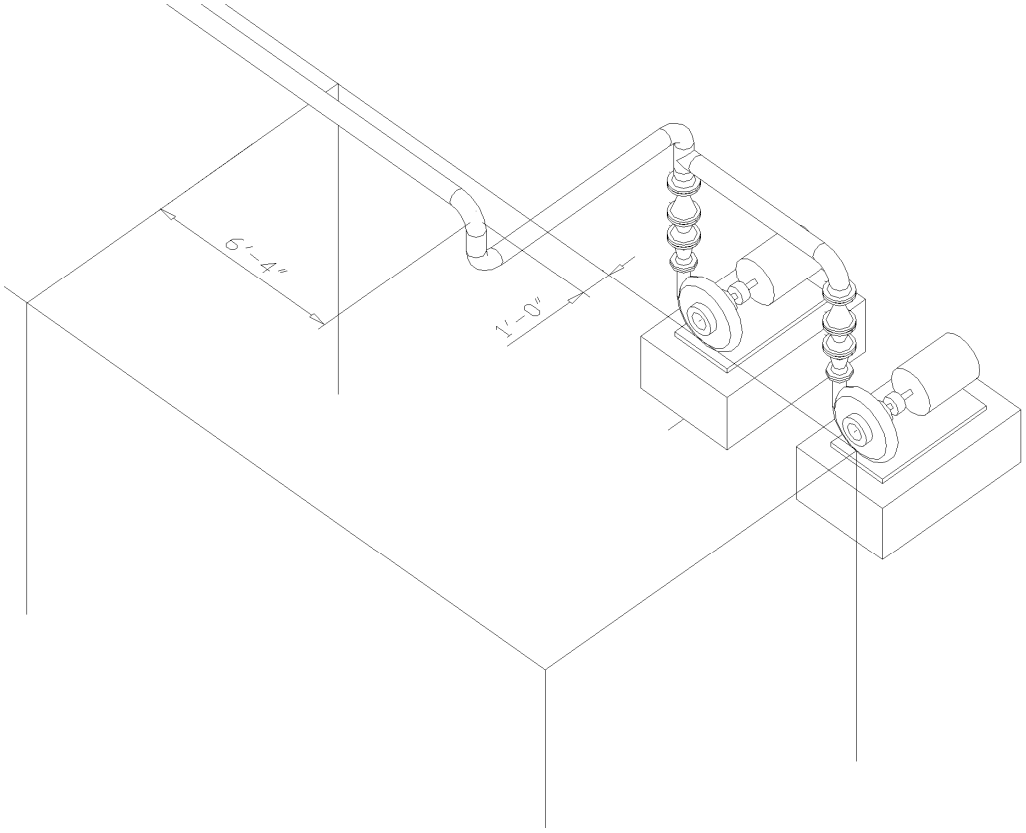
Specify next point or [Slope/Position/Elevation/Undo/Reference]: **E**
Enter elevation change or [World/Pick] <World>: *hit a return to use the world option*
Enter world elevation: **12' (3658)**
Specify next point or [Slope/Position/Elevation/Undo/Reference]: *pick a point down the pipe rack*
Specify next point or [Slope/Position/Elevation/Undo/Reference]: *hit enter to finish*

This provides a routing line for the desired piping run. Run the long radius buttweld routing function from Pipe>Accessory>Auto Router>Buttweld LR.

Command: **ROUTEBWLR**
Enter an option [Elbows only/Pick points/Select polyline] <Select polyline>: *return to select the routing line*
Select polyline or 3dpoly: *select the 3dpoly line*

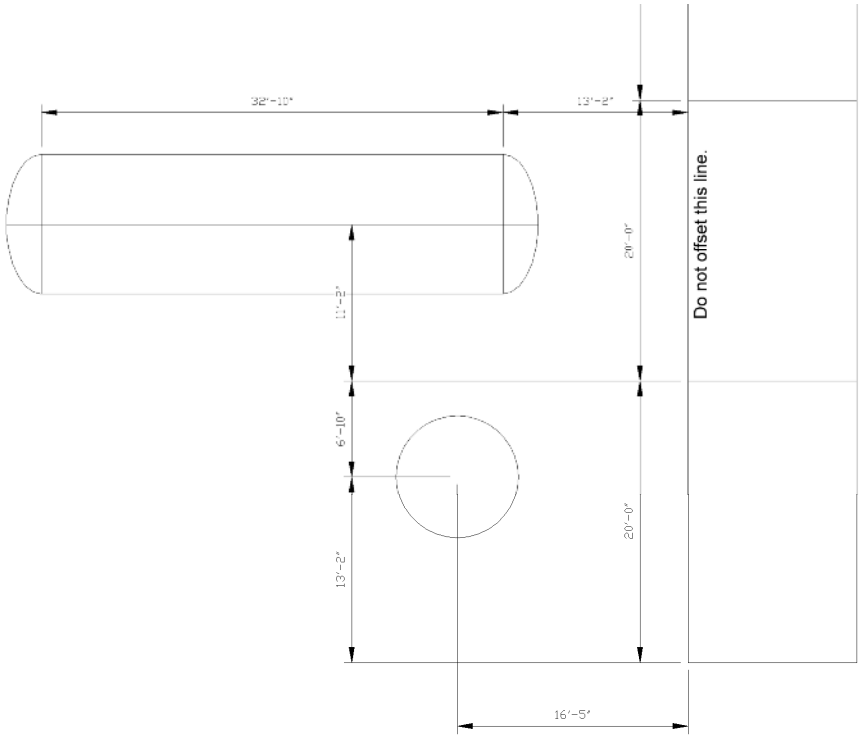
Change the view using the AutoCAD pulldown View>3D View>Southeast and convert everything to solids. This can be achieved with the mode conversion function. Choose from the pulldown menu, Pipe>Accessory>Mode Convert>3D Solids and convert all the piping to solids. Then utilize the AutoCAD HIDE command to produce the view below.

Command: **CONVERTSOLID**
Pick piping components for solid modeled conversion:
Select objects: *select object by crossing or window*
Other corner: 83 found
Select objects: *hit enter to end selection*


6'-4"
1'-0"

Equipment Development

If the last section was not completed, open TUTOR6.DWG (TUTOR6M.DWG). On the other end of the pipe rack, a vertical tower and vessel will be developed. Use construction lines to assist in the development of these components as shown below. As mentioned above, caution should be used to insure that all construction lines are at a 0' (0mm) elevation. Do not offset the longitudinal members of the pipe rack. This places construction lines at 15'-0" (4572mm) in elevation. Object snapping to an intersection of a line at 0' (0mm) and 15' (4572mm) is extremely difficult.



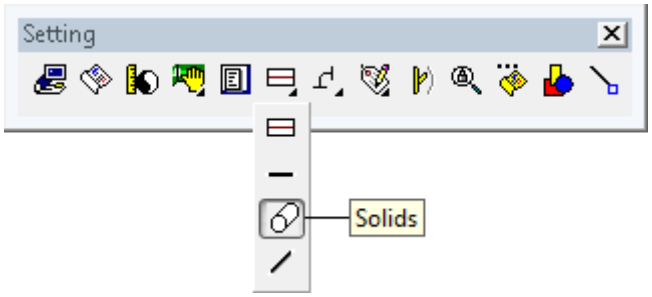
Utilizing solids for the equipment provides the best solution for both orthographic plans and elevations or any type of model. After completing the vessel, be sure to leave the center and seam construction lines. They will be used later to construct nozzles on this piece of equipment.

Draw the construction lines required to draw these two vessels as shown in the figure above (the seam of the horizontal vessel is 13'-2" (4013mm) from the pipe rack and the centerline is 33'-2" (10109mm) from the end of the pipe rack; the vertical vessel is 16'-5" (5004mm) from the pipe rack and is located 13'-2" (4013mm) from the end of the pipe rack). Use the short blue lines on the tutorial drawing to offset.

Starting with the horizontal vessel, use the Plant>Utility>Elevation>Change function to relocate the centerlines of the horizontal vessel to an elevation of 8'-8" (2642mm).

Command: **CHANGELEV**
Elevation change [BOP correction/Arbitrary] <Arbitrary>: *enter to accept arbitrary location*
Enter world elevation: **8'8 (2642)**
Select objects: *selection construction centerlines*
Select objects: *return to finish*
Pick base point: **ENDP** of *pick one of the construction lines*

Set 3D solids from the Settings toolbar.



Draw the vessel with the Plant>Equipment>Vessel command. Once created in solids, the vessel can not be changed to double line mode. Equipment does not have the capabilities to be converted. The desired mode must be set first.

The vessel dimensions are as follows.

Command: **VESSEL**
Enter head type [Flat/Dished/Hemispherical/Ellipical] <Elliptical>: *enter to accept 2:1*
Enter vessel OD: **8' (2438)**
Enter vessel seam-seam: **33'4 (10160)**
Enter head straight flange distance <0">: *accept 0, enter*
[Middle point] Pick head center at seam: **INT** of *the construction lines drawn*
Pick direction: *pick a direction*

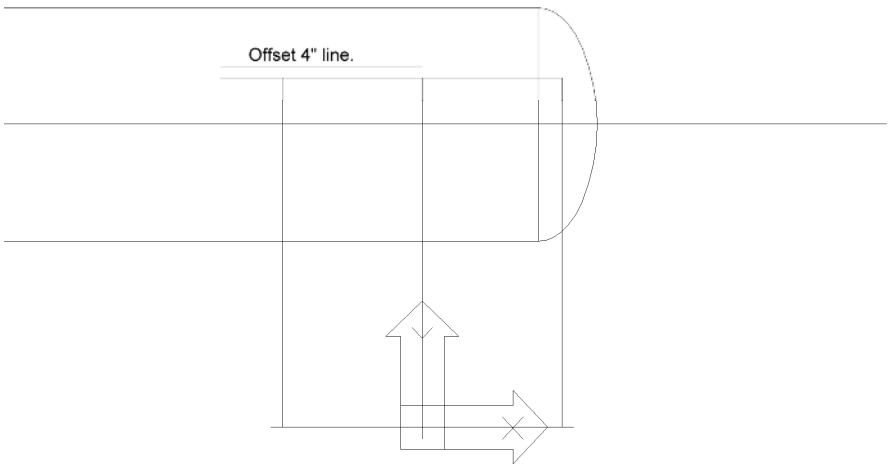
The next step to is to develop the vertical tower. For the skirt of this vessel, use CADWorx Plant's TANK command and choose the open roof. From the pulldown menu, choose Plant>Equipment>Tank.

Command: **TANK**
Enter roof option [Flat/Conical/Open] <Open>: *enter for open roof*
Enter tank OD: **8' (2438)**
Enter height of tank: **10' (3048)**
Enter bottom plate underlap: **4 (102)**
Pick center point of bottom: **INT** of *construction lines*

At this point, re-adjusting the view will be necessary from the AutoCAD pulldown View>3D Viewpoint>Front. Using the Tools>Toolbars>View toolbar also provides a convenient way of adjusting the view. After entering this view, type VIEWNORTH at the command prompt to re-orient the UCS. For the VIEWNORTH location, pick the end point of the previously placed skirt's centerline (or tank).

Command: **VIEWNORTH**
[Xtext/Ztext/Elevation/Rotate/COordinate/CLip] <Pick point, or enter>: **ENDP** of *centerline at the top*

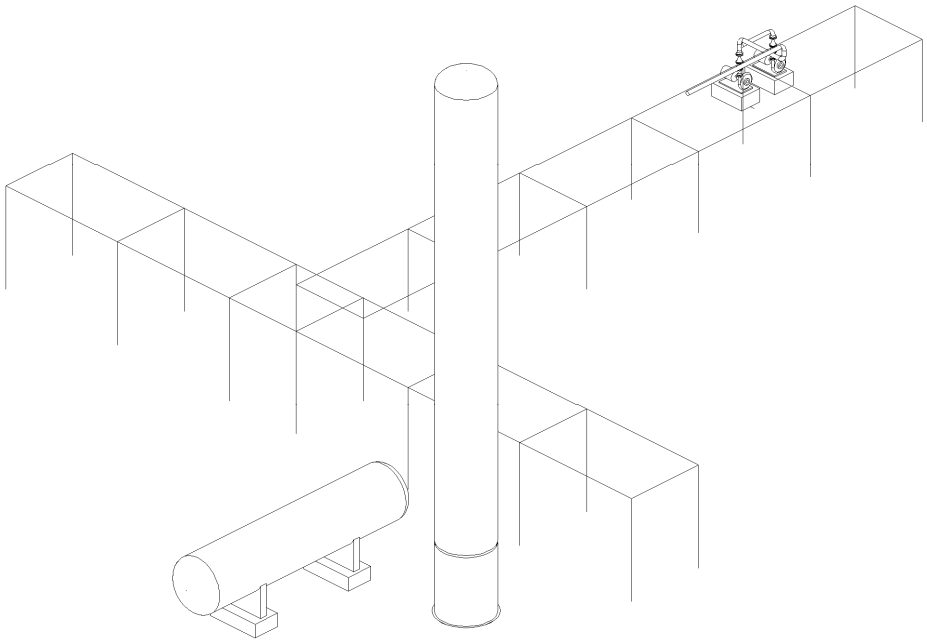
This will place the working plane in a north view. The remainder of the vertical tower can now be drawn. Draw a pline from the end point of the centerline to provide an offset for locating the seam of the vessel (see the figure below). Offset the construction line 4" (102mm). This provides a perfect location to start the vertical vessel.



From the pulldown menu, choose the Plant>Equipment>Vessel command again.

Command: **VESSEL**
Enter head type [Flat/Dished/Hemispherical/Ellipical] <Elliptical>: *enter to accept 2:1*
Enter vessel OD: **8' (2438)**
Enter vessel seam-seam: **66'8 (20320)**
Enter head straight flange distance <0">: *accept 0, enter*
[Middle point] Pick head center at seam: **ENDP** of *offset polyline*
Pick direction: *pick direction*

Provide some pedestals and supports for the horizontal vessel and a foundation for the vertical tower and this is the basis for the equipment layout.



Equipment Nozzles

If the last section was not completed, open TUTOR7.DWG (TUTOR7M.DWG). Placement of nozzles is easily accomplished by placing the UCS centered on a head and providing an elevation and rotation for the particular nozzle. Zoom in closer to the bottom of the vertical vessel.

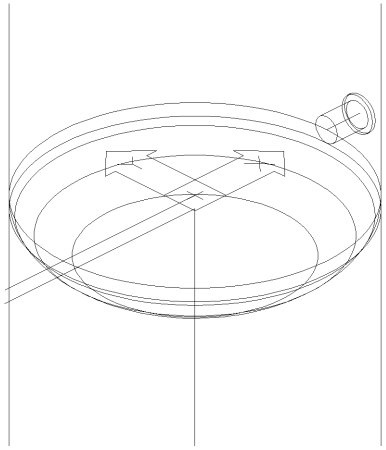
Type VIEWFLAT and place the UCS in the center of the circle that forms the seam of the bottom head.

Command: **VIEWFLAT**
[Xtext/Ztext/Elevation/Rotate/COordinate/CLip] <Pick point, or enter>: **CEN** of *the circle indicating the bottom tangent line*

Now place a 6" Nozzle (LWN) at the 0 degree quadrant of this circle as shown in the figure below.



Command: **LWN**
6" LONG WELD NECK, RF 150LB
Pick start point or [Length/Plain end] <last point>: **P**
Pick plain end point <last point>: **QUA** of *the tangent line of the bottom head*_qua of
Pick direction: *pick direction as shown*



Here the nozzle can be rotated to any degree based on a rotation point located at the UCS (0,0,0). After rotating the nozzle, it can be moved to its proper elevation with the AutoCAD MOVE command or the CHANGELEV command used earlier. Utilize the ROTATE and MOVE command to relocate this particular nozzle.

Command: **ROTATE**
Current positive angle in UCS: ANGDIR=counterclockwise ANGBASE=0.00
Select objects: *select the nozzle*
Select objects: *enter to finish*
Specify base point: **0,0**
Specify rotation angle or [Reference]: **45**

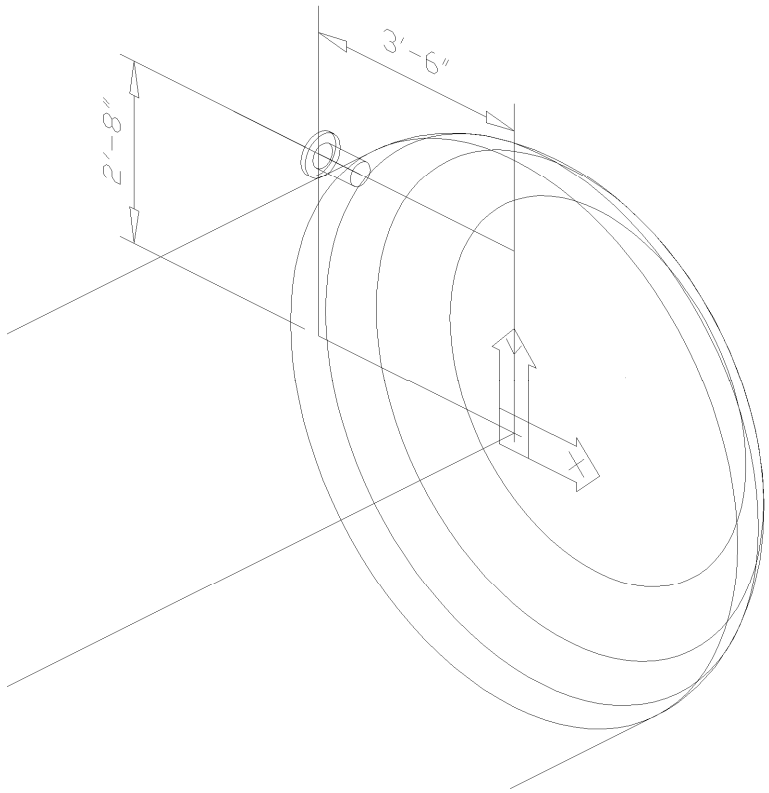
Command: **MOVE**
Select objects: *select the nozzle*
Select objects: *enter to finish*
Specify base point or displacement: *pick anywhere*
Specify second point of displacement or <use first point as displacement>: **@0,0,1'8 (@0,0,508)**

This relocated the nozzle 45 degrees around the vessel up 1'-8" (508mm) from the seam line. Using the pulldown Pipe>Utility>Elevation>Change would have allowed relocation of the nozzle by using real world coordinates (based on a distance from 0' (0mm) in elevation).

Add several more nozzles using the same procedure. Locate a 6" nozzle at 0 degrees and 3'-4" (1016mm) up, a 4" nozzle at 90 degrees and 30'-0" (9144mm) up, a 6" nozzle on the very top and a 10" nozzle on the bottom. For the top and bottom nozzles, relocate the UCS to a VIEWNORTH or to a VIEWWEST position.

Note: There is a small circle that the "center of" object snap can be used to locate the apex of the elliptical head. Erase or temporarily remove items so that it will be visible (the centerline of the skirt blocks the view of the one above). Although it looks like an AutoCAD node, use the "center of" object snap to snap to it.

The horizontal vessel will provide a new challenge by requiring two parallel nozzles on the side for mounting a level gauge. Place the UCS position to a VIEWEAST position in the end of the vessel head as shown in the figure below. Locating the nozzles will require offsetting a new vertical construction line 3'-6" (1050mm) and locating the face of the nozzle. Draw a new horizontal construction line and offset it upward 2'-8" (813mm). Trying to pick along the seam of the head with the object snaps "intersection of" or "nearest of" will not be possible due to the nature of the solids ("quadrant of" is possible). Although, construction aids are thought of as lines – they could also be circles (draw a circle over the seam), thus allowing any object snap around the seam line.


3'-6"
2'-8"

After placement of the two 3” nozzles, relocate them along the horizontal vessel similar to the vertical tower nozzles.

Command: **MOVE**
Select objects: *select the two nozzles*
Select objects: *enter to finish the selection set*
Base point or displacement: *pick any point*
Second point of displacement: **@0,0,40 (@0,0,1016)**

After performing this move procedure, the nozzles appear to be sitting in space. If they were drawn in double line mode as shown, mode convert them with the Pipe>Accessory>Mode Convert>3D Solids function. This provides a way of creating a union (of solids) with the horizontal vessel. After performing the mode convert, use the Union command.

Command: **UNION**
Select objects: *select the shell of the vessel*
Select objects: *select the top nozzle*
Select objects: *select the bottom nozzle*
Select objects: *enter to end and union all together*

In order to develop more tutorial piping, create several more nozzles. Draw a 6” nozzle on top 16’-8” (5080mm) from the seam line, 4” nozzle on the bottom 3’-4” (1016mm) from the seam line and a 4” nozzle on the side opposite from the level gauge. Union all the nozzles on the vertical tower and the horizontal vessel as above. Centerlines of the nozzles can be removed to avoid appearing in a bill of material generation. Turn PICKSTYLE off (grouping) to separate the nozzles and the centerlines. After using the UNION command, the centerlines will no longer be grouped with the graphics of the component.

In the real job environment, it is strongly suggested to develop each vessel or piece of equipment in separate drawings. These drawings could be easily named per any project requirement such as V-101 or P-203. Then XREF all the equipment into a master model. Wait to the end of the job to union all the equipment together. This allows the user to make modifications on the equipment anytime during the project. If the nozzles are unioned on the equipment, it is very difficult to change them. It is possible with the AutoCAD SOLIDEDIT command. Please refer to the AutoCAD documentation for explanations on this functionality.

This drawing is relatively sparse. All the components have easy access without the nuisance of other components and piping. When a drawing becomes crowded and it is difficult to select a component, XREFs can solve the problem. This area, with the two vessels, could have easily been built in a separate drawing and XREF’ed into this drawing and then edited with the AutoCAD REFEDIT command. To go one step further, both vessels could have been created in two separate drawings. When working on an XREF in another drawing, the components can easily be modified. All parts of the model can be XREF’ed together to form a completed model.

Inserting Components

If the last section was not completed, open TUTOR8.DWG (TUTOR8M.DWG). Type VIEWNORTH and pick the center of the vertical tower's seamline at the top. Type the AutoCAD PLAN command and this will provide a view that will be suitable for the next exercise.

Command: **VIEWNORTH**
[Xtext/Ztext/Elevation/Rotate/COordinate/CLip] <Pick point, or enter>: **CEN** of *pick the seam line at the top of the vessel*

Command: **PLAN**
Enter an option [Current ucs/Ucs/World] <Current>: *hit enter to accept Current*
Regenerating drawing.

Zooming in closer to the top of the tower will be necessary as shown in the figure below.

Command: **ZOOM**
Specify corner of window, enter a scale factor (nX or nXP), or
[All/Center/Dynamic/Extents/Previous/Scale/Window] <real time>: *pick a point as shown below*
Specify opposite corner: *pick another point*

Set the main size to 6" and the reduction size to 8". Set the drawing mode to single line. On the top of the vertical vessel, place a 6" gasket on the 6" nozzle. To select the face of the nozzle, after removing the centerline, requires using a "center of" object snap and picking the outer circle that forms the outside diameter of the flange. After the gasket, draw a 6"x8" relief valve with 8" gasket and flange attached to the relief end. Now, reset the main size to 8" after placing the relief valve (this component does not reset sizes like concentric reducers).

Pick these commands from the Plant>Toolbars>Gasket toolbar.

Command: **GAS**
6" GASKET, 1/8" THK, 150LB
Pick start point <last point>: **CEN** of *pick the flange at the top of the vessel*
Pick direction: *pick a direction upward*

Command: **RELF**
8"x6" RELIEF VALVE, 150LB X 150LB FLG.
Pick start point or [Length/Relief/Corner] <last point>: *enter to connect to the last gasket*
Pick corner direction: *pick a direction upward*
Pick relief end direction: *pick a direction to the right*

Set the main size back to 8".

Command: **MAIN**
Nominal main size <6">: **8**

Command: **GAS**
8" GASKET, 1/8" THK, 150LB
Pick start point <last point>: *enter to connect to the last point*
Pick direction: *pick a direction to the right*

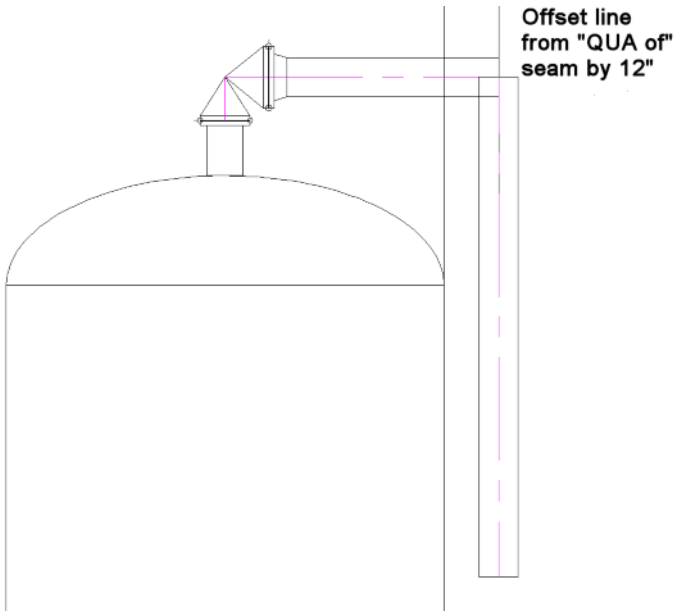
Command: **FLGW**
8" FLG, RFWN 150LB S/STD BORE, ASTM A-105
Pick start point or [Buttweld end] <last point>: *enter to connect to the last point*
Pick direction: *pick a direction to the right*

At this point, draw a construction line from the quadrant of the head as shown in the figure below. Offset this line by 1'-0" (305mm) to locate the 8" vertical pipe. At this point draw an 8" pipe from the flange located on the end of the relief valve and draw it "perpendicular to" the 8" vertical pipe. From here, draw a vertical piece of pipe from the end of the horizontal piece down the side of the vertical vessel. This will leave a square corner which will be used to insert an elbow.

Command: **PIPW**
8" PIPE, S/STD SMLS, ASTM A-106 GR B
Pick start point or [TOP/BOP] <last point>: *hit enter to connect to the last flange*
Pick end point: **PER** to *the vertical offset line*

Hit enter to re-issue the pipe command.

Command: **PIPW**
8" PIPE, S/STD SMLS, ASTM A-106 GR B
Pick start point or [TOP/BOP] <last point>: *hit enter*
Pick end point: *pick a direction anywhere down the side of the vessel*



The elbow can now be placed in the corner of the two pieces of pipe. Pick the elbow function and choose the corner option.

Command: **90LR**
8" ELL, 90%%D LR S/STD, ASTM A-234 GR WPB
Pick start point or [Corner] <last point>: **C**
Pick corner point: **ENDP** of *the two pieces of pipe*
Pick first direction: *pick direction*
Pick second direction: *pick other direction*

To continue this piping, change the view point with the DDVPOINT command. Enter 340 degrees from the X axis and 40 degrees from the XY plane. Type VIEWFLAT and pick a pipe rack cross member (endpoint of columns will not work). This provides a view similar to the view below.

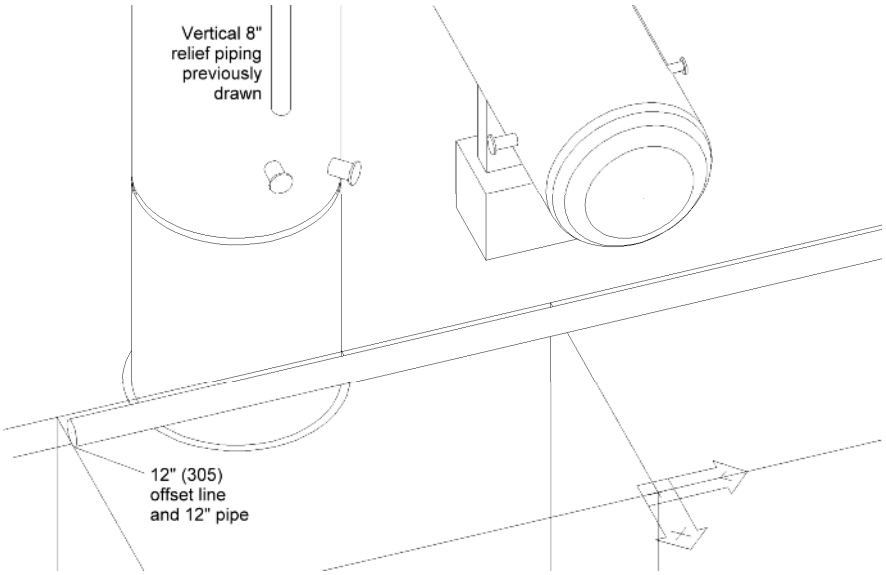
First, grip and drag the centerline of the vertical pipe down the side of the vessel for future modification.

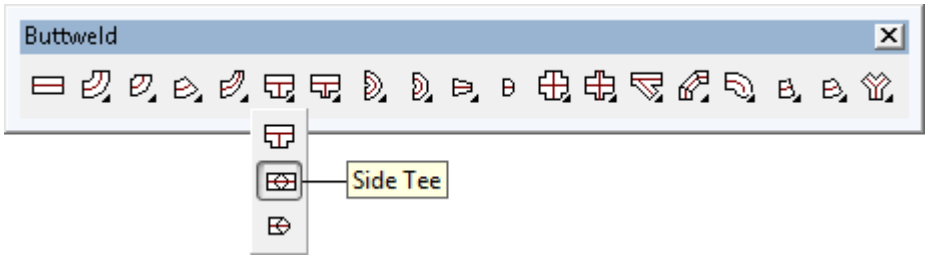
Command: *pick the pipe and then grip the end point*
** STRETCH **
Specify stretch point or [Base point/Copy/Undo/eXit]: *pull it down*

Next a 12” pipe can be set on the pipe rack in order to accept this 8” vertical relief piping. The next step is to draw the 12” pipe. Draw a short line from the corner of the pipe rack as shown in the figure below. Offset this line 1’-0” (305mm). Issue the pipe function from the toolbar. Utilize the BOP option to place the bottom of pipe equal to the TOS. Set the size to 12”.

Command: **PIPW**
12" PIPE, S/STD SMLS, ASTM A-106 GR B
Pick start point or [TOP/BOP] <last point>: **B**
Pick start point: **ENDP** of *the offset line and the pipe rack cross member*
Pick end point: *pick a direction down the pipe rack*

Now there are enough construction points to allow the remaining piping to be drawn. Utilizing the AutoCAD XYZ filters, select an 12”x8” vertical tee (from the toolbar flyout), and locate it along the 12” pipe with the Y filter of the vertical piece of pipe.



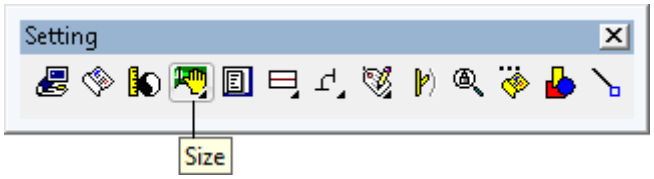


Command: **TERW**
12"x8" TEE, REDUCING S/STD, ASTM A-234 GR WPB
Pick start point or [Center] <last point>: **C**
Pick center point: **.Y** of **NEA** to *pick the centerline of the vertical relief piping at the tower*
(need XZ): **NEA** to *pick anywhere along the 12" pipe setting on the rack*
Pick direction: *pick direction parallel with the rack*
Enter direction [Up/Down] <Up>: *hit enter to accept or up*

This last prompt allows re-adjustment of the component's distance from the end of the pipe. Here, a different distance could have been enter if so desired. This is useful when a random point is selected on a pipe and then an exact known distance can then be entered to re-adjust the component's location.

Enter distance or [Round distance] <13'-2">: *hit enter to accept this distance from end of pipe*

This automatically inserts the 12"x8" reducing tee into the 12" pipe in line with the vertical 8" relief piping. All that remains now is to tie the two pipe runs together. Type VN (view north) and select the endpoint of the vertical 8" pipe. Set the main size back to 8" by utilizing the COMP2SIZE (see page 301) function. The toolbar flyout is shown below.



Command: **COMP2SIZE**
Select component to set size:
Main size set to 8"...

Now that the size is 8” and the UCS has been relocated to accommodate the remainder of the piping, place an 8” elbow on top of the tee and complete the pipe run utilizing the automatic insertion methods demonstrated.

Command: **90LR**
8" ELL, 90%%D LR S/STD, ASTM A-234 GR WPB
Pick start point or [Corner] <last point>: **ENDP** of *the top to the reducing tee*
Pick corner direction: *pick a direction upward*
Pick other direction: *pick a direction toward the vertical vessel*

Command: **PIPW**
8" PIPE, S/STD SMLS, ASTM A-106 GR B
Pick start point or [TOP/BOP] <last point>: *hit enter to select the end of the elbow*
Pick end point: **PER** to *the vertical piece of pipe at the vertical vessel*

Zoom in closer to the horizontal pipe and the vertical relief piping at the vessel.

Command: **90LR**
8" ELL, 90%%D LR S/STD, ASTM A-234 GR WPB
Pick start point or [Corner] <last point>: **C**
Pick corner point: **ENDP** of *the horizontal pipe at the vertical pipe*
Pick first direction: *pick horizontal direction*
Pick second direction: *pick vertical direction*
Enter distance or [Round distance] <1'-11 5/8">: *hit enter, the distance will not be the same as this one*

Erase the remaining piece of pipe that might have been left over from the insertion. If there is no other remaining pipe, the gripped pipe earlier was dragged within the 12” (305mm) range of the elbow just placed. If a gap between the elbow and the vertical piece of pipe exists, then the pipe was not dragged down far enough. If this is the case, just drag the pipe down to the elbow.

Next the horizontal vessel will require piping all the way back to the pumps created in the very beginning. The same construction line that was used to offset the 12” pipe 1’-0” (305mm) from the edge will be used again to offset the next 6” pipe 2’-0” (607mm) from the edge (1’-0” (305mm) from the 12” pipe). Perform this offset and place a 6” pipe parallel to the 12” pipe. Utilize the BOP option similar to the 12” pipe run.

Set the size to 6” and reset the UCS back to the top of the pipe rack. Type VIEWFLAT to relocate the UCS.

Command: **VIEWFLAT**
[Xtext/Ztext/Elevation/Rotate/COordinate/CLip] <Pick point, or enter>: **ENDP** of *the pipe rack as located earlier*

Command: **MAIN**
Nominal main size <8">: **6**

Command: **PIPW**
6" PIPE, S/STD SMLS, ASTM A-106 GR B
Pick start point or [TOP/BOP] <last point>: **B**
Pick start point: **ENDP** of *the offset line and the pipe rack cross member*
Pick end point: *pick a direction down the pipe rack*

Type VIEWNORTH and pick the center of the horizontal vessel at the head apex or tangent line for the next exercise. Issue the AutoCAD PLAN command and hit enter to accept the current UCS position. This should provide a view similar to the figure below.

Attach a 6" pipe run to the nozzle on top of the horizontal vessel. The first requirement is to place a gasket on the nozzle using the “center of” osnap and then a flange. Draw a piece of pipe from the flange and then another elbow as shown below. After that stretching the piping to the same BOP as the 8” pipe will be demonstrated.

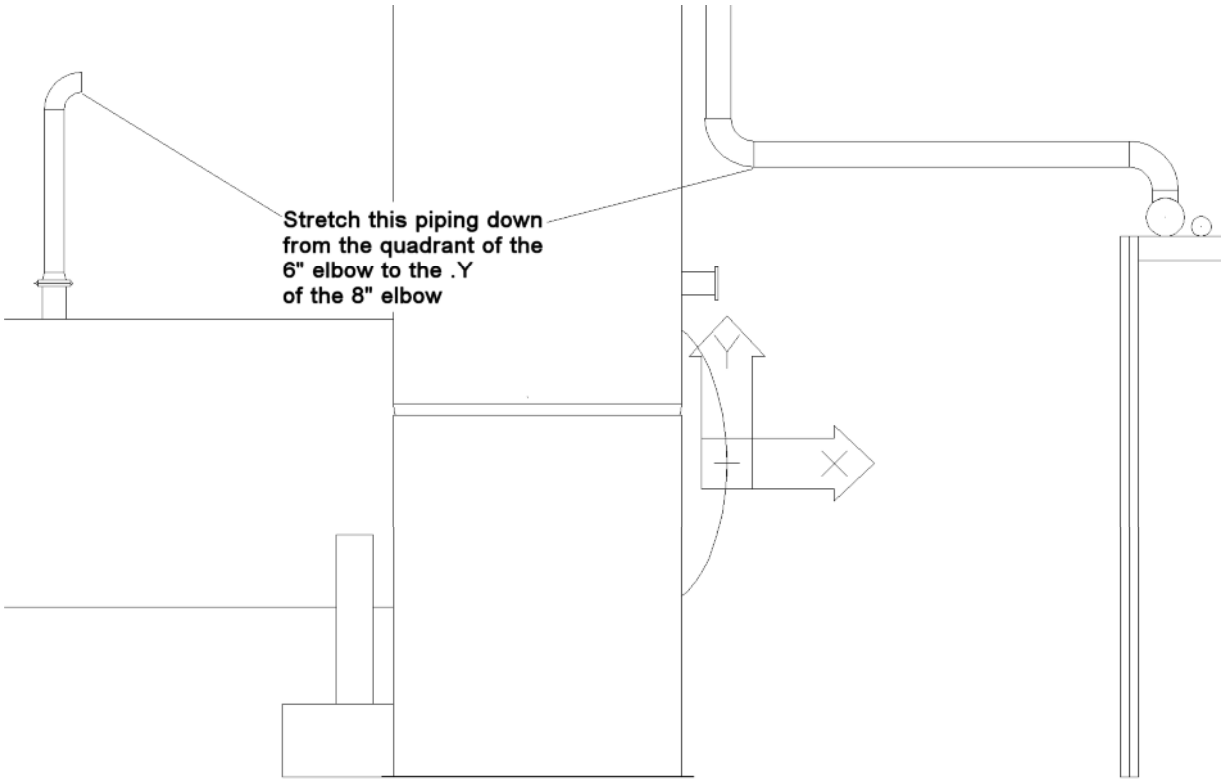
Command: **GAS**
6" GASKET, 1/8" THK, 150LB
Pick start point <last point>: **CEN** of *the flange on top of the horizontal vessel*
Pick direction: *pick a direction upward*

Command: **FLGW**
6" FLG, RFWN 150LB S/STD BORE, ASTM A-105
Pick start point or [Buttweld end] <last point>: *hit enter to connect to the last component*
Pick direction: *pick a direction upward*

Command: **PIPW**
6" PIPE, S/STD SMLS, ASTM A-106 GR B
Pick start point or [TOP/BOP] <last point>: *hit enter*
Pick end point: *pick a direction upward*

Command: **90LR**
6" ELL, 90%%D LR S/STD, ASTM A-234 GR WPB
Pick start point or [Corner] <last point>: *hit enter*
Pick corner direction: *pick a direction upward*
Pick other direction: *pick a direction toward the rack*

Stretch the 6” piping to match the BOP of the 8” piping. This will be accomplished by using the "quadrant of" object snap of the elbows.


Stretch this piping down
from the quadrant of the
6" elbow to the .Y
of the 8" elbow

Command: **STRETCH**
Select objects to stretch by crossing-window or crossing-polygon...
Select objects: *pick the first corner of a crossing around the 6" elbow*
Specify opposite corner: *pick the second corner of the crossing*
Select objects: *enter to finish the selection set*
Specify base point or displacement: **QUA** of *the 6" elbows bottom quadrant*
Specify second point of displacement or <use first point as displacement>: **.Y** of **QUA** of *the 8" elbows bottom quadrant*
(need XZ): *pick a point upward from the 6" nozzle*

Next draw a 6” piece of pipe from the other end of the new elbow to the pipe setting on the pipe rack.

Command: **PIPW**
6" PIPE, S/STD SMLS, ASTM A-106 GR B
Pick start point or [TOP/BOP] <last point>: **ENDP** of *pick the 6” elbow*
Pick end point: **.X** of **NEA** to *the centerline of the 6” piping on the rack*
of (need YZ): *pick anywhere in the present plane*

Next change the view point and finish the connection to the 6” pipe on the rack. In the DDVPOINT command, set from X axis to 315 degrees and from XY plane to 30 degrees. Zoom in on this spot and type VIEWWEST to relocate the UCS. Pick the end point of the 6” pipe setting on the rack for the relocated UCS. Draw two short radius elbows and connect the piping together (long radius elbows will not fit).

Command: **VIEWWEST**
Xtext/Ztext/Elevation/Rotate/Coordinate/<Pick point, or enter>: **ENDP** of *the pipe setting on the pipe rack*

Command: **90SR**
6" ELL, 90%%D SR S/STD, ASTM A-234 GR WPB
Pick start point or [Corner] <last point>: **C**
Pick corner point: **.X** of **ENDP** of *pick the horizontal piping coming from the vessel*
(need YZ): **NEA** to *pick anywhere along the 6” pipe setting on the rack*
Pick first direction: *pick a direction to the right*
Pick second direction: *pick a direction upward*
Enter distance or [Round distance] <31'-2">: *this is the distance from the end of the pipe to the corner of the elbow*

Command: **ERASE**
Select objects: *select the left over pipe*
Select objects: *enter to finish*

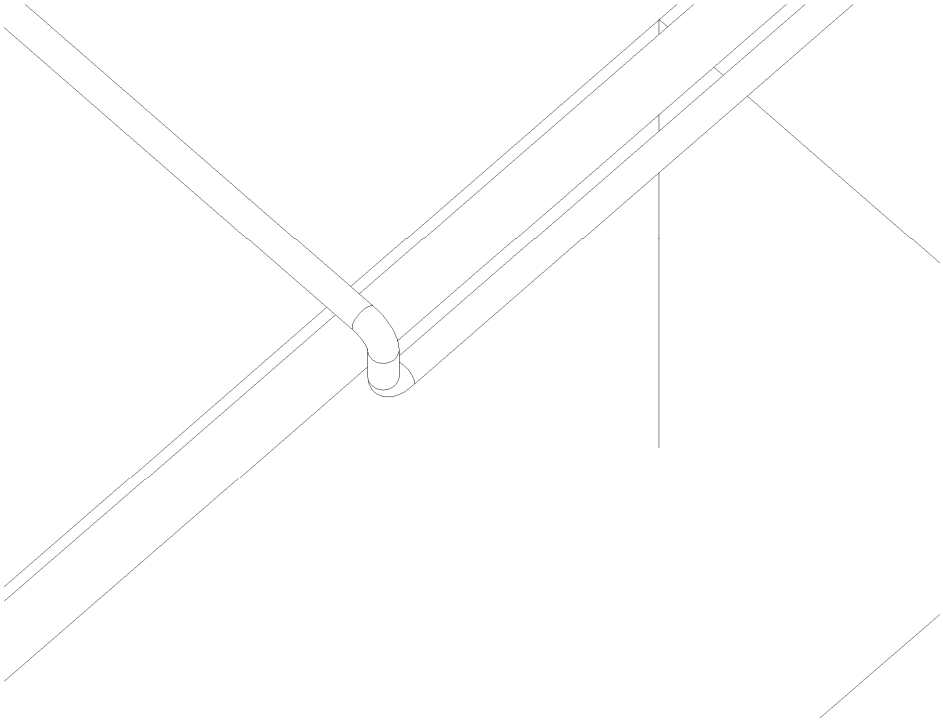
Command: **VIEWNORTH**
[Xtext/Ztext/Elevation/Rotate/COordinate/CLip] <Pick point, or enter>: **NEA** to *anywhere along the upper pipe*

Command: **90SR**
6" ELL, 90%%D SR S/40, ASTM A-234 GR WPB
Pick start point or [Corner] <last point>: **C**
Pick corner point: **ENDP** of *pick the end of the upper pipe*
Pick first direction: *pick a direction to the left*
Pick second direction: *pick a direction downward*

Command: **PIPW**
6" PIPE, S/STD SMLS, ASTM A-106 GR B

Pick start point or [TOP/BOP] <last point>: *hit enter to connect to the last component*
Pick end point: *endp of pick the lower elbow*

The result should appear as below.



The next step requires tying this 6” run with the discharge piping performed earlier in the tutorial. Zoom in where the pipe rack changes elevation, the end of the pipe coming from the pumps, and the end of the previous completed piping. Type VIEWFLAT and pick the end of the lower pipe run. This allows adjustment of this pipe. Grip the pipe and drag it past the other pipe setting on the upper pipe rack (the graphic of the pipe, when working in solids, will automatically be adjusted). Type VIEWFLAT and pick the end of the pipe on the upper pipe rack. Grip it and drag it past the pipe just modified on the lower rack. Now the situation exists where elbows can easily be placed using the XY filters.

From the elbow flyout as shown previously, choose the elbow which is in a vertical position. Here, choose the corner option and select the Y location of the pipe coming from the two pumps.

Command: **H90LR**
6" ELL, 90%%D LR S/STD, ASTM A-234 GR WPB
Pick start point or [Corner] <last point>: **C**
Pick corner point: **.Y** of **NEA** to *the pipe coming from the two pumps*
(need XZ): **NEA** to *anywhere along the pipe going to the vessel*
Pick end direction: *select a direction toward the vessel*
Enter direction [Up/Down] <Up>: **D**
Enter distance or [Round distance] <5'-6">: *hit enter, this number will be different*

This should place an elbow pointing down to the pipe coming from the pumps. Next, place an elbow on this piece of pipe. Issue the same elbow function from the same flyout.

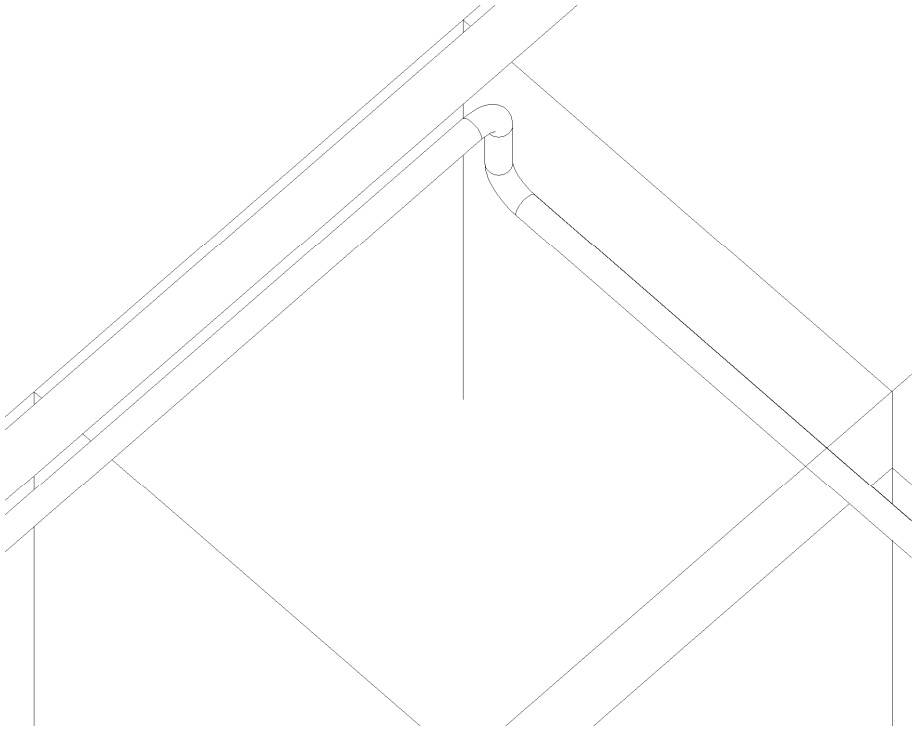
Command: **90LR**
6" ELL, 90%%D LR S/STD, ASTM A-234 GR WPB
Pick start point or [Corner] <last point>: **C**
Pick corner point: **.X** of **NEA** to *the pipe going to the vessel*
(need YZ): **NEA** to *the pipe coming from the pumps*
Pick end direction: *pick a direction toward the pumps*
Enter direction [Up/Down] <Up>: *hit enter to place in a vertical direction*
Enter distance or [Round distance] <2'-11 13/16">: *hit enter, this number will not match*

Erase the two fragments of pipe left over. Draw a piece of 6” pipe between the two new elbows. This can be accomplished without relocating the UCS.

Command: **PIPW**
6" PIPE, S/40 SMLS, ASTM A-106 GR B
Top/Bop/<Start point of pipe>: **ENDP** of *pick the buttweld of the elbow*
Select end point of pipe: **ENDP** of *pick the buttweld of the other elbow*

Command: **ERASE**
Select objects: *select the remains of the upper pipe*
Select objects: *select the remains of the lower pipe*
Select objects: *enter to finish*

The end result should appear as below.



Automatic Elevations

For this tutorial, start a new drawing.

The easiest way to draw a plant or unit is to first create the plans. This method has been used for a long time. Start with a plot plan, establish limits and then produce detailed orthographic drawings. This method is still used by many pipe designers in today's work force.

With the advent of computers, we can create three-dimensional systems that can provide extremely accurate models. With these models, an instant bill of materials can be generated, automatic isometrics can be generated, and even engineering analysis can be performed. Timesavings within a project can be enormous, maybe not on the front end, but certainly on the back end.

As demonstrated in the first section, elevations can be used to update plans. In this section two horizontal pieces of pipe drawn at different elevations will be modified with the automatic elevation capabilities within CADWorx Plant (adding components in a vertical position). This provides a means of creating a full three dimensional model without ever having to change view points as demonstrated in the previous exercises.

The first thing this exercise requires is two pieces of pipe drawn at different elevations. Select 2D double line mode from the settings toolbar (this is probably the default startup mode). Set the main size to 6" and set a specification to 150 (150_M). Draw a 6" pipe at any location within the drawing. After that, draw an elbow pointing up placed at the end of the pipe. From this elbow draw another elbow which, will be pointed down (starting at the corner location of the elbow – use the corner option). Finally, draw another piece of pipe onto the buttweld of this elbow. The sequence is outlined below and should appear similar to the figure below.

Command: **PIPW**
6" PIPE, S/STD SMLS, ASTM A-106 GR B
Pick start point or [TOP/BOP] <last point>: *pick any point on the drawing*
Pick end point: *pick any other point on the drawing*

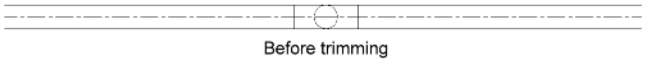
Command: **H90LR**
6" ELL, 90%%D LR S/STD, ASTM A-234 GR WPB
Pick start point or [Corner] <last point>: *hit enter to connect to the last place pipe*
Pick corner direction: *pick a direction parallel with the pipe*
Enter direction [Up/Down] <Up>: *hit enter to accept up*

Command: **H90LR**
6" ELL, 90%%D LR S/STD, ASTM A-234 GR WPB
Pick start point or [Corner] <last point>: **C** *enter for center*
Pick corner point: **ENDP** of *pick corner of last elbow*
Pick end direction: *pick direction parallel with pipe*
Enter direction [Up/Down] <Up>: **D** *this component needs to point down*

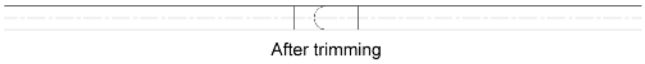
Command: **PIPW**
6" PIPE, S/STD SMLS, ASTM A-106 GR B
Pick start point or [TOP/BOP] <last point>: *hit enter to select last elbow*

Pick end point: *pick direction parallel with previous pipe*

Command: **TRIM**
Current settings: Projection=UCS, Edge=None
Select cutting edges ...
Select objects: *select anywhere on the last elbow*
Select objects: *enter to finish selection*
Select object to trim or shift-select to extend or [Project/Edge/Undo]: *pick circular part of previous elbow*



Before trimming



After trimming

This is a very simple layout all drawn in 2-dimensional space – flat on the piece of paper. There is no elevation or any 3-dimensional properties at all. If we utilize the AutoCAD ID command and select the endpoint of the pipe, it would return 0'-0" (0mm) for the Z coordinate. But this is not the 3-dimensional model that is desired.

From the pulldown, select Pipe>Utility>Elevation>Change. Use this function to elevate the piping. Elevate the left elbow and pipe to an elevation of 1'-8" (508mm). Use the same function to elevate the right side of the piping to an elevation of 10'-0" (3048mm). After elevating both sides, utilize the same function to apply a BOP correction factor (elevate all the piping selected by ½ its diameter). As expected the piping on the left is elevated to 1'-11 5/16" (592mm) and the piping on the right is elevated to 10'-3 5/16" (3132mm).

Command: **CHANGEELEV**
Elevation change [BOP correction/Arbitrary] <Arbitrary>: *enter for default*
Enter world elevation: **1'8 (508mm)**
Select objects: *select pipe and elbow on left side*
Select objects: *enter to finish selection*
Pick base point: **NEA** to *anywhere along the pipe on the left*

Command: **CHANGEELEV**
Elevation change [BOP correction/Arbitrary] <Arbitrary>:
Enter world elevation: **10' (3048mm)**
Select objects: *select pipe and elbow on right side*
Select objects: *enter to finish selection*
Pick base point: **NEA** to *anywhere along the pipe on the right*

Command: **CHANGEELEV**
Elevation change [BOP correction/Arbitrary] <Arbitrary>: **B**
Select objects: *select all*
Select objects: *enter to finish selection*

Instead of using the function as outlined above, the same procedure could have been accomplished by relocating the present elevation with either the UCS or the ELEV command (extreme caution should be used with this command, it needs to be set back to 0 elevation for use with CADWorx Plant within AutoCAD). The CADWorx Plant Point and Shoot UCS also has an elevation option which can be used. One problem associated with drawing pipe in this manner is intersection points are difficult to pick. The "apparent intersection" osnap and XYZ filters are major aids in these situations and all users should become very efficient with using them.

For this exercise, the automatic elevation feature of CADWorx Plant will be used. We will select the two elbows and create an elevation, modify it, and return it to the plan. The first step requires starting the Pipe>Accessory>Plan to Elevation>North function.

Command: **NORTHELEV**
Select objects: *select the two elbows in the center*
Select objects: *enter to finish selection*
Pick reference point: **ENDP** of *the lower pipe*

Here, we need to pick a point which is immediately outside the selected components. This point should be a point which will not change (the elevation will not be modified at this point). This will be a reference point used to return the selected elevation back to the plan.

Component information [ON/Off] <Off>: **ON**

This option turns on the information inside the components. This option can be used to turn information off in an elevation which would only provide a graphical representation. This would be useful in preventing the components in an elevation from appearing in the bill of material.

Draw pipe end baseballs [Yes/No] <No>: *return for no*

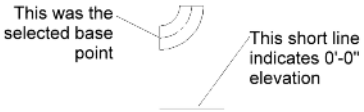
This option will provide the “S” shaped symbol which draftsmen use to indicate that the pipe is perpendicular to the plane in which we are working. Baseballs will not be required.

Delete selection for future replacement [Yes/No] <No>: **Y**

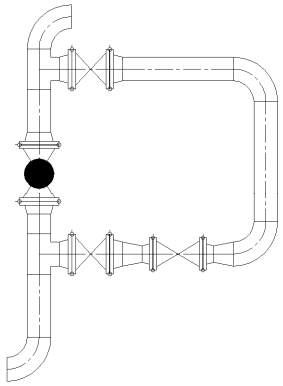
We will delete the components for future replacement. After modifying the components, the area of the plan that requires replacement will be clean.

Pick base point for elevation: *pick anywhere on the drawing to place the elevation*





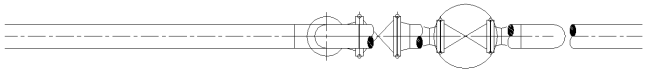
As we can see in the figure above, only the two elbows are available. This is all that was drawn in the plan. Here, add a control loop and then re-insert the modified elevation back into the plan. This provides a means of creating a true 3-dimensional drawing without ever having to change the view point as exercised in the previous sections. Modify the elevation by adding in the components shown in the figure below. The reducing component below is 4”.



As in the the earlier topic, re-insert the elevation back into the plan. There is the option to save the elevation or delete it with the delete selection option. Utilize the same reference point which was used above in the NORTHELEV function.

Command: **ELEV2PLAN**
Select objects: *select all the objects in the elevation*
Select objects: *enter to finish selection*
Pick reference point: **ENDP** of *the same reference point used above*
Component information [ON/Off] <Off>: **ON**
Draw pipe end baseballs [Yes/No] <No>: *enter to accept no*
Delete selection for future replacement [Yes/No] <No>: **Y**
Pick base point and rotation: **ENDP** of *the same reference point*

Shown in the figure below is the end result after some typical drafting practices and symbols are applied. The broken pipe symbols are from the *Pipe>Utility>Break>Manual-Auto* function.

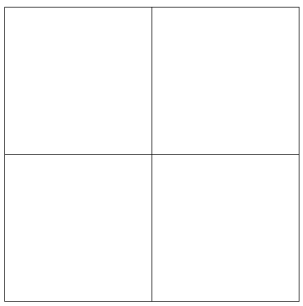


As demonstrated in this exercise the view point has never changed nor has the drawing been displayed or drawn in a 3-dimensional sense. This is a fully functional 3-dimensional drawing. This particular piping system can be sent to the automatic isometric generator or straight to a stress analysis model. A bill of material can generated which will provide an exact count of components. Automatic bolting can be generated and applied. Simply stated, its a 3-dimensional model which looks like it was drawn utilizing 2-dimensional means. Although, as noted in a previous section, a 3-dimensional model developed in model space used in conjunction with paperspace layouts are a far more effective way of creating plans and elevations from models.

Paperspace & CADWorx

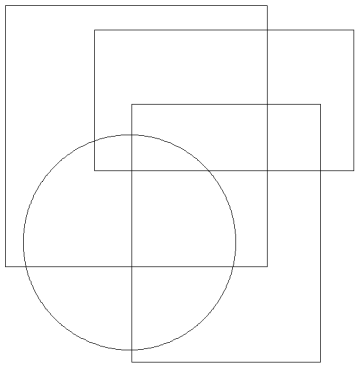
First, let's define model space. Model space is when TILEMODE is on (equals 1). TILEMODE, as AutoCAD explains, is when working in the model space and non-floating view ports are available. These non-floating view ports are created with the AutoCAD VPORTS command. The only way of arranging these view ports are right beside each other. In other words, like a tiled floor. This is the space where all CADWorx Plant orthographics, models, isometrics, etc. are created.

Model space - VPORTS command - TILEMODE =1 (on)



Next is paper space. Paper space is when TILEMODE is off (equals 0). This is where floating view ports are available. These floating view ports are created with the AutoCAD MVIEW command. Unlike the VPORTS command, these view ports are able to overlap each other. This space is used only for annotation and borders. The Mview is only a hole cut through the paper looking into the model space (although it is an actual entity with a layer, size, linetype, etc.).

Paper space - MVIEW command - TILEMODE = 0 (off)

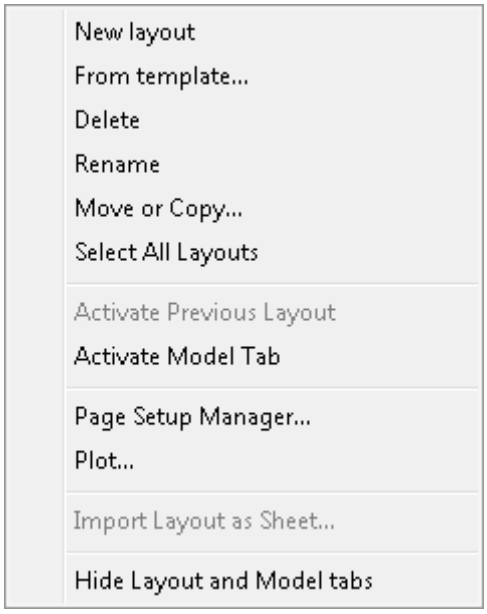


Within paper space, the MVIEW command is available. MVIEW is used to create these floating view ports. Multiple view ports can be created. CADWorx Plant automatically creates the ViewL (installation layer name) layer for use with these view ports. Zooms in these floating view ports can be scaled with the Pipe>Utility>Zoom Factor function and the AutoCAD PROPERTIES command.

The MSPACE command is used to enter one of these view ports created with the MVIEW command. This view port is like cutting a hole in the paper and looking through to the model space. The MSPACE command will show crosshairs while in the view port and a pointer when outside of the view port. While in this view port the UCS icon will appear as the traditional icon (broken pencil if out of plane).

The Layout tab at the bottom of the AutoCAD screen also allows the user to enter paper space. The Model tab as we explained above, is the traditional model space.





Right click on the Layout tab and the user can create multiple layouts and even name these layouts anything desired - Plan, Elevation, Section, etc. The Model button when picked in Model space will switch the environment to paper space. This button should only be used while in paper space to avoid more confusion. When in paper space - it acts as the MSPACE and PSPACE commands or just double click outside the view port and this returns the environment to paper space.

Model space (TILEMODE = 1) is the opposite of paper space (TILEMODE = 0). MSPACE and PSPACE commands can only be used in paper space. MSPACE (model space) is used when entering a view port created with the MVIEW command and PSPACE (paper space) is used when exiting a view port created with the MVIEW command.

Now back to the tutorial. Open the TUTOR9.DWG (TUTOR9M.DWG). From the pulldown Pipe>Setup and then select the *Borders* button; toggle *Predefined*; then toggle *Insert in paper space*; then select the 34x22 Ansi D (841x594 ISO-1) border from the list box on the right. Pick *OK* for all the remaining dialogs and pick *OK* for the title box attributes. This places the border in paper space. The next step is to create view ports to view the model with. Set the ViewL layer current and on.

Command: **MVIEW**
Specify corner of viewport or
[ON/OFF/Fit/Hideplot/Lock/Object/Polygonal/Restore/2/3/4] <Fit>: **4**
Specify first corner or [Fit] <Fit>: **INT** of *pick the lower left corner of the inside border area*
Specify opposite corner: **INT** of *pick the upper right corner of the inside border area*
Regenerating model.

This created 4 Mviews. These Mviews will be used to create a plan and elevations of the pumps. The upper right will be used for an isometric view of the pumps. The upper left will be used for a plan. The lower left will be a front elevation and the lower right will be a side elevation.

Let's start with the upper right isometric. Currently the four views show four isometrics. Double click on the Paper button (PSPACE) at the bottom. At this point, pick the upper right view port created with the MVIEW command. This should provide crosshairs in the view. The remaining view ports should not be accessible unless a pick is performed inside of one. While in the view port, issue the DDVPOINT command and enter 315 and 30. This should leave the view in an isometric orientation the same as shown in the figure below. Now issue the ZOOM command and pick a window around the pumps. This window should be similar to below. This view port will not be scaled.

Pull the crosshairs out of the present view port and pick the upper left view port. This will be the plan view. The crosshairs will change to a regular mouse pointer and will require a single pick to activate this view port created with the MVIEW command. This is all performed while the *Model* button is activated (MSPACE). Use the AutoCAD PLAN command and utilize the *Current UCS* option. With the upper left view port activated, choose the Pipe>Utility>Zoom Factor>Architect and select 3/8 = 1'-0" (metric at 1:20). This function will prompt for a center point. Pick between the two pumps. If necessary, utilize the PAN command within the Mview. This is accomplished with the standard AutoCAD PAN command. The two pumps should be in a plan view ready to apply annotations and dimensions.

With the *Model* button activated, pull the crosshairs over to the lower left view port (crosshairs will change to pointer). This view port will need to be activated with a single pick of the pointer. After activating this view port, issue the DDVPOINT command and enter 270 an 0 degrees. This will provide a front elevation of the pumps. Once again, pick the Pipe>Utility>Zoom Factor>Architect, and select 3/8" = 1'-0" (metric at 1:20). When asked to pick a center, again select between the two pumps. In this view the UCS icon appears as a broken pencil inside of a box. This must be changed in order to PAN the view within the view port. Type VIEWNORTH to relocate the UCS and just hit enter. This positions the UCS in plane with the view. To add annotations or dimensions within the view port would require this adjustment.

The last view port will become a side view. Enter the view port as mentioned before. Issue the DDVPOINT command and select 0 and 0. This will adjust the view port showing the side of the pumps. Issue the Pipe>Utility>Zoom Factor function and pick the center. Here the VIEWWEST command will need to be set for annotation and dimensioning. Pan as required.

The dimensioning of these view ports can be performed in the view port itself or within paper space. In the view port itself, the AutoCAD setvar DIMSCALE should be set to 0. <Produc_Name> and AutoCAD will automatically size all the text, arrow heads, etc. The setvar DIMASSOC will handle all the scaling and sizing of the dimensions. This setvar will also control the way the dimension are updated when objects are changed in the model space environment. Please refer to the AutoCAD documentation for more information on this setvar.

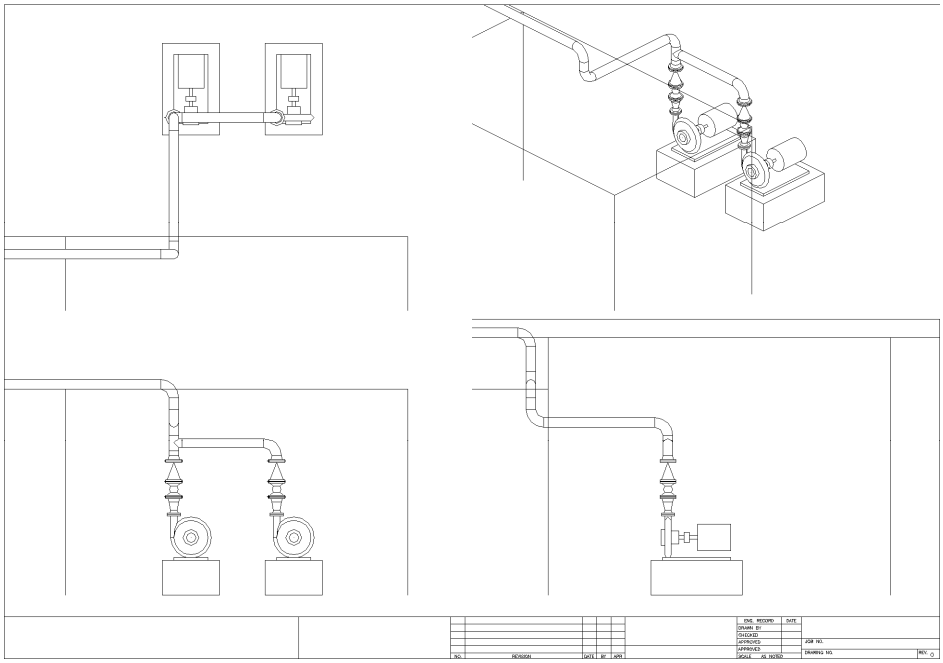
The last step requires the alignment of the views within the view ports. The best way to accomplish this feat is to issue the AutoCAD MVSETUP command. This can be located under the Pipe>Utility>MVSetup. This command has an excellent Align option which will work both in the horizontal and vertical. The following sequence of commands will demonstrate this command.

Command: **MVSETUP**
Enter an option [Align/Create/Scale viewports/Options/Title block/Undo]: **A**

Enter an option [Angled/Horizontal/Vertical alignment/Rotate view/Undo]: **H**
Specify basepoint: **INT** of *column on the pipe rack in the lower right view port*
Specify point in viewport to be panned: **INT** of *column of the pipe rack in the lower left view port*

Enter an option [Angled/Horizontal/Vertical alignment/Rotate view/Undo]: **V**
Specify basepoint: **INT** of *upper pipe rack in the upper left view port*
Specify point in viewport to be panned: **INT** of *upper pipe rack in the lower left view port*

Enter an option [Angled/Horizontal/Vertical alignment/Rotate view/Undo]: *hit enter to finish*
Enter an option [Align/Create/Scale viewports/Options/Title block/Undo]: *hit another enter*



Note the *Create* option in the MVSETUP command. This command can be used to automatically generate all the view ports and view adjustments for each one. This tutorial demonstrated the manual method for purposes of understanding the basic principle. This completes the views of the previous model which was drawn in the beginning of this tutorial. They could have been set to different scales and more of the drawing could have been shown.

The most efficient way to create profiles is with the AutoCAD SOLPROF command. Using solids and the SOLPROF command provides a nice way of creating silhouettes of these items in view ports. Solids can also be plotted with hidden line removal. The figure above shows the four completed views with hidden line removal. To plot this drawing, freeze the ViewL layer, set the Paper button active (PSPACE) and issue the MVIEW command. This command will have an *Hideplot* option. This option will request an *On* or *Off*. Turn *Hideplot* on. It will then request selection of the MVIEW(s) to plot hidden. It is not necessary to hide lines within the plot dialog.

For creating a manual isometric, open TUTOR2.DWG (TUTOR2M.DWG). This is the isometric model created earlier in the tutorial. This drawing already has a border in paper space. After opening the drawing, the status bar indicates that the drawing is in model space. Single click on the Layout tab, and place the drawing in paper space. This environment shows the border. Now, all that is required is to activate an view port with the MVIEW command. Set the ViewL layer current prior to doing this exercise.

Command: **MVIEW**
Specify corner of viewport or
[ON/OFF/Fit/Hideplot/Lock/Object/Polygonal/Restore/2/3/4] <Fit>: **INT** of *the lower left of the available border*
Specify opposite corner: **MID** of *the upper available border*
Regenerating model.

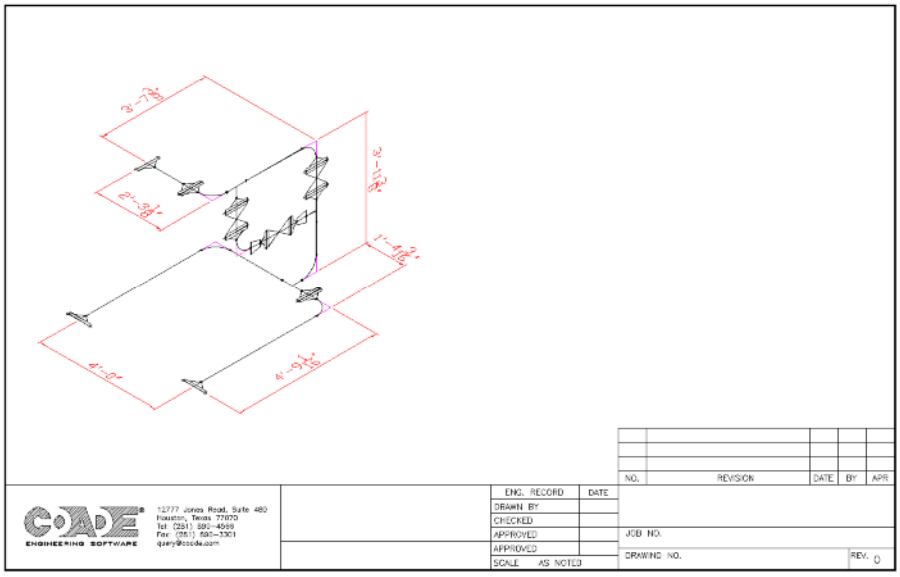
This cuts a hole through the paper showing the model in the background. Enter the view port by double clicking on the *Paper* button on the status bar or double click on an area over the view port. The *Paper* button changes to *Model*. For isometric purposes, utilize the DDVPOINT command to place the model in line with a paper space isometric rotation (the model needs to line up with the AutoCAD ISOPLANE command). To accomplish this, place the *From X* to either 45, 135, 225, or 315 but the from *XY plane* must be set to 35.3 exactly. This would be the same as issuing one of the views from the AutoCAD pulldown View>3D View - Southeast, Southwest, Northeast, or Northwest views. Utilize the ZOOM and PAN commands to better position the model within the view port. After you have finished positioning the view, lock it with the AutoCAD PROPERTIES command or the CADWorx Plant ZOOMLOCK command.

With the AutoCAD setvar DIMSCALE set to 0, dimensioning is easily applied. The only requirement is to locate the CADWorx Plant's Point and Shoot UCS to the plane which requires dimensioning (an elbow creates a plane). The *3 point* and *Object* option of the UCS command is also very useful. Use the *Object* option to pick an existing dimension to align the UCS. Many of these options no longer appear on the command line, although they are still available - 3P and OB.

Place a few dimensions as shown in the next figure. Increase the setvar DIMTXT variable to 3/16" to allow the dimensions to look like they are 1/8" due to being out of the plane (30 degrees out of view). Placing dimensions at the top will require moving the UCS to the desired position. For example, type VIEWFLAT and pick a corner at one of the upper elbows.

Mode convert the model to isometric after setting the Pipe>Utility>Drawing Control>Elbow Centerline on. This provides corners on the elbows to dimension to.

For dimensions that seem upside down, the *Ztext* option of the CADWorx Plant's Point and Shoot provides a way of placing dimensions correctly. Type VIEWFLAT and the enter Z for the *Ztext* option. Change to VIEWNORTH for dimensioning the vertical leg of the model. Pick a point at the lower elbow.



An easier way to dimension will be discussed in the Auto Dimensioning part of this tutorial. Save this model for this section and the Bill of Material section.

Automatic Isometrics

Isometrics can be automatically generated with CADWorx Plant. Open TUTOR9.DWG (TUTOR9M.DWG). Enter the PLAN command and use the *Current UCS* option. The drawing is now ready to create an isometric.

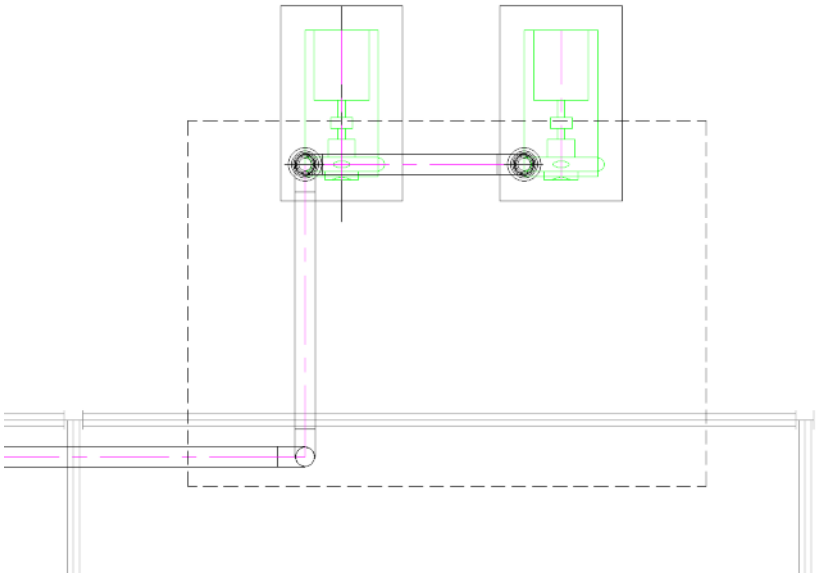
The first time the Pipe>Accessory>Automatic Isometric>ISO Out function (command line ISOOUT) is used within a drawing session, an isometric configuration dialog will appear. The second time this function is requested, the drawing environment will contain information regarding this configuration and this dialog will not appear again unless requested. Configuration changes required after the initial ISOOUT, during the same drawing session, will require the Pipe>Accessory>Automatic Isometric>Config function to be run - ISOCFG. For further discussion regarding the configuration of the automatic isometric, refer to configuration under the automatic isometric section of the CADWorx Plant manual.

The two most important rules about the isometric configuration are as follows.

1. Units within the Startup section of the Setup dialog have to match the border drawings. In other words, if the ANSI border drawings (_ANSI_BI.DWG, _ANSI_C.DWG, etc.) are used, the drawing prototype needs to be set to English/Inch (SystemMeasure = 1) and the Imperial.dwt should be used as a drawing template. If the ISO border drawings (_ISO_A3I.DWG, _ISO_A2.DWG, etc.) are used, the drawing prototype must be set to Metric/Inch or Metric/Metric (SystemMeasure = 0 or 2) and the Metric.dwt should be used as a drawing template. These startup units must be set prior to beginning a new drawing. This is discussed in more detail in the manual under Startup Defaults.
2. In the same content, anything that requires a size will need to be set correctly. Do not leave the TEXTSIZE set at 0.125 when using metric border drawings. This 0.125 text will be so small, it will be almost invisible. It should be set to 3.0 for metric. The correct configuration file must be used (run SETUP and *Edit config file* button)

CADWorx Plant supplies many border template drawings. These drawing can be used or any other border can be used. For border preparation, refer to the automatic isometric section of the CADWorx Plant manual. This will explain how to set up user border drawings.

Create an isometric of the discharge piping from the pumps. Stop selection at the location where the piping turns flat onto the rack. From the pulldown, select the Pipe>Accessory>Automatic Isometric>ISO Out function. When prompted for the file name, enter any name without the extension. After the file name entry, the configuration dialog will appear. Refer to the configuration under the automatic isometric section in the CADWorx Plant manual for more information regarding this dialog.



Command: **ISOOUT**
Enter north direction <90>: *hit enter to accept this orientation*

The first prompt that appears requests the north direction expressed in degrees. If a north arrow is located in the drawing which has the name N1.DWG, this function will automatically inspect this block and determine the required orientation. Any block can be used for a north arrow if its name is N1.DWG.

Enter an option [Database/Line number/Select components] <Select components>: *hit enter to select*
Select objects: *select the complete discharge pipe by window*
Select objects: *enter to end the selection*

The next prompt indicates the total number of selected entities and the exact amount of CADWorx Plant components.

Exported 27 component(s) to <INSTALL-DIR>\Tutorial\Tutor9.pmi...

At this point, a data file has been written which will generate an isometric. Before generating the isometric, let's examine the data file which was just written. Use Window's Notepad to open this data file. It will have an extension of PMI.

```
#$ JOB SPECIFIC
       3.0000
1    0--MM, 1--ENGLISH, 2--MM (WITH ENGLISH NOMINAL)

#$ AUTO ISO
<INSTALL-DIR>\DEFAULT_I.ISO
Tutor9.dwg
1.5708

#$ ELEMENTS
1                  18.9700      78.0000       0.2800
0.0000 1806
       6.0000       6.6250       0.0000       0.0000 3
       0.0000       0.0000       0.0000       0.0000
    1180.0000     939.0000     119.8125    1180.0000
861.0000     119.8125
    1180.0000     939.0000     119.8125    1180.0000
939.0000     119.8125
    1180.0000     900.0000     119.8125    1180.0000
939.0000     119.8125
6"
PIPE
PIPE, S/40 SMLS, ASTM A-106 GR B

(("76.35.38.280" "3DS" "1T" 2)("_150" "S40" "PIP_W")("PIPW"
6.0000 6.6250)"(0.0 0.0 0.0 0.0)")

.
.
many other components
.
.

#$ END
```

At the top of the data file, notice that the system measure was set to English – the configuration file was <INSTALL-DIR>\DEFAULT_I.ISO – the originating drawing file was TUTOR9.DWG (TUTOR9M.DWG) – and the north direction was 1.5708 radians which converts to 90 degrees. The most important item about this PMI (piping module isometric) file, is that its associated with the DEFAULT_I.ISO configuration file. If this file is moved, its name and path need to corrected in this file. This file can be created and located in a single project directory. There could be several project directories with a configuration file for each. Examine the configuration file.

```
#$ CONFIG
DIM              1
BLOCK            1
BOM              1
MARK             0
TAG              45
ISO-SNAP         1
CUTS             1
F-WIDTH          100
S-WIDTH          0.02
ISO-SCALE        30
NODES            1
CTR-LINE         1
BOLT             1
DISTANCE         AUTO
TEXTSIZE         0.125
FRACTION         1

FLANGE           1
DWG              <INSTALL-DIR>\SUPPORT\_ANSI_BI.DWG
#$ DWG_COUNT
1099-x-0
16
#$ ATTRIBUTE
CUSTOMER         COADE
LINE_NO          #line_number
JOB_NO           1099
DWG_NO           #dwg_number
REV_NO           0
NAME             #name
DATE             #date
SCALE            NONE
TAG              001-A23
DWG_REF          #dwg_reference
LINE_SIZE        #line_size
WELD_PROC        WPC-101
INSULATION       2-CAL
REMARKS          NONE
MATERIAL         #pipe_material
SPEC             #specification
SCHEDULE         #schedule
RATING           #rating
TEMP             245 F
PRESS            210 PSI
#$ END
```

This file contains all the settings which are in the configuration dialog. Dimensions are on – insert a border drawing – create a bill of material – etc. The line that causes the most trouble is the border name. If a border file is moved to another location, then the AUTOISO function will report that there are problems in this file and just abort.

Now, start a new drawing and re-import the isometric file. Use a new drawing to import an isometric (if not a new drawing an error will occur with a message indicating that it is not a new drawing). Make sure the correct drawing template file is used. Most people shut off the startup dialog (OPTIONS command under the *System* tab) which disables the ability to select the correct template. Close this dialog down only if you know what is going on with the registry and the previous use of drawing templates. Refer to the AutoCAD documentation for explanations on the start up dialog and it purpose. The developers of this program use the start up dialog with the template option - always.

From the pulldown, select the Pipe>Accessory>Automatic Isometric>ISO In. This provides a file selection dialog. Enter the file name which was exported above.

Command: **AUTOISO**
Drawing preparation...OK

This indicates that all preliminary startup processes were completed successfully.

Reading and listing component number: 27...finished

Component number indicates how many components were read in and will be used in the automatic isometric. If bolts were applied in the orthographics and re-applied in the configuration dialog, there will be double quantities of bolts in the bill of material.

Select the desired view...

Select desired view...
*** Southeast view shown ***
Enter an option [eXit/southEast/Northeast/northWest/Southwest] <next view>: *return for the next view*
*** Southwest view shown ***
Enter an option [eXit/southEast/Northeast/northWest/Southwest] <next view>: *return for the next view*
*** Northwest view shown ***
Enter an option [eXit/southEast/Northeast/northWest/Southwest] <next view>: **X** *to exit from the view selection*

Enter an option [Pan/+/-/Accept] <Accept>: **+** *this is used to enlarge, decrease or pan the view*
Enter an option [Pan/+/-/Accept] <Accept>: **+**
Enter an option [Pan/+/-/Accept] <Accept>: *hit enter when finished*

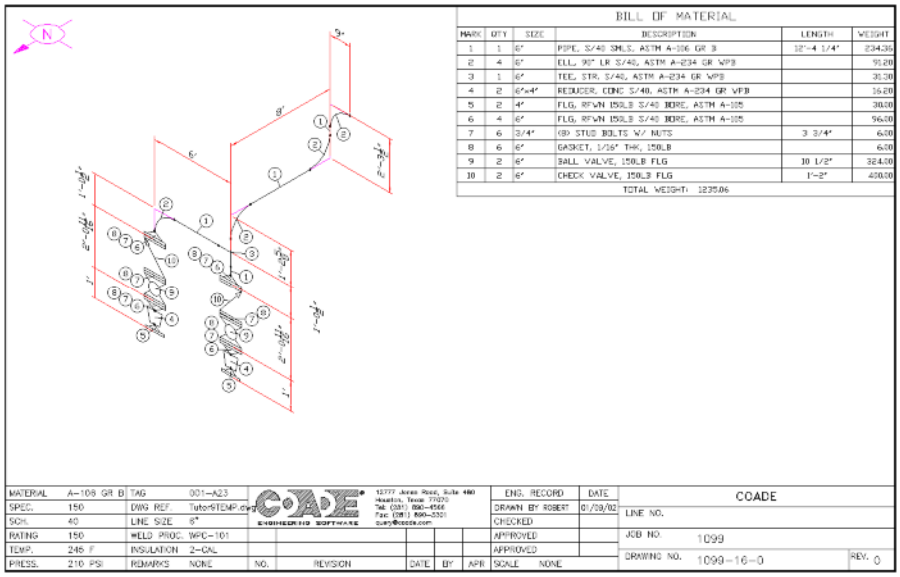
Converting components to Isometric single line...finished
Auto Dimensioning...finished
Auto Bolts...finished
Generating Bill of Material...finished
Paperspace zoom lock toggled on...

The pipe lengths in the bill of material are available in one total length or individual lengths. All 4" pipe may be grouped into one bill of material item. The 2" pipe would be grouped the same way. This method is accomplished by using total lengths in the configuration. For different pieces of pipe listed on separate rows with their own individual lengths requires the individual cut lengths method. This last method would be used primarily for fabrication purposes. All these controls are available in the configuration dialog.

The marks can be set to automatic placement, manual placement or off. Automatic placement is the preferred method as you can see by this example. All of these features are automatically set and controlled in the configuration dialog.

In the finished isometric, there is a single view port which contains the isometric model in the model space. The dimensions are also in the model space. The isometric is orientated either 45, 135, 225, or 315 degrees from the X axis. It is 35.3 degrees from the XY plane. 35.3 is a very special number because it aligns the model exactly with the AutoCAD ISOPLANE command as mentioned previously. This is very useful for placing text in paper space. From the pulldown Pipe>Text>ISO Text, different text styles are available (0, 30 and 330 styles).

The view port can be modified. The DDVPOINT command can be used to re-orient the model if required. Some isometrics can have several 45 degree runs which will appear as straight lines within a drawing. This type of view port can be rotated slightly (5 or 10 degrees) with the DDVPOINT command, thus allowing for a better view. The only problem with this procedure is that it requires re-applying the automatic dimensioning and bill of material functions. Shown below is the finished isometric without any modification.



CAESAR II Importing and Exporting

The scope of this tutorial will be summarized in major steps as outlined below. Following the summarization, exact steps required to produce the results will be outlined. Open TUTOR11.DWG (TUTOR11M.DWG) to use with this section of the tutorial. CAESAR II is required to perform this section of tutorial.

Note: This section of the tutorial requires the user have CAESAR II installed on their system.

The first step is exporting the drawing information into a CAESAR II input file. Start CAESAR II and open the input file created. The file created by CADWorx Plant will be the same input "A" file required by CAESAR II. In CAESAR II, the model will then be modified with a small expansion loop, anchors and hangers, and last, the pressure and temperature will be added. It is recommended that the job be error-checked at this point to catch any mistakes. After this is finished, the model geometry needs to be saved.

Next the model will be imported back to CADWorx Plant. The same CAESAR II input file created above will be used. When importing the input file, the drawing will update or add the components that were modified in CAESAR II with a skeleton geometry that will be clearly visible. Components removed in CAESAR II will be totally deleted. A small modification will then be made in CADWorx Plant. The model will be exported back to CAESAR II as the same input file. Once in CAESAR II, the same input file will be viewed to show what modifications have been made and the calculations will be run.

This last step will provide a fully annotated stress isometric. Selecting the Pipe>Accessory>CAESAR II>Stress ISO pulldown, will present a series of dialogs. These dialogs will provide choices of border drawings and many other options. Another option provides the choice of views for the isometric. After the isometric has been created and dimensioned, the option to annotate the drawing is available by accessing Pipe>Accessory>CAESAR II>Annotation option from the pulldown.

Steps to complete these tasks are described below.

Step 1: Create a CAESAR II input file with CADWorx/PIPE.

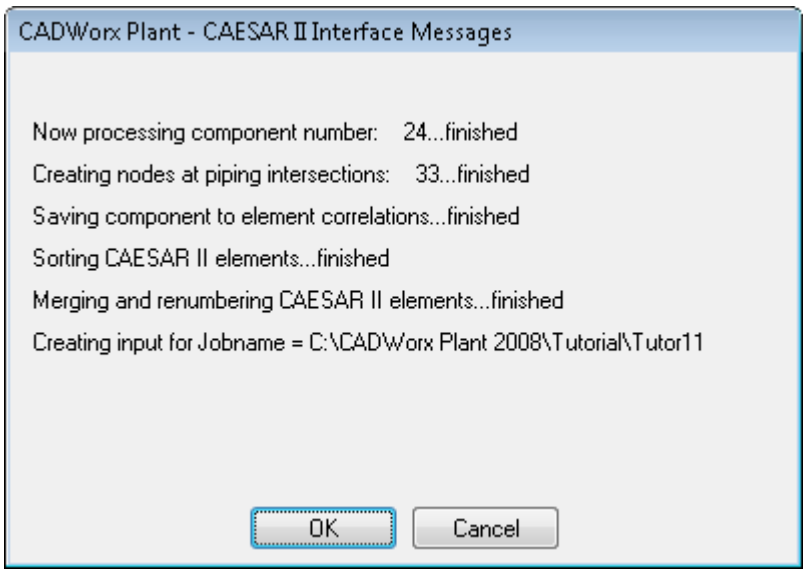
From the first pulldown in CADWorx Plant use the Files>Open and select the TUTOR11.DWG (TUTOR11M.DWG). This will present a common dialog for selecting files. The drawing is a typical system built in CADWorx Plant which is to be sent to CAESAR II.

From the pulldown menu, select Pipe>Accessory>CAESAR II>System Out. After being presented the CAESAR II Input file dialog, use the default name TUTOR11 for the CAESAR II file in the file edit box. Respond to the following AutoCAD command line prompts as indicated.

Command: **C2OUT**
Enter an option [Database/Line number/Select components] <Select components>:
Select objects: *select all*
Select objects: *enter again to finish selection*

This next line allows the user to control noding within CAESAR II. Please refer to the C2OUT command for more information on this subject.

Provide start locations [Yes/No] <No>: *enter* **N** *for no*
Writing component number: 24...finished



The piping layout has been successfully transferred to a CAESAR II input file named TUTOR11._A.

Step 2: Utilize CAESAR II to make modifications, define operating conditions, and add hanger and anchors.

Start CAESAR II and open the CAESAR II input file which has just been created. Under the File>Open pulldown, browse for the appropriate file. Select the Input>Piping pulldown which will provide the job's spreadsheets.

Enter [Alt] P or pick the Plot pulldown and from this graphics screen choose the Options>Node Numbers pulldown. This will provide the graphics screen showing the same piping configuration as was in CADWorx Plant. Notice the arrangement of the node numbering system. This is exactly the way an experienced stress analyst would number the system. Also notice how continuous rigid elements have been merged together, and tees and bends have been modeled as though entered by a stress analyst.

From the graphics screen, pick the View>Spreadsheet pulldown and enter to the job's spreadsheets. We now want to make some typical modifications, such as adding restraints, loops, operating conditions, etc. On the first spreadsheet, double click on the Restraints toggle (middle-top of this screen). This provides the restraints section on the right. Place the cursor in the Node: edit box and enter 10. Utilizing the flyout on the Type: edit box, place an ANC (anchor) for this location.

Page down (hit the page down key) to the next spreadsheet. Double click on the Restraints toggle and enter 30 in the Node: edit box. Enter +Y in the Type: edit box.

At this point, a new 5'-0" (1524mm) expansion loop will be added to the line. Page down several spreadsheets until element 50-60 is showing. Pick in the DY: edit box and change the length from 10'-0" to 15'-0" (3048mm to 4572mm).

This new loop needs a hanger. Page down to element 60-70 and then enter a single node to place a new node for the hanger. Enter 62 as the new node number. Enter 3'-0" (914mm) as the distance from node 60. The only entries required are the new node number and the distance. When finished with this dialog, pick the OK button.

Now add the hanger at node 62. Double click on the Hanger toggle and enter 62 for the Node: number for the hanger's location. Either edit the hanger information or pick the OK button on the hanger data dialog.

Make the next node 64, which requires a bend. Page down to element 62-70, and break this element. From the Model>Break pulldown, enter a single node 64 at a distance of 3'-0" (914mm). Pick the OK button to end this dialog. Now double click on the Bend toggle to place a bend at the end of this location. This builds a long radius bend.

For the last element of the loop, utilize the Edit>Insert pulldown option. Then pick the After option to place another element after element 62-64. Enter 66 in the To: edit box and enter -5'-0" (-1524mm) in the DY: edit box (upper-left of the screen). This will also have a bend, so double click on the Bend toggle.

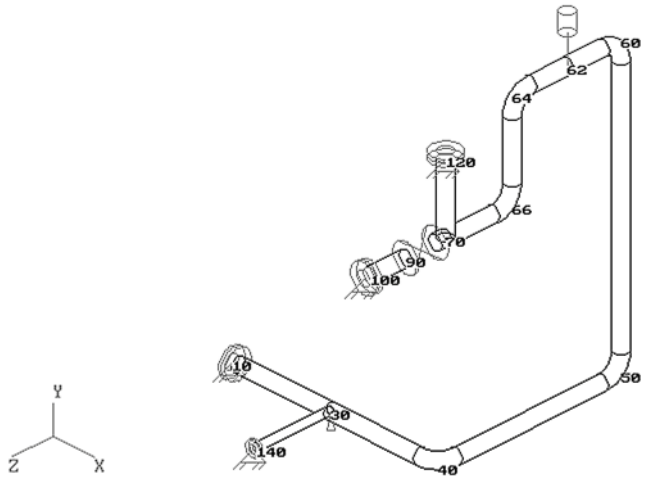
Now, connect the remaining system to new node 66. Page down to the 64 to 70 spreadsheet. Change the node 64 to the new node 66. This ties the loop together.

Anchors will be required at all the remaining end points. Page down to the spreadsheet for node 100 to 110 and double click on Restraints toggle (middle-top of the screen) and place 110 in the Node: edit box and select an ANC for the Type: edit box. Page down to nodes 120 to 130 and repeat the same procedure above for adding an anchor to node 130. Then do the same for node 150 at nodes 140 to 150.

The last requirement is setting the operating conditions. Hit the [Home] key to enter to the first spreadsheet. Double click on the Allowable Stress toggle (middle-center of screen). Enter 20,000 PSI (120,000 Kpa) for both SC and SH1. Utilizing the Code: flyout (lower-right of the screen), enter B31.1.

Enter 250 F (175 C) in the Temp 1: edit box (lower left of the screen). Enter 300 PSI (1000 Kpa) in the Pressure 1: edit box (lower left). Type [Alt] P for plotting, N for nodes and V for volume to view the new piping system. From the Options pulldown chose Restraints, Anchors, and Hangers.

The CAESAR job should appear as below.



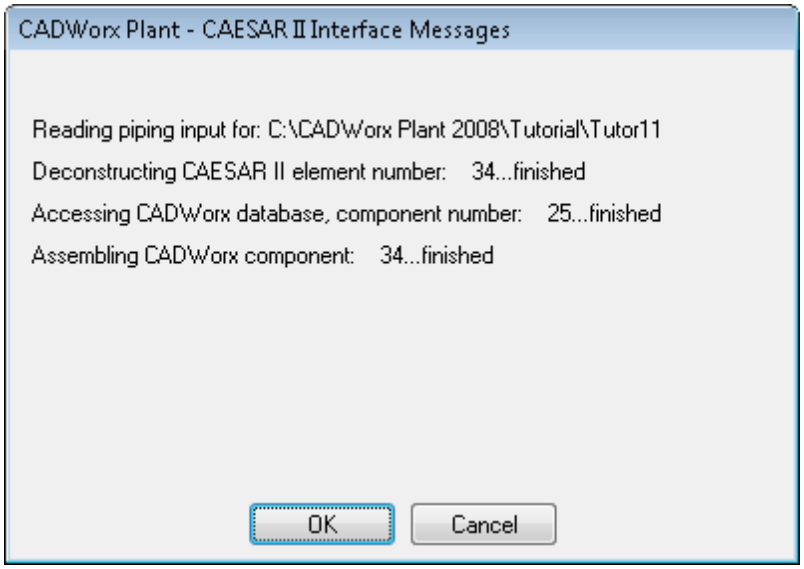
The last requirement is to error check the job and save the input file. Hit [Esc] while in the spreadsheets to enter to the PIPING QUIT MENU. Then enter on the Start Run (or enter "1"). Hit enter on all the remaining status screens. There should be 0 errors, 0 warnings, and 2 notes generated. Any other result indicates the user made an error. Hit [Esc] to enter to the main menu.

Step 3: Starting CADWorx Plant to import the modified CAESAR II input file.

Next, import the input file that was modified by the stress analyst in order to update the piping drawing. Then minor modifications will be required which should then be sent back to CAESAR II one more time.

From the pulldown menu, Pipe>Accessory>CAESAR II>System In. From the CAESAR II input file dialog, change to the directory where the input file is located and pick TUTOR11 from the list in the list box. The next prompt will ask for a default specification. This specification will not be used here but is always requested by the system as a backup default.

Command: **C2IN**
Please set a default Specification <150>: *enter to accept 150* (150_M for metric)
Prepared E:\CADWORX PIPE 2002\SPEC\150.spc specification...



After reading the input file, the next prompt requests the level of intervention required. Please refer C2IN (see page 194) of the users manual for further information regarding these options.

Enter an option [Full intervention/Partial/None] <None>:
Working component number: 34

Note: Notice that the areas changed by the CAESAR II analyst are readily apparent.

From the pulldown menu, select Pipe>Accessory>Mode Convert>3D Solids. The following command line prompts will convert the entire model to 3D solids.

Command: **CONVERTSOLID**
3D solids conversion...
Select objects: **ALL**
Select objects: *enter to finish selection process*

Relocate the UCS, with CADWorx Plant's enhanced UCS. Type VIEWNORTH and use an AutoCAD object snap "endpoint of" to reposition the UCS. Select the lower end point of the vertical 8" pipe to the far right.

Command: **VIEWNORTH**
[Xtext/Ztext/Elevation/Rotate/COordinate/CLip] <Pick point, or enter>: **ENDP** of *the lower end of vertical pipe*

From the pulldown menu, select Pipe>Toolbars>Buttweld to provide the toolbar required to pick the tee as demonstrated in the previous part of this tutorial. Respond to the following AutoCAD command line prompts.

Command: **TESW**
8" TEE, STR. S/STD, ASTM A-234 GR WPB
Pick start point or [Branch/Center] <last point>: **ENDP** of *the bottom buttweld of the upper right elbow*
Pick main end direction: **@0,-100** *or pick direction*
Pick branch end direction: **@100,0** *or pick direction*

From the same toolbar select the pipe component and respond to the following AutoCAD command line prompts.

Command: **PIPW**
8" PIPE, S/STD SMLS, ASTM A-106 GR B
Pick start point or [TOP/BOP] <last point>: **ENDP** of *pick the branch connection of the previous drawn tee*
Pick end point: **@36,0 (@914,0)**

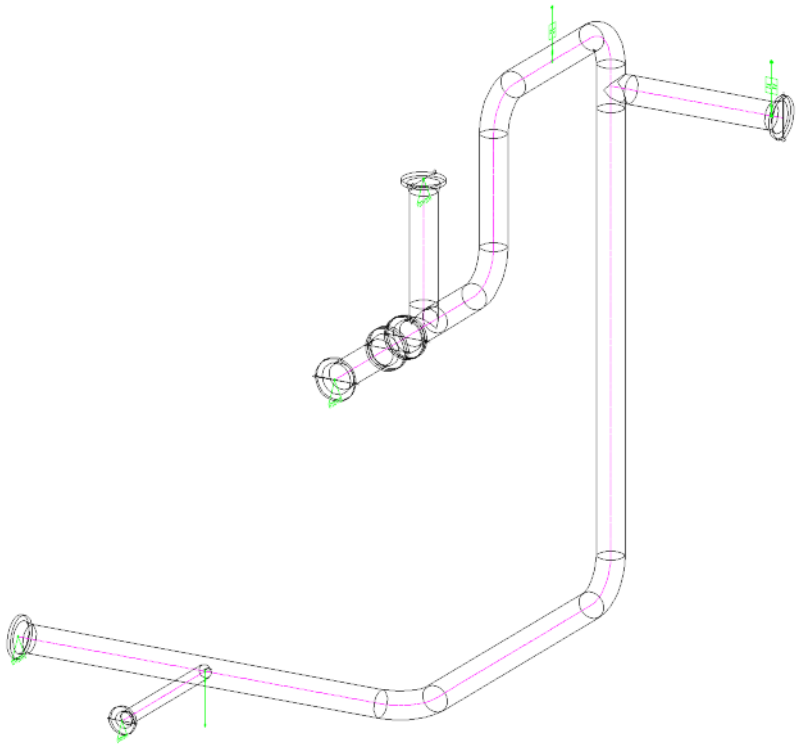
From the Pipe>Toolbars>Flanges toolbar, select the weld neck flange and respond to the following AutoCAD command line prompts.

Command: **FLGW**
8" FLG, RFWN 150LB S/STD BORE, ASTM A-105
Pick start point or [Face end] <last point>: *enter to accept the last point*
Pick direction: **@100,0** *picking a direction is also allowed*
8" GASKET, 1/8" THK, 150LB
Automatically placed...

Adding a hanger to the end of the new section is required for this tutorial. From the pulldown menu, select Pipe>Toolbars>Restraints to provide the restraints toolbar. Pick the spring hanger button and follow the command line prompts as indicated below.

Command: **SPRING**
Pick location on component for restraint: **ENDP** of *the right end of the new horizontal pipe.*

Enter or pick rotation]: **@0,100** *or pick upward direction*
Enter depth: **17 (500)**
Enter length or diameter <1'-5">: *enter to accept default*
Enter assembly identification number <17.00-3 SPRING HANGER>: *enter to accept default*



From the pulldown menu, select Pipe>Accessory>CAESAR II>System Out. From the CAESAR II input file dialog, type in a name for the CAESAR II input file (TUTOR11 or accept the default) in the file edit box. After selecting OK, respond to the following AutoCAD command line prompts as indicated.

Command: **C2OUT**
Enter an option [Database/Line number/Select components] <Select components>: *enter for selection*
Select objects: **ALL**
Select objects: *enter to finish selection*
Provide start locations [Yes/No] <No>: *enter to accept no*
Writing component number: 39...finished

The piping layout has been successfully transferred to CAESAR II input file format again.

Step 4: Start CAESAR II and rerun the job.

Start CAESAR II and open the CAESAR II input file which has just been created. Under File>Open pulldown, browse for the appropriate file. Select the Input>Piping pulldown which will provide the job's spreadsheets.

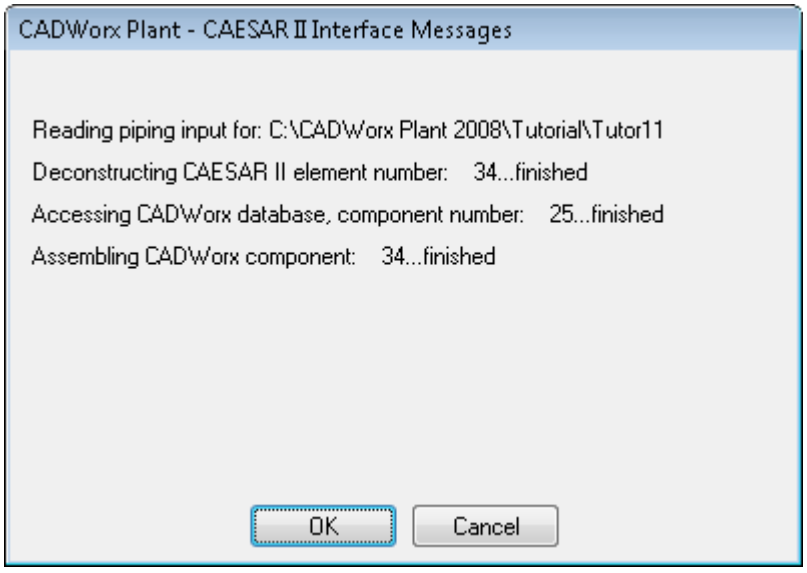
Type in [Alt] P (for plot), V (for volume) and N (for nodes) to provide the graphics screen showing the modified piping configuration as was in CADWorx/PIPE, along with the new hanger. From the Options pulldown chose Restraints, Anchors and Hangers.

Analyze the job, and exit CAESAR II. Pick the X in the upper right corner while in the spreadsheets and select the Start Batch Run option. This will run the job and place the user at the output screen. View the result as desired. Exit this menu to return to the main menu.

Step 5: Utilize CADWorx Plant for the creation of a stress isometric.

Start a new drawing. From the pulldown menu, select Pipe>Accessory>CAESAR II>Stress ISO. In the CAESAR II input file dialog, type in the name (TUTOR11) or pick it from the list box. Respond to the following AutoCAD command line prompts.

Command: **C2ISO**
Drawing preparation...OK
Select CAESAR II input file...
Please set a default Specification <>: **150 (150_M for metric)**
Prepared E:\CADWORX PIPE 2002\SPEC\150.spc specification...


CADWorx Plant - CAESAR II Interface Messages
Reading piping input for: C:\CADWorx Plant 2008\Tutorial\Tutor11
Deconstructing CAESAR II element number: 34...finished
Accessing CADWorx database, component number: 25...finished
Assembling CADWorx component: 34...finished
OK
Cancel

Select border drawing...finished

From the Border Drawing dialog either type in _ANSI_C.DWG (_ISO_A2.DWG for metric) or use the list box to locate and pick the border drawing. When finished, pick the OK button. The next dialog is the Enter Attributes dialog for the border drawing (these attributes are contained in the border drawing). Fill these in or just pick OK and fill them in later with the DDATTE command in AutoCAD.

The next dialog is the Stress Isometric Configuration dialog. This dialog is used to turn dimensioning on or off, control fitting widths, choose the scale of the isometric, and many other options.

Pick the Please Pick Mview button. This is a mandatory button which will not let the dialog exit unless it is selected. It requires that two points are picked as indicated below.

Pick first corner: *pick the lower left corner of the available drawing area*
Pick second corner: *pick the upper right corner of the available drawing area*

Pick the OK button when finish selecting the Mview. At this point the program will automatically create an Mview and begin to draw the components into the new drawing. After this is finished, it will prompt for a view as indicated by the following command line prompts.

Reading and listing component number: 39
Select desired view...

*** Southeast view shown ***
Enter an option [eXit/southEast/Northeast/northWest/Southwest] <next view>: **X** *for exiting*
Enter an option [Pan/+/-/Accept] <Accept>: *increase view with +, pan or decrease view, enter to accept*

Now the stress isometric will automatically be drawn.

Converting components to Isometric single line...finished
Auto Dimensioning...finished
Pick stress ISO arrow location: *pick point in the lower right corner*
Paperspace zoom lock toggled on...

Important note: The stress ISO arrow is a very important item within the drawing. It contains all the information regarding the scaling factor used in the drawing. If it is omitted, the annotation routines will appear to function improperly unless the scaling has been set to 100 percent.

The basic stress isometric has been created and it now requires annotation. Next, develop annotation to produce a input data stress isometric. From the pulldown, select Pipe>Accessory>CAESAR II>Annotation. CADWorx Plant will respond with the Stress Annotation dialog.

Pick the *Point* button in the *Model* area at the upper left. This will fill in the *Categories* list box. This lists data that is associated with specific point locations on the isometric. Picking the NODES in the *Categories* list box will display the Node Selection dialog. Pick the *Select all* button and then pick the *OK* button.

After the procedure above, return to the Stress Annotation dialog. In this dialog, change the characteristics of the annotation with the options on the left side of this dialog. Place the nodes manually or have the program do it automatically. Use *Auto* placement and *Circled* text for this exercise. This will invoke the automatic tagging algorithm. For nodes number equal to a 1000 or more, the user might want to increase the BOMTagRadiusFactor to 2.0 or more. This provides a bigger circle for larger node numbers such a 1000, 10000, etc. The text size is controlled by the lower left box. Check the textsize in this box to verify that it agrees with the current system or requirements. Prior to text placement, the program prompts for Monotxt style. This style uses the same amount of space for an "I" as for an "M". This is the prefer text for aligning columns in the summary outputs. Select the *OK* button in the Stress Annotation dialog and the command line should appear as below.

Command: **C2TEXT**
Monotext style [Yes/No] <Yes>: *hit enter to accept new monotext style*
Labeling NODES

For manual node number annotation, a rubber band line will appear to the actual point on the drawing where the node belongs. Use this line to establish the desired location of the annotation.

Pick point: *pick a point (turn <Ortho off> with the [F8] key)*
Skip/<Ok>: *enter*
Pick point: *pick a point*
Skip/<Ok>: **S** *for skipping an annotation that is not desired will proceed until all the NODES selected have been placed*
Pick point: *pick a point*
Skip/<Ok>: *enter*

Since both input data and results stress isometrics are going to be created here, choose the AutoCAD PROPERTIES command and change all the nodes to a different layer for further use with the results isometric.

Utilize the pulldown menu to restart the Stress Annotation dialog. Select Pipe>Accessory>CAESAR II>Annotation (an enter would have re-started this command if it was the last command used). If this was a previously created isometric (from another drawing session), the necessary files might not be available, thus requiring that a new _A file be re-selected.

Command: **C2TEXT**
Use last data file [Yes/No] <Yes>: *enter*

Hit the *Point* button in the *Model* area to place other desired RIGID WEIGHTS and RESTRAINT TYPES annotation as needed. This will complete the input data stress isometric.

Now create a results stress isometric. Save the drawing using the AutoCAD SAVEAS command if desired. From the pulldown, select the Pipe>Utility>Layer Control>Delete and select any of the text that is not the nodes. This will delete all but the nodes in the paper space.

Command: **LAYERDELETE**
Select entity layer to delete: *pick the annotated text just placed above*
Layer TEXT deleted...

From the pulldown menu, select Pipe>Accessory>CAESAR II>Annotation.

Command: **C2TEXT**
Use last data file [Yes/No] <Yes>: *enter*

Utilize the same procedures as indicated above to place point results.

REST LOADS (OPE) *select all then clear Prog Design VSH by individually picking them in the list box*
HANGER REPORT *Results - Points button - select all*
STRESSES *Results - Points button - pick a few large ones individually by just picking in the list box*

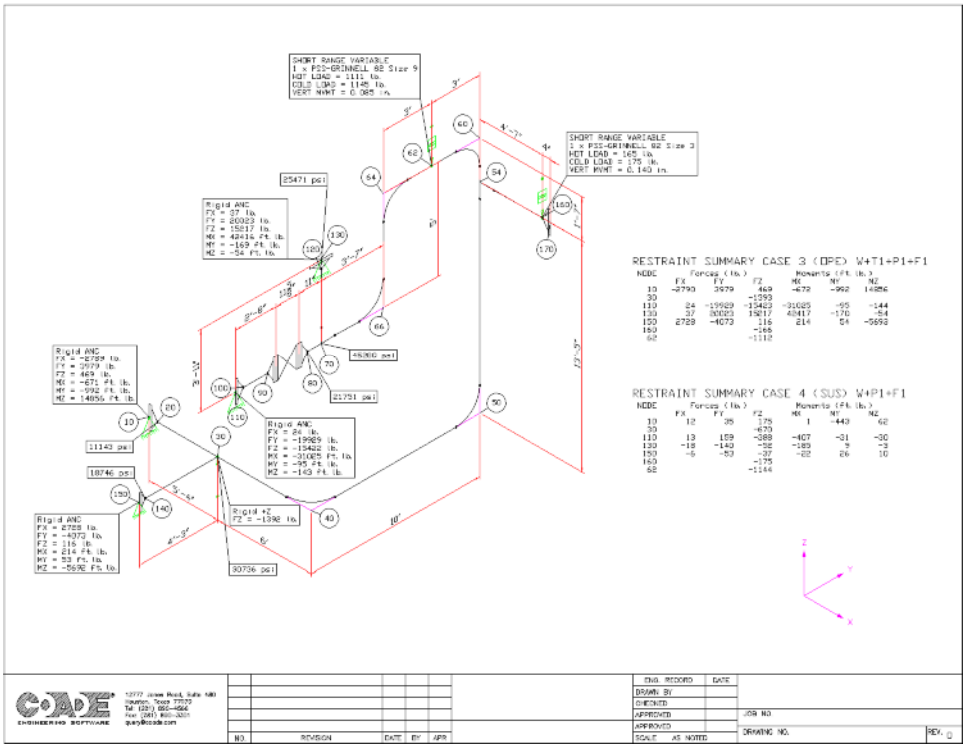
To place summary results, pick the *Summary* button in the *Results* area. This will gray out the *Placement*, *Text* and *Leader* areas of the dialog. The only available option is to place a table of summaries on the drawing. The *Textsize* box allows change to the heading size and the normal textsize for this procedure. This should include the following:

REST LDS (OPE)
REST LDS (SUS)

Follow the AutoCAD command line prompts after picking the categories from the *Categories* list box.

Labeling REST LDS (OPE)W+T1+P1+FOR
Pick left justified summary location: *pick a point in the upper left*

The results isometric can now be saved with the AutoCAD SAVEAS command. There are many more options which are available for use with CAESAR II. The Pipe>Accessory>CAESAR II>Multiple ISO will not be covered in this tutorial but can be very useful when the model is too large and it requires breaking up the CAESAR II model into multiple drawings.



Automatic Dimensioning

Two different uses of automatic dimensioning will be discussed here. It will require the use of two of the tutorial drawings. First, open the TUTOR10.DWG (TUTOR10M.DWG). The components have been converted to single line mode using the mode conversion utility. The CADWorx Plant Point and Shoot UCS has been repositioned to the VIEWNORTH and VIEWWEST to convert the flanges into a vertical position.

The first thing that should be discussed is the AutoCAD setvar DIMSCALE, DIMTXT, DIMASZ, etc. DIMSCALE should be set to 0 as mentioned previously in the paper space topic. This causes the dimension text and arrows to automatically size themselves. The DIMASZ and DIMTXT setvars should be set to the desired plotted size. (DIMTXT and DIMASZ = 1/8 or 3mm, approximately). Within this environment, the dimensions should scale correctly.

While in an active view port (*Model* button), from the pulldown menu, select Pipe>Text>Dimension>Automatic.

Command: **AUTODIM**
Enter an option [Flange/Offset/Selection] <Selection>: *enter to the start selection*
Select objects: **ALL**
Select objects: *enter to finish selection set*
Pick centroid point of dimensions: **ENDP** of *the elbow in the middle of the piping, next to reducer*
Auto Dimensioning...finished

The *Flange* option allows dimensioning to flange faces. If this option is turned off, the dimension will just dimension to the corners and tees of the piping system. The *Offset* option allows the user to adjust the distance from the component to the dimensioning.

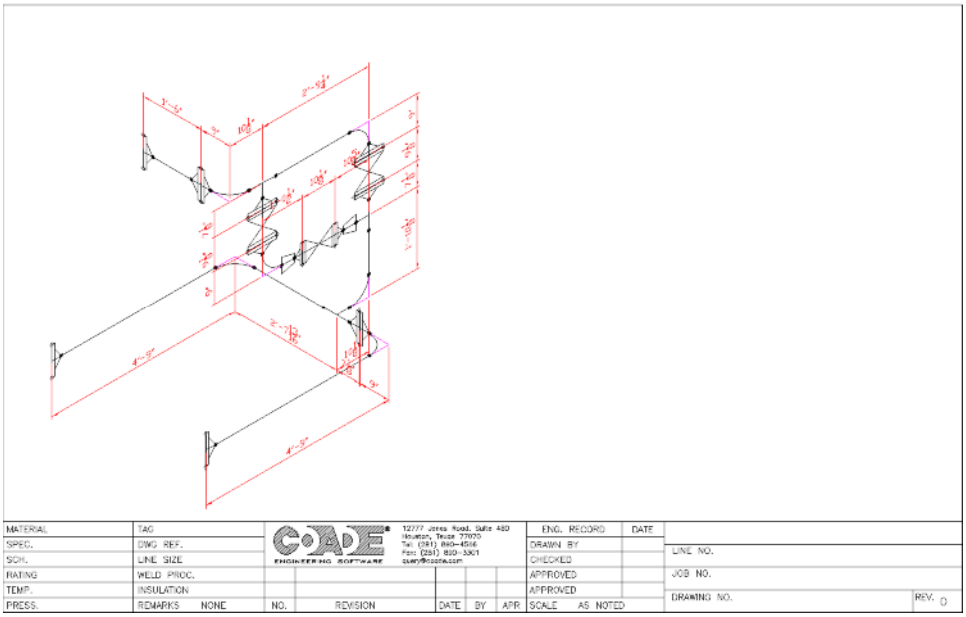
Note: The system must have at least one turn in it to use this feature. If the message below is displayed, then the system is lacking an elbow or tee.

Inadequate selection...aborting *Cancel*

When modifying dimensions that are already within the drawing, the UCS Object option is very useful. Issue the UCS command and utilize the *Object* option and select the dimension. This sets the UCS and the dimension in the same plane. It is very easy to stretch, grip and correct overlapping dimensions when using this option of the UCS command. Also, the CADWorx Plant Point and Shoot UCS is helpful.

Something else that is certainly worth mentioning is that dimensions can be re-scaled only if the setvar DIMASO is on (DIMASSOC in AutoCAD 2002). If the view port is re-sized by zooming, the dimensions can be updated to the correct scale.

Command: **DIM**
Dim: **DIMTXT**
Enter new value for dimension variable <3/16">: **0.125 (3)**
Dim: **UPDATE**
Select objects: **ALL**
Select objects: *enter to finish selection set*
Dim: **EXIT**

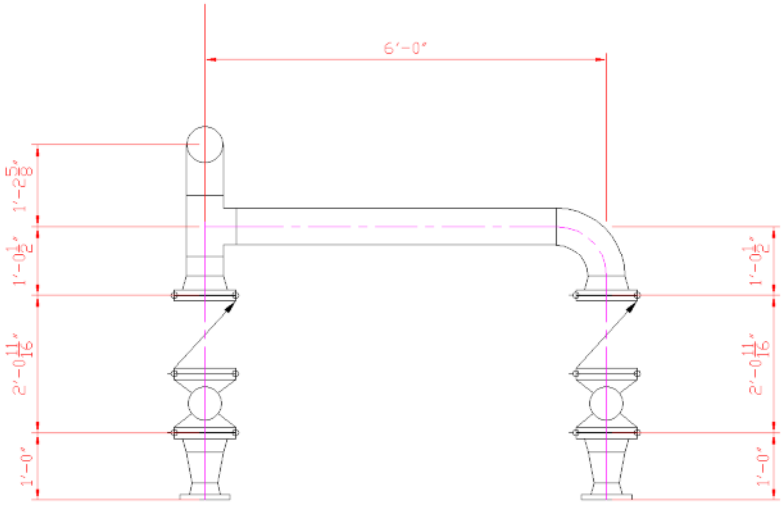


MATERIAL	TAG	COADE ENGINEERING SOFTWARE 12777 Jones Road, Suite 480 Houston, Texas 77070 Tel: (281) 890-4516 Fax: (281) 890-3301 query@coade.com		ENG. RECORD	DATE	
SPEC.	DWG REF.			DRAWN BY		LINE NO.
SCH.	LINE SIZE			CHECKED		
RATING	WELD PROC.			APPROVED		JOB NO.
TEMP.	INSULATION			APPROVED		DRAWING NO. REV. 0
PRESS.	REMARKS NONE	NO.	REVISION DATE BY APP	SCALE AS NOTED		

Next open TUTOR4.DWG (TUTOR4M.DWG). From the pulldown menu Pipe>Setup, execute this function and set the scales to ¾"=1'-0". From the pulldown menu Pipe>Dimension>Automatic, pick the automatic dimensioning one more time. Experiment with the *Flange* option (set it to off). Also notice that the dimension in this plan were placed in the flat position. The display controls this aspect of automatic dimensioning. If the view is in a plan view, the dimension are placed flat. If the view is in an isometric view, they are placed in the vertical.

Command: **AUTODIM**
Enter an option [Flange/Offset/Selection] <Selection>: **F**
Flange dimensioning [ON/Off] <Off>: *hit enter to accept off*
Enter an option [Flange/Offset/Selection] <Selection>: *enter to start selection*
Select objects:
Select objects: *enter to finish selection*
Pick centroid point of dimensions: *pick a point in the middle of the piping*
Auto Dimensioning...finished

As expected, the flanges are not dimensioned as in the isometric above. The drawing should appear as the figure below. Type "U" to undo the dimensioning and try turning the Flange option back on. Then pick a point on the right side of the piping. After that undo the dimensioning and redo it by picking a point on the far left. In fact, try several different locations such as the very top, top left, bottom left, bottom right, top right, etc. All these locations will produce different results. Choose small sections at a time to automatically dimension instead of the entire drawing.



Bill of Material - Tutorial

Open drawing TUTOR10.DWG (TUTOR10M.DWG). This isometric drawing will have a bill of material applied. Choose the pulldown, Plant>Bill of Material>Run>Cut from the pulldown menu.

Command: **BOMCUT**
Enter tag option [Counter/Manual placement/Automatic placement] <Automatic>: *return to accept automatic*
Select upper right corner of BOM heading: *pick a point at the corner of the border*
Enter an option [Line number/Select components] <Select components>: *return to select*
Select objects: *select all the objects in the view port*
Select objects: *hit enter to finish*

This will automatically place a schedule and all the associated tags on the drawing in paper space. The Bill of Material generator will perform special tag optimization when this facility is turn on in the configuration dialog. For more information on BomTagOptimize (see page 53) variable, refer to the startup variables.

If the modeled piping view was panned slightly, the component marks will not be correct. They can be adjusted by the MOVE command. Return to paperspace with the PSPACE command. Move the material marks by window and select a base point at a leader's endpoint. Next, select a second point of displacement, Osnap, midpoint of, or the component's centerline. Osnap mode selection is allowed from paper space to model space. This will correct the alignment of the material marks.

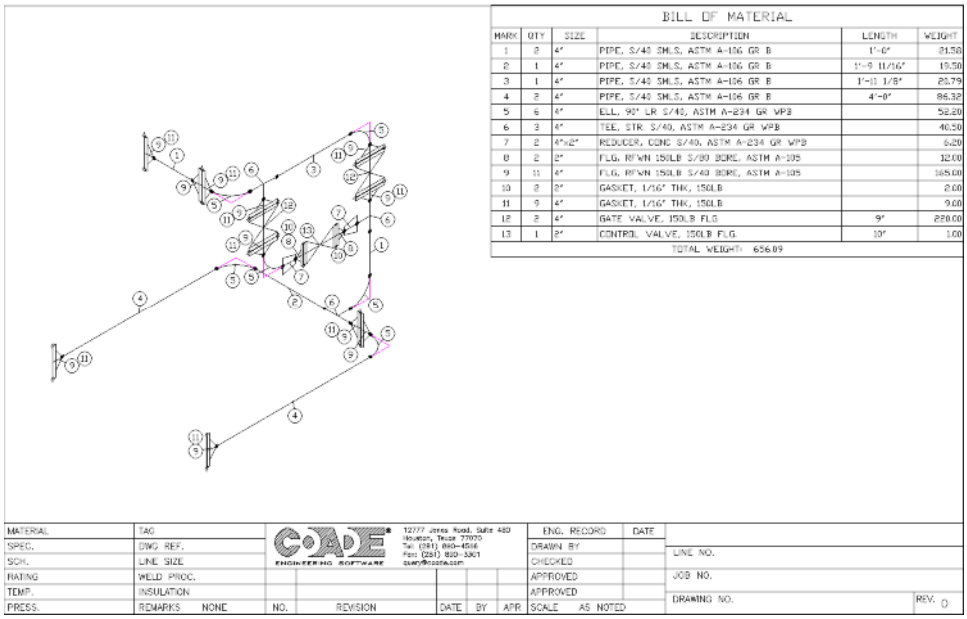
From the pulldown Plant>Bill of Material>Setup, any column or width can be applied to the bill of material. Within this dialog, sorting order and sort by columns can be established.

The system variables BOMScheduleDirection, BOMScheduleSpacing, BOMGridFactor, BOMTagOptimize, BOMTagRadiusFactor, and BOMTagSpacingFactor can all be used to control the bill of material characteristics.

From the pulldown Plant>Bill of Material>Tags, there are functions which assist with the placement or relocation of material tags. The TAGRELOCATE command allows material tags to be relocated automatically. The leader to the component will also be adjusted with this function. The TAGINSERT function allows the placement of a single graphic mark symbol (it will not be consistent with the bill of material). The TAGTOGGLE function allows the tags to be toggled off. This is similar to the toggle in the various dialogs for the automatic and stress isometrics.

The bill of material schedule is developed and grouped together. For additional lines within the bill of material schedule, just select one of the existing lines and copy it to the new location. Use the AutoCAD DDEDIT command to modify this new line item. The Plant>Bill of Material>Export function provides the capability to extract the components in the drawing to a spreadsheet of database.

Refer to the bill of material section for further information regarding the procedures discussed in the previous paragraphs. The bill of material for the above isometric should appear as below.



BILL OF MATERIAL

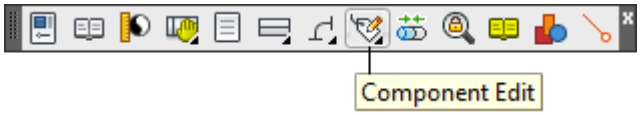
MARK	QTY	SIZE	DESCRIPTION	LENGTH	WEIGHT
1	2	4"	PIPE, S/40 SMLS, ASTM A-106 GR B	1'-6"	21.50
2	1	4"	PIPE, S/40 SMLS, ASTM A-106 GR B	1'-9 11/16"	19.50
3	1	4"	PIPE, S/40 SMLS, ASTM A-106 GR B	1'-11 1/8"	20.79
4	3	4"	PIPE, S/40 SMLS, ASTM A-106 GR B	4'-0"	86.32
5	6	4"	ELL, 90° LR S/40, ASTM A-234 GR WPB		52.20
6	3	4"	TEE, STR S/40, ASTM A-234 GR WPB		40.50
7	2	4"x2"	REDUCER, CONC S/40, ASTM A-234 GR WPB		6.20
8	2	2"	FLG, RFWN 150LB S/80 BORE, ASTM A-105		12.00
9	11	4"	FLG, RFWN 150LB S/40 BORE, ASTM A-105		165.00
10	2	2"	GASKET, 1/16" THK, 150LB		2.00
11	9	4"	GASKET, 1/16" THK, 150LB		9.00
12	2	4"	GATE VALVE, 150LB FLG	9"	220.00
13	1	2"	CONTROL VALVE, 150LB FLG	10"	1.00
			TOTAL WEIGHT: 656.09		

MATERIAL	TAG	CADE ENGINEERING SOFTWARE 12777 Jones Road, Suite 480 Houston, Texas 77070 Tel: (281) 890-4566 Fax: (281) 890-3361 query@cade.com				ENG. RECORD	DATE	
SPEC.	DWG. REF.					DRAWN BY		LINE NO.
SCH.	LINE SIZE					CHECKED		
RATING	WELD PROC.					APPROVED		JOB NO.
TEMP.	INSULATION					APPROVED		
PRESS.	REMARKS NONE	NO.	REVISION	DATE	BY	APR	SCALE AS NOTED	DRAWING NO. REV. 0

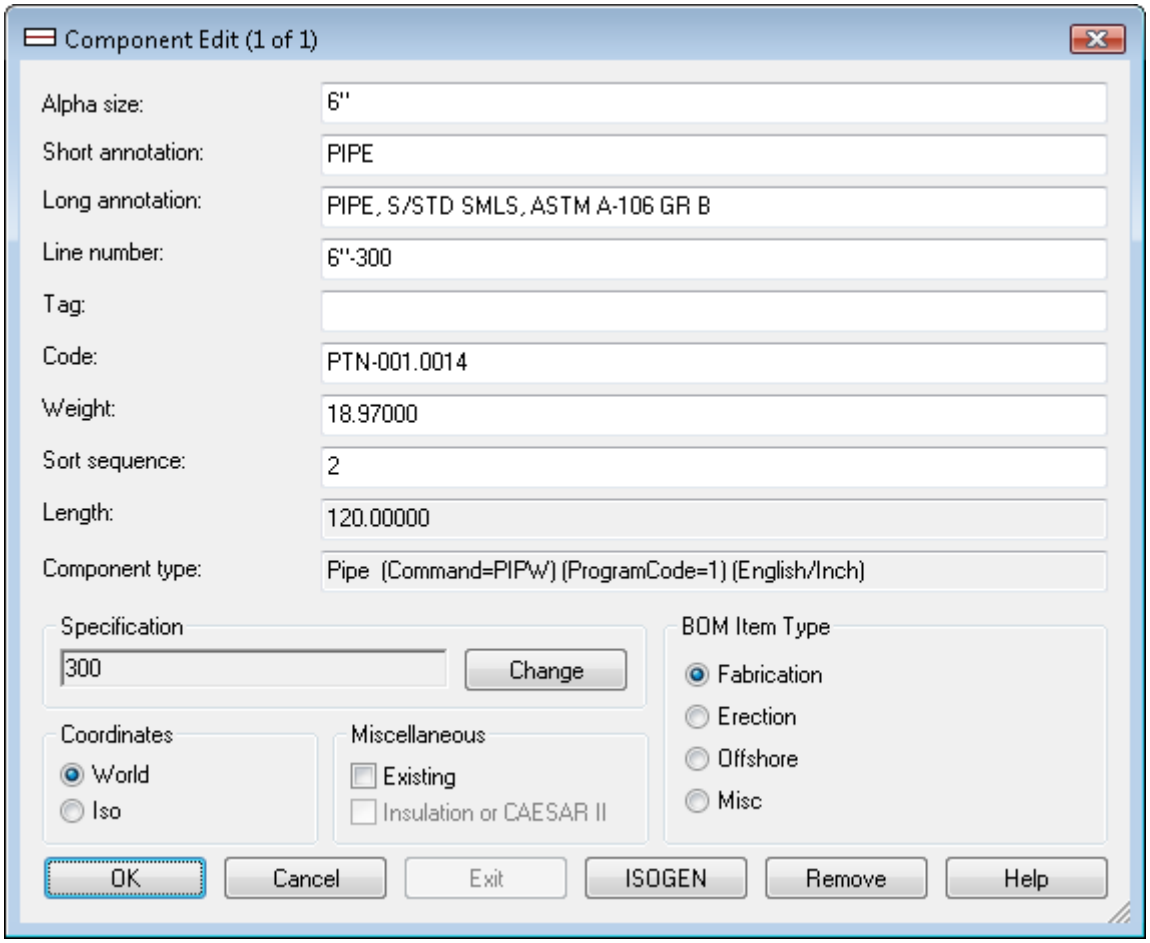
Component Editing

Open TUTOR2.DWG (TUTOR2M.DWG). From the Pipe>Utility>Edit Component, select any pipe or component within the drawing. This function is also on the Settings toolbar which can be accessed from the Pipe>Toolbars>Settings pulldown as shown below. This function can also be started at the command line by typing CEDIT.

Command: **CEDIT**



After selecting a CADWorx Plant component, the following dialog is displayed.



This dialog allows various fields of the component to be changed. The first edit boxes allow descriptions to be changed. The first *Alpha size* will not allow the physical size of the component to be changed. There is not any automatic mechanism which will automatically change all the 2" components to 4". This is only an alpha description.

The edit boxes allow changes as required. The *Weight* edit box allows the weight per unit to be changed on pipe. For other components, it will indicate the total weight. The *Sort sequence* is used for the bill of material. If the components location in the bill of material needs to be last, enter 999 in the *Sort sequence* edit box.

The *Change* button provides the capability to change the specification used in components already placed in a drawing. For example, if a AA1 specification was used for a component placed in the drawing, this function could change it to a new spec such as BA1SS.

Caution should be used when changing a 150# based specification to a 300# based specification. This function will change the associated data file and the next mode conversion will show a problem of over lapping components due to lengths that are different in the two specifications. A 300# RFWN flange is longer than a 150# RFWN flange.

The *Remove* button allows the data inside of the component to be removed. This could be used for marking existing piping within a new plan or piping drawing. This will remove the information forever.

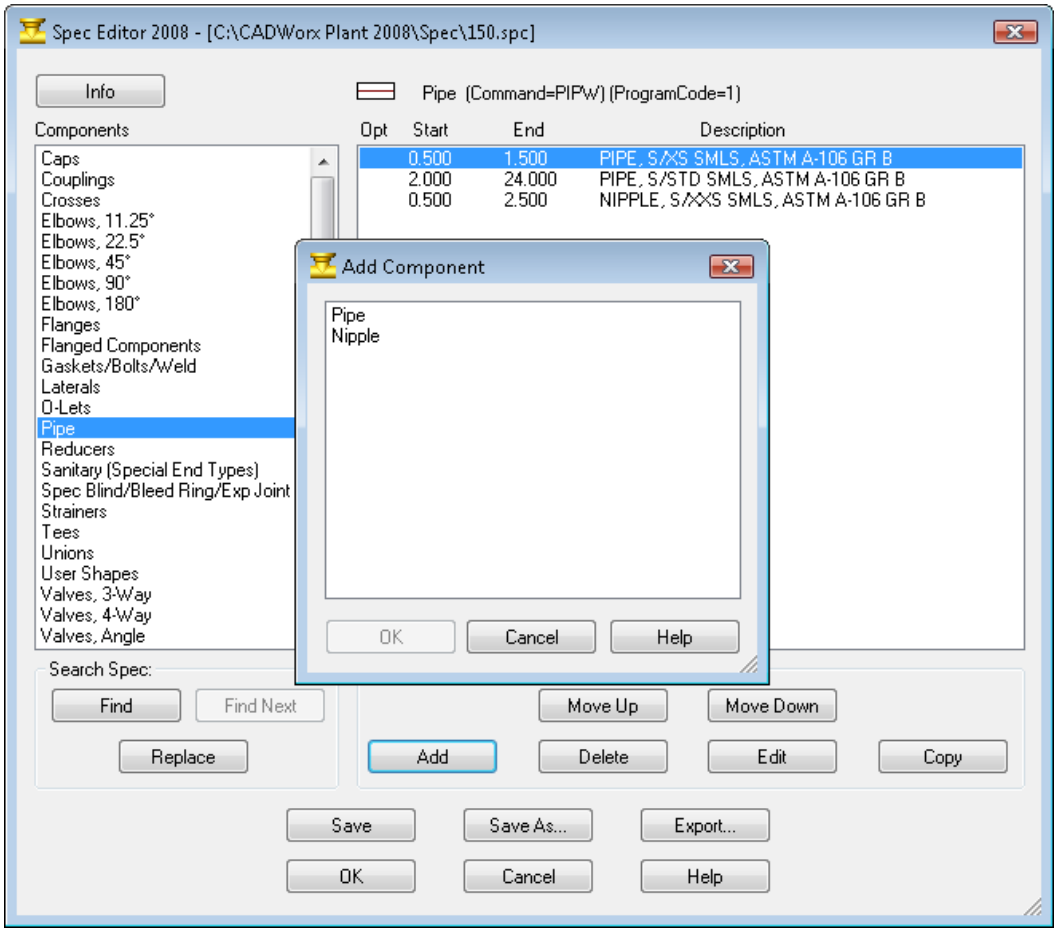
The Coordinates box allows the length of the pipe to be altered. This will not change the physical length on the drawing. It will change the length that appears in the bill of material. If the *Iso* option was used, the length could be changed in the *Length* edit box above (this edit box would not be disabled). Stretching the component would not change the length recorded inside of it. If this option was not used and the component was stretched, the length would automatically update.

The *Existing* option in the Miscellaneous box allows the component to be placed in a state where the component would not appear in the bill of material but would retain all the information about itself. This function will also convert the component to the Exist layer (installation layer name). The CAESAR II information indicates whether the component has stress analysis information attached and would allow it to be removed if this toggle was un-checked.

Specification Editor

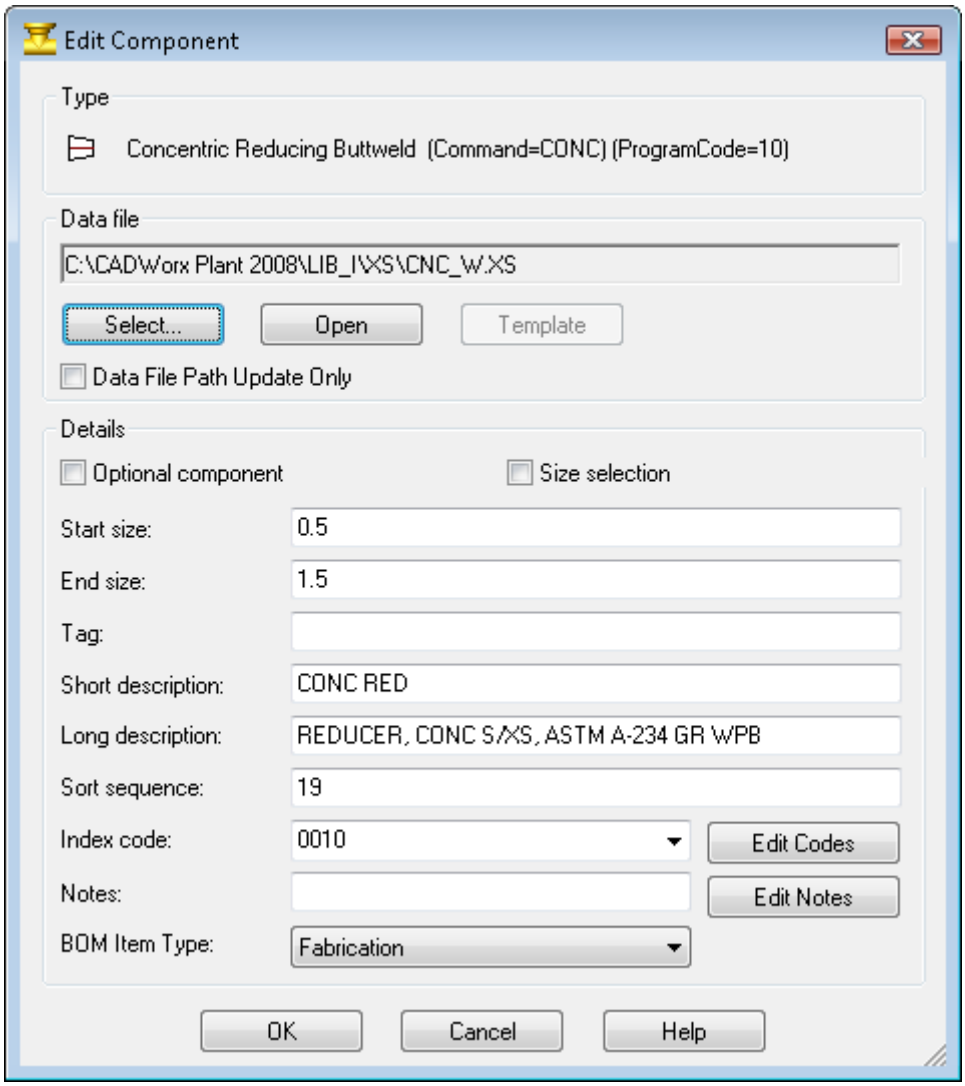
In this section of the tutorial, three simple modifications will be made to one of the template specifications. The first exercise will involve adding a new pipe component that will handle 26" - 42" pipe. This is not presently included in the template specification. The second part of this exercise demonstrates the use of optional components which can be toggled on or off.

Open the 150.spc (150_M.spc) from the pulldown menu Pipe>Accessory>Specifications>Editor or run the SPECEDIT command. Presented is a listing on the left of all the component categories within the specification. Adding a new pipe component is simple. Single click the category on the left side of the specification editor named PIPE. This displays all the components in this group in the right list box. Now click the *Add* button on the bottom of this dialog. This displays a dialog which allows adding a component which is available in the PIPE category. The components in the right list box are components already placed in the spec while the components in the add dialog are available for adding.



Add a new pipe component (shown above in the dialog). This provides the Edit dialog where changes to the size and descriptions can take place. The *Edit* button can be used in the same respect to edit this added component.

Click the *Open* button on the dialog. This presents a standard file selection dialog. After double clicking one of the directories, data files will be presented to choose from. Double click on the STD library and then single click on the PIP_W_LG.STD data file. This file selection dialog updates dynamically with each data file that is chosen. Select the *OK* button for this data file.



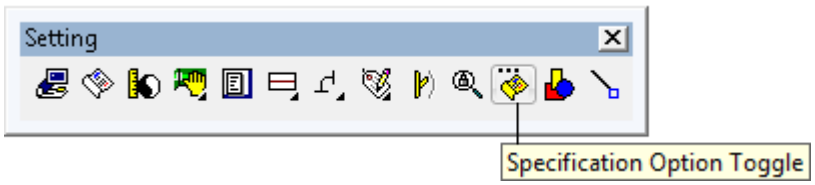
This places all the required information into the Edit dialog which can then be saved as a new component within the specification. Change the *Ending* size to 42”, modify the *Short* or *Long* description if desired. Select *OK* for this entry and it will be added to the specification. By picking the *SaveAs* button, the specification can be saved under any name with this modification and will be automatically used whenever 26”-42” pipe is selected. Choose the *SaveAs* button and save this specification as TUTOR.SPC. If this specification is presently set, it will automatically load into memory. If not the user will have to set it.

Command: **SPEC**
Specification <>: **TUTOR**
Preparing TUTOR specification...finished

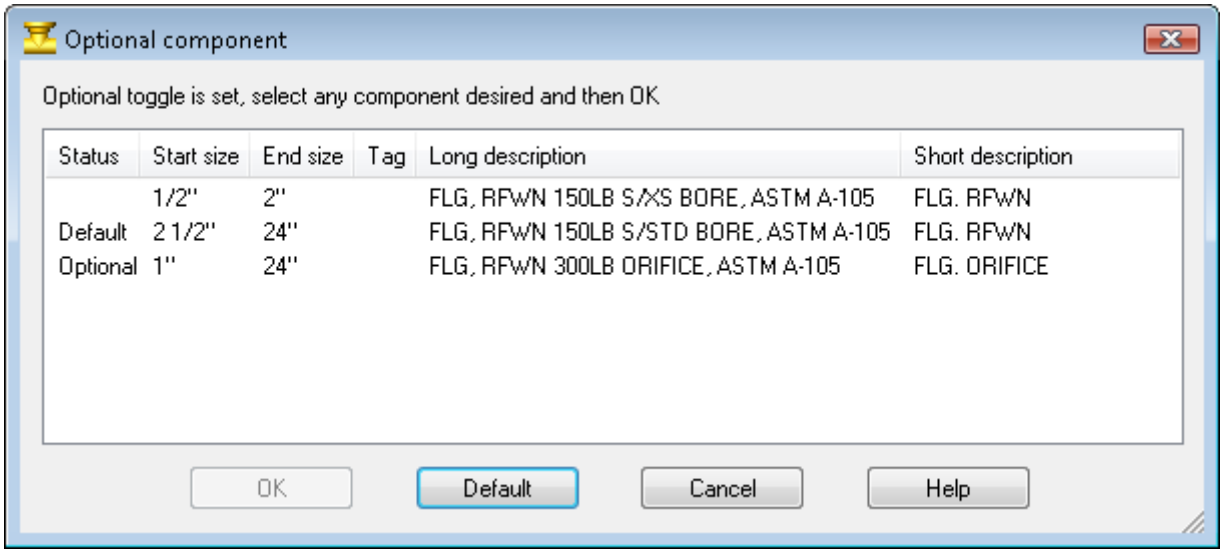
A TUTOR specification now exists in CADWorx Plant (in memory and saved on disk) which can be used over and over again.

The second part of this section deals with the use of optional components. Open the TUTOR.SPC specification with the editor as demonstrated earlier, and select the FLANGES in the left list box. This indicates that available to the user in this specification are 300# orifice flanges. Highlight the flanges and pick the *Edit* button. Notice in the top right corner of this dialog is an *Optional* toggle. This toggles the component for use as optional. It will not be used unless requested. Close this specification without modification.

To toggle this option on requires the user to activate a toggle from the pulldown, Pipe>Accessory>Specification>Toggle or type SPECOPTION from the command line. There is also a button for it on the Settings toolbar.



The toggle displays “All components will be dialogued...” or “Specification dialog turned off...”. This option provides a dialog which allows the user to select the 300# orifice flanges. This option also controls the ability to place a schedule 80 pipe (from the ½” to 2” line item range) while placing a 8” pipe (therefore making the 8” pipe schedule 80). Below is the dialog which will be presented each time a component is placed while the option toggle is on.



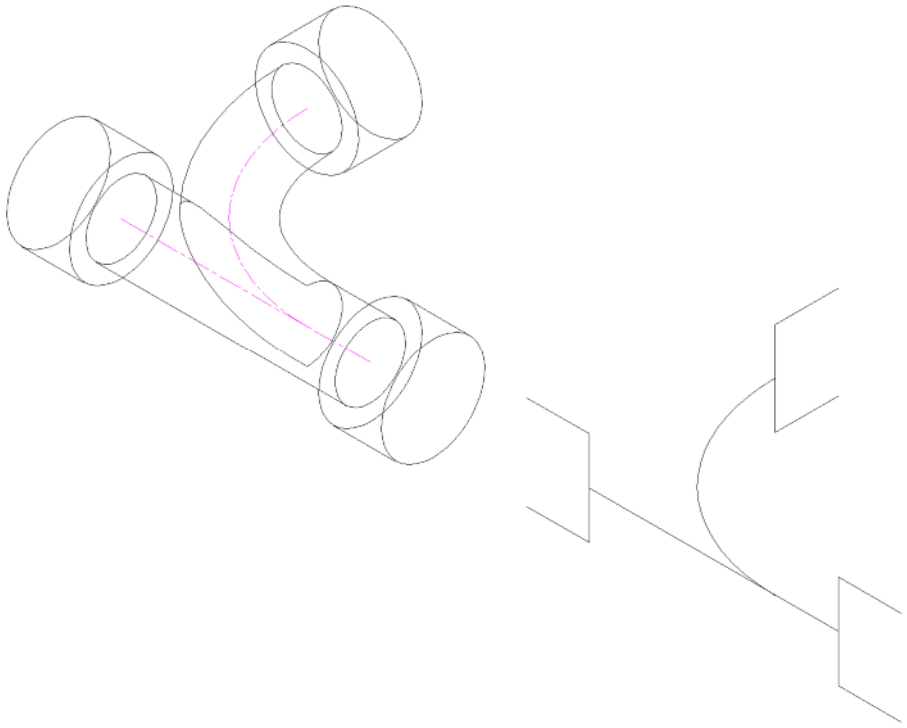
There are many other useful functions within the specification editor which were not discussed here. The search and replace capabilities provide an excellent way of changing grades or materials to any user requirement. Please refer to the specification editor (see page 254) for more complete documentation on all these functions.

User Shapes

In this topic, a typical user shape will be created and added to a specification. CADWorx Plant will automatically create all the data files and blocks required. Create the drawing as shown below or open the drawing TUTOR12.DWG (TUTOR12M.DWG). The size of this component will be 4". This is a typical PVC swept tee for plumbing purposes.

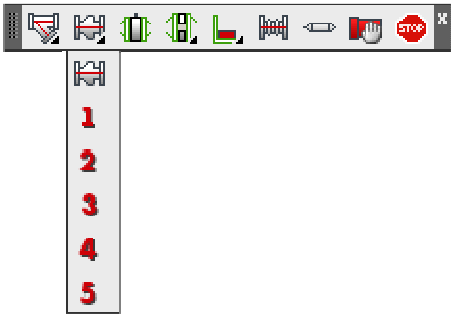
For the 3D solids component, the main body of the swept tee was drawn using the PIPW command and the swept part was drawn using the *Select arc* option of the BENT command (from the pulldown *Plant>Toolbars>Buttweld*). These two parts were drawn at 4" and then joined together with the AutoCAD UNION command. The hubs were drawn with the PIPW command with the size set to 6". PICKSTYLE was turned off and the centerlines were erased.

For the single line component, the 3D solids were copied and converted to single line mode using the CONVERTSINGLE command. The 6" hubbed ends were erased and redrawn with the AutoCAD PLINE command as shown. They were drawn vertical for use with automatic isometrics (the isometric program automatically places components vertically).



Step 1 - Creating the component...

The user should run the USERCREATE command (pulldown *Plant>Toolbars>Misc*) first. This command should be repeated for the various sizes of the swept tee. The user would be required to redraw each size as the 4" size above was drawn above.



Command: **USERCREATE**
[Append existing file/Create new file] <Create new file>: *enter to create a new file*

Since this is a new component, the user should use the *Create new file* option. If the user were adding a new size, the *Append existing file* option can be used.

Enter name of library directory <USE>: *enter to accept the USE sub-directory*
Enter name of file, no extension <COMP>: **SWEPTTEE** *this is the file name*
Enter specification <150>: *enter to accept the default specification*
Enter short description: **SWEPT TEE**
Make component reducing [Yes/No] <No>: **N**
Enter weight: **15 (6.8)**
Enter density: **0.2838 (7.8417)**

Here the size can be changed as required. The 4" needs to be set and will be recorded in the data file.

6" main size [Yes/No] <Yes>: **N**
Nominal main size <6">: **4**

At this point of the command, the program can be used to create non-symmetrical as above or the old symmetrical shapes of the previous version of CADWorx Plant. In this example, use the *Block* option as shown below.

Enter an option [Pick points/Select polyline/Block]: **B**

Here, both modes of the component can be created. If the user was not interested in isometrics, they could just select the 3D option.

Enter an option [2d/3d/Both] <Both>: **B**

Next, a reference point must be selected. As the prompt indicates when creating both 2D and 3D components, start with the 2D representation that was previously drawn. Reference points create an axis through the component. Pick the start point and then a directional vector through the component which is later used to locate the user shape in the drawing. It is the same as a base point in a block. The two reference points establish a rotation for the shape.

Start with 2D representation...
Pick reference point: *pick at the lower right connection of the main run of the 2Dcomponent*
Pick end point: *pick at the upper left connection of the main run of the same component*

Connection points must be recorded for use with the program. Connection points are simply locations where other components will attach to the user shape.

When finished, hit enter.
Pick connection point: *pick at the lower right connection of the main run of the 2D component*
Next connection point: *pck at the upper left connection of the main run of the same component*
Next connection point: *pick at the branch connection point of the single line component*
Next connection point: *enter to finish*

2D representation...
Select objects: *select by window all the graphics which make up the 2D component*
Specify opposite corner: *pick a point to complete selection window*
Select objects: *enter to finish the selection set*

The next prompt allows the user to relocate the reference point for selecting the 3D portion of the graphics required to make the 3D component.

Reselect reference point [Yes/No] <Yes>: *enter*
Pick new reference point: *pick at the lower right connection of the main run of the 3D component*
Pick end point: *pick at the upper left connection of the main run of the same component*

3D representation...
Select objects: *select by window all the graphics which make up the 3D component*
Specify opposite corner: *pick a point to complete selection window*
Select objects: *enter to finish the selection set*

Finished...

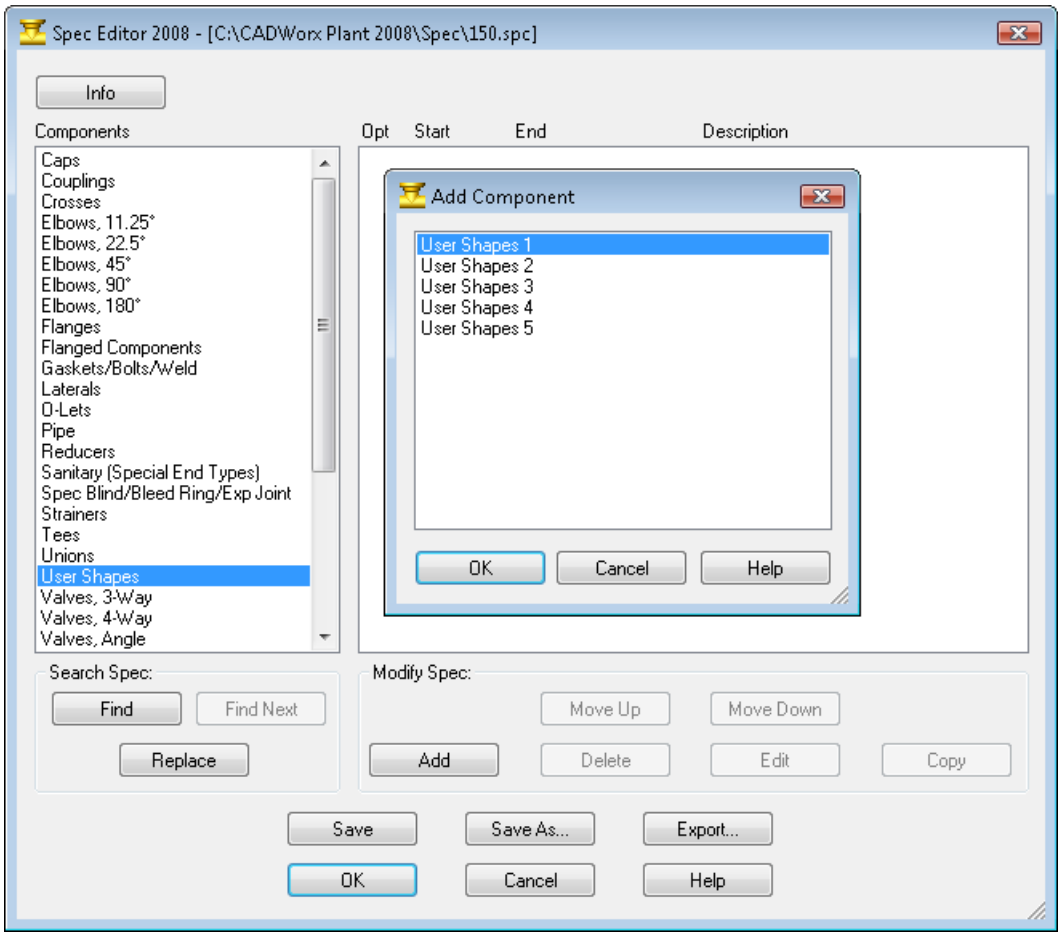
The final prompt indicates that the command is finished and that everything was completed successfully. As mentioned above, the user will have to redraw the component for the next size and repeat this same process. This process creates several files within the directory used above. In this example, the program creates a data file named SWEPTTEE.USE. This file controls the sizes and the connection points. This example also created two blocks which were named SWEPTTEE[4.0-2D].DWG and SWEPTTEE[4.0-3D].DWG. If the reducing option had been used, it would have names such as SWEPTTEE[4.0x3.0-2D].DWG. As the user appends to the program more blocks would be created with different sizes such as SWEPTTEE[2.0-2D].DWG.

This example uses the USE sub-directory under the <INSTALL-DIR>\LIB_I (<INSTALL-DIR>\LIB_M) which is referenced by the specification. It is important not to locate this file in some directory which is not referenced by the specification to be used. The program will not be able to find the blocks and their control data file.

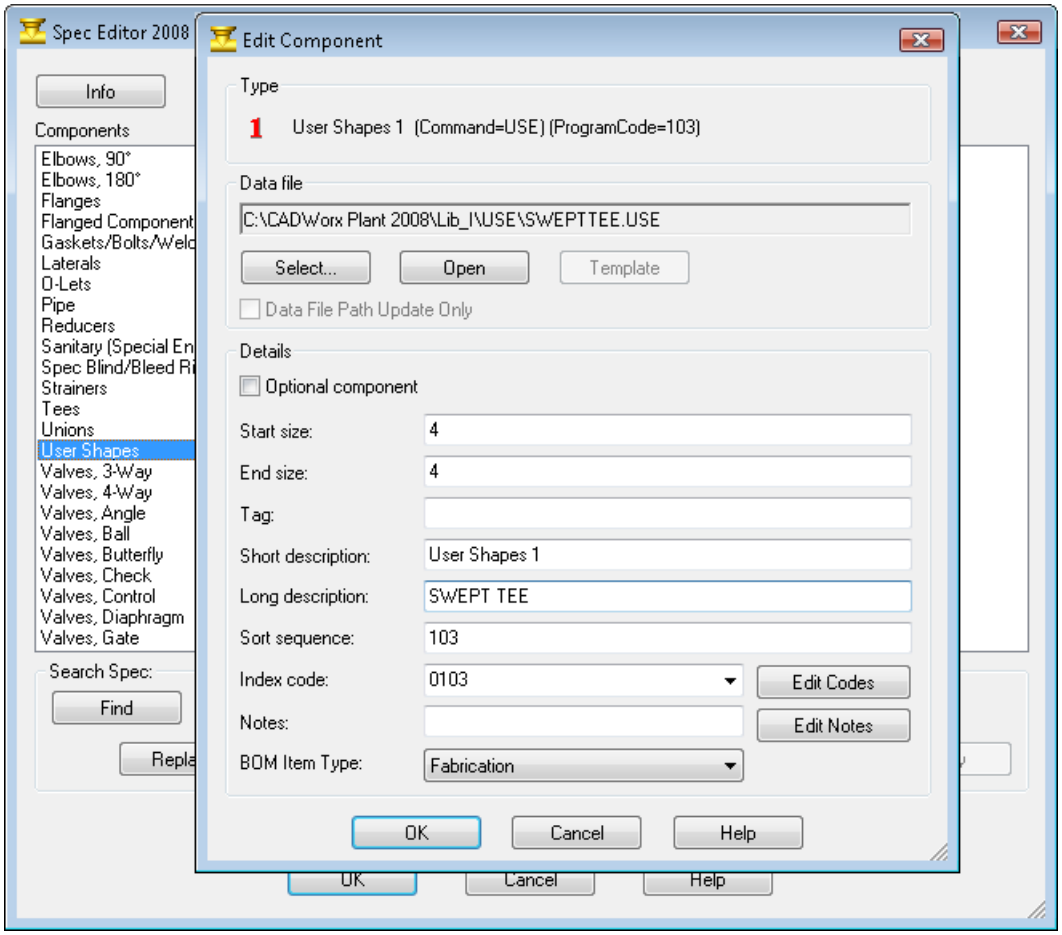
Step 2 - Adding the component to the specification...

The next step is to add the component to a specification. The user must add this component to the specification for it to be used in the drawing environment. Open up the 150.SPC (150_M.SPC) with the specification editor.

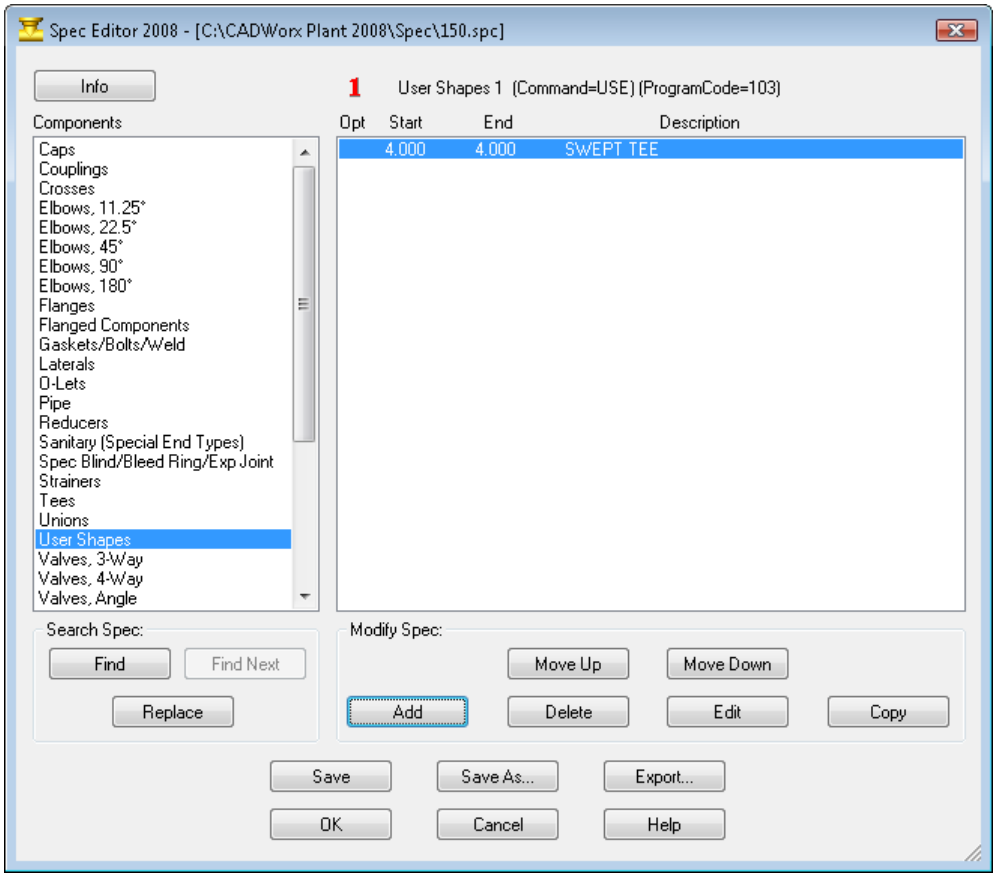
Command: **SPECEDIT**


Spec Editor 2008 - [C:\CADWorx Plant 2008\Spec\150.spc]
Info
Components
Opt
Start
End
Description
Caps
Couplings
Crosses
Elbows, 11.25°
Elbows, 22.5°
Elbows, 45°
Elbows, 90°
Elbows, 180°
Flanges
Flanged Components
Gaskets/Bolts/Weld
Laterals
O-Lets
Pipe
Reducers
Sanitary (Special End Types)
Spec Blind/Bleed Ring/Exp Joint
Strainers
Tees
Unions
User Shapes
Valves, 3-Way
Valves, 4-Way
Valves, Angle
Add Component
User Shapes 1
User Shapes 2
User Shapes 3
User Shapes 4
User Shapes 5
OK
Cancel
Help
Search Spec:
Find
Find Next
Replace
Modify Spec:
Move Up
Move Down
Add
Delete
Edit
Copy
Save
Save As...
Export...
OK
Cancel
Help

On the left hand side of the dialog, highlight the USER SHAPES. Pick the *Add* button at the bottom and select USER 1. Pick the *OK* button which would provide the user with the *Edit Component* dialog.



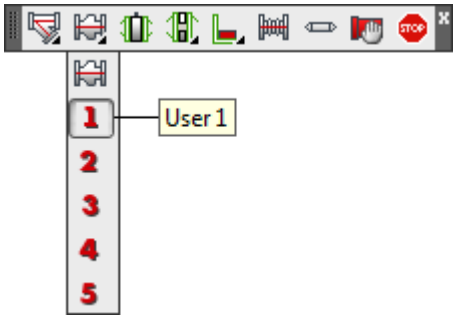
Utilize the *Select* button and choose the data file created from the USERCREATE command above - <INSTALL-DIR>\LIB_I\USE\SWEPTTEE.USE (<INSTALL-DIR>\LIB_M\USE\SWEPTTEE.USE). This example has only 4", otherwise the start and ending size could be adjusted. The short and long description can be modified or changed. The optional component could be used if there were more than one user components, When finished, pick the *OK* button and then save this spec as 150NEW.SPC (150_MNEW.SPC)or some other user name.



The new user shape has been added to the specification and is now ready to be drawn into the drawing environment.

Step 3 - Drawing the component...

For all the user shapes, there is only one command which inserts them into the drawing. The user must use the toolbar icons for starting the programs - USER1 through USER5. After setting the spec, 150NEW (150_MNEW), pick the USER1 icon from the toolbar as shown below.



Command: **USE** *(this command must be selected from the toolbar)*
4" SWEPT TEE
Pick location: *pick a point*

This part of the prompt allows the user to rotate the component, mirror it in the X, Y, Z and to roll it up or down or at any angle desired. The component can be placed in the drawing in virtually any position.

Pick rotation or [Up/Down/Roll/mirrorX/mirrorY/mirrorZ]: **X**
Pick rotation or [Up/Down/Roll/mirrorX/mirrorY/mirrorZ]: *pick a rotation*

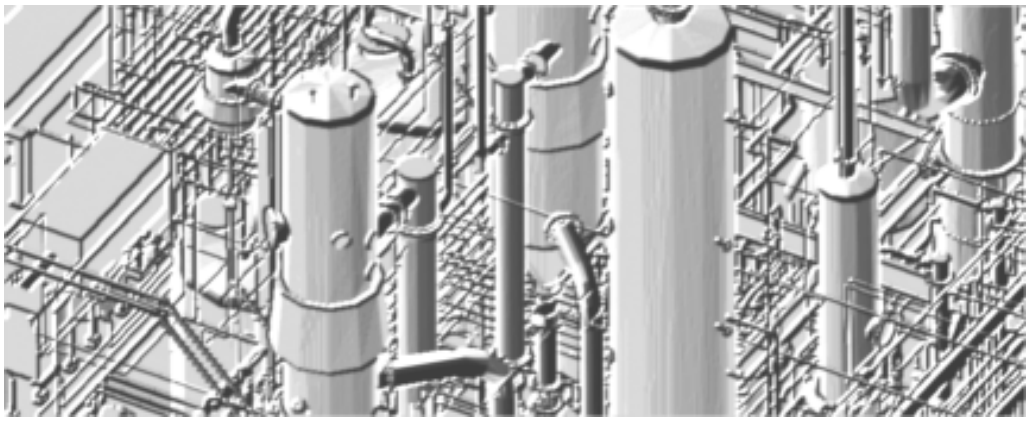
From the AutoCAD pulldown, View>3D View>Southeast, select the desired view and then mode convert it to solids.

Command: **CONVERTSOLID**
3D solids conversion...
Select objects: *pick the component*

The component will now act as any other CADWorx Plant component with the exception of converting to 2D double line mode. If conversion to one of these mode occurs, only a small X will be left in the drawing. At anytime, conversion back to solids or isometric mode can be applied. This component can be imported into the automatic isometric program and will appear in the bill of material generation.

CHAPTER 15

FAQ



In This Chapter

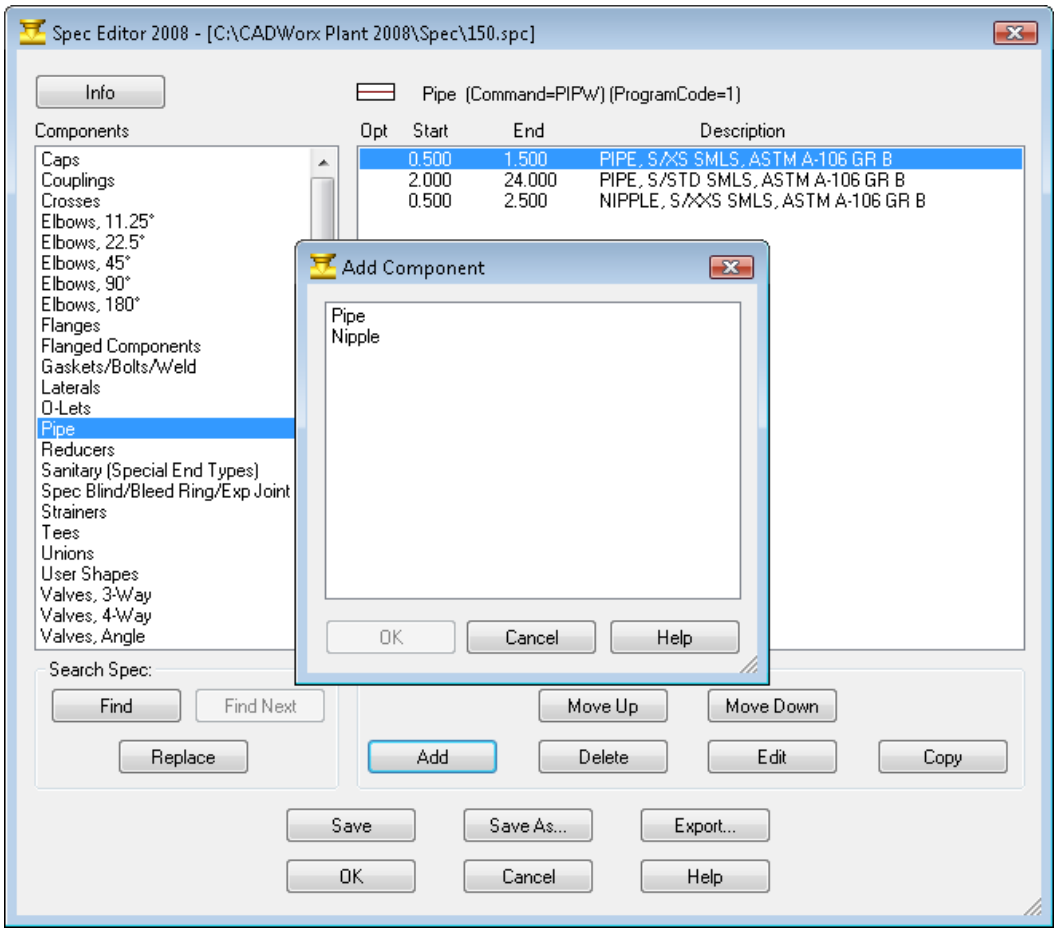
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Adding a Component

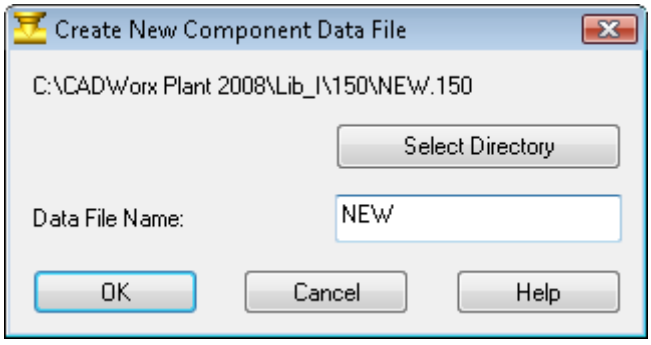
Adding components is extremely easy with CADWorx Plant. The following steps outline the procedure.

1. The user should open the specification they are working with using the Specification Editor (see page 254).
2. Select the appropriate component type from the list on the left side of the window.

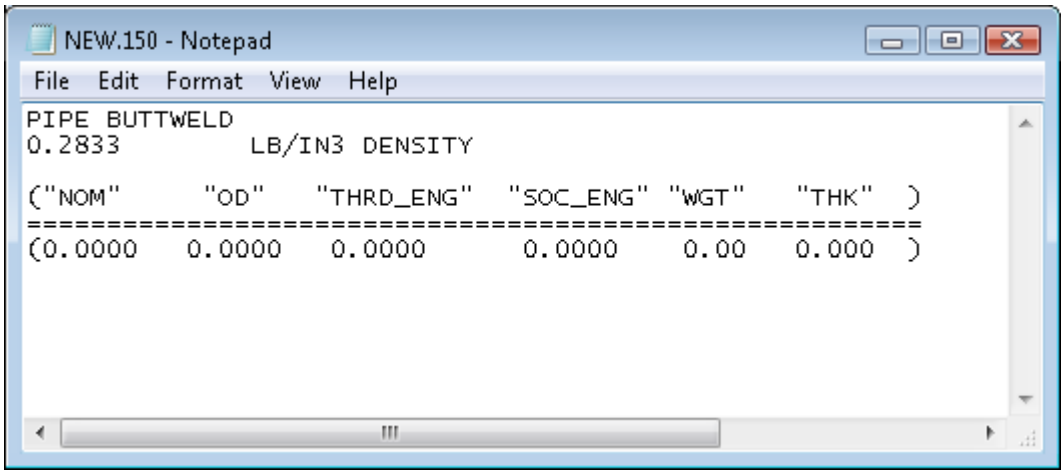
3. Click on Add and select one of the available components as shown below.



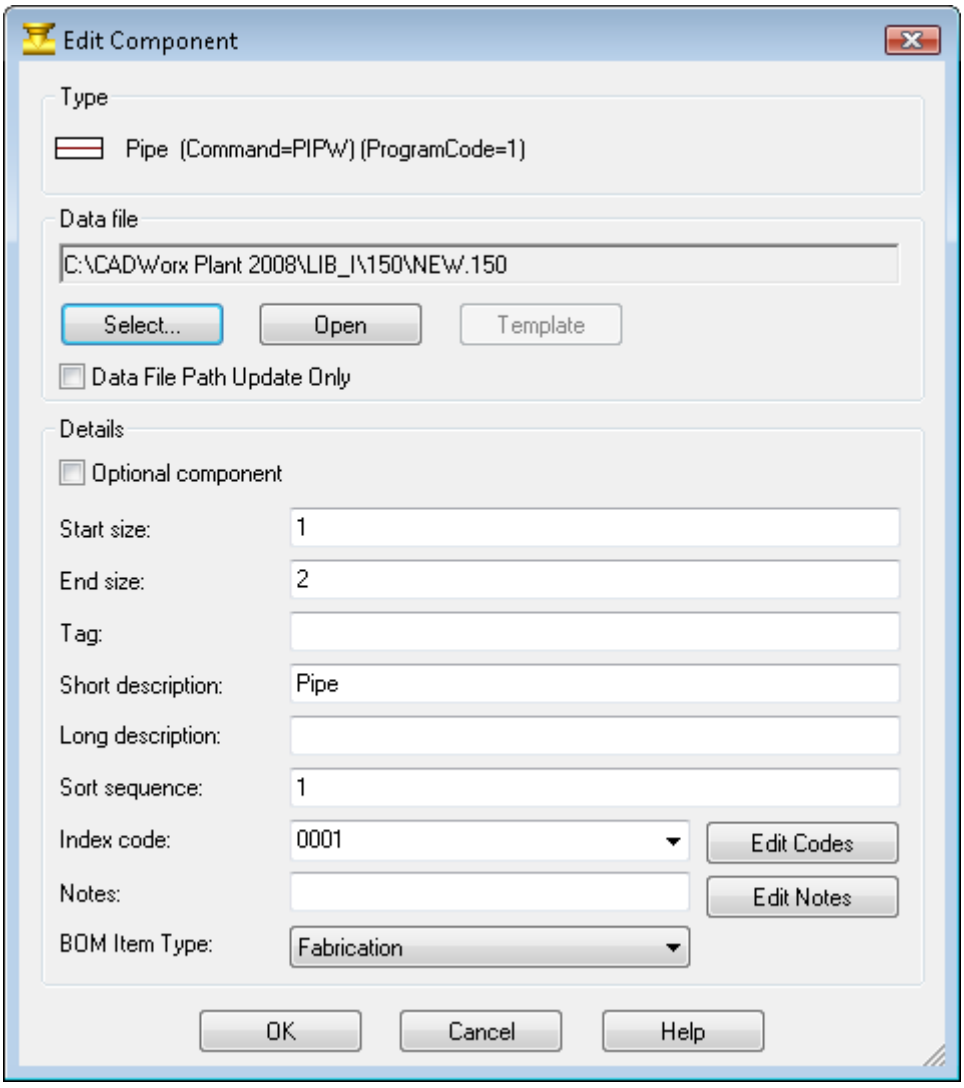
4. Click on the Template button and select a directory to create the new data file. Then, give the new data file a name and click OK.



5. The new data file will be created with a sample entry and displayed using Notepad as shown below.



6. After adding the required sizes to the data file, close Notepad. Then, click on OK on the Edit Component dialog to add the new component to the spec.



7. To add the same component to other specifications, simply add the component as outlined earlier except use the Select button to select the newly created data file.

Changing a Component's Spec

A component's specification can be changed through a number of ways:

1. Using the CEDIT (see page 304) command, the user can select a component and then select the specification change button to select a different specification.
2. Using the SPECCHG (see page 271) command, the user can filter/select one or more components and change the specification.
3. If the live database (see page 231) is being used, the user can change the specification in the database and use the SYNC (see page 233) command to update the drawing from the database.

Note: Caution should be used when changing a 150# based specification to a 300# based specification. This function will change the associated data file and the next mode conversion will show problems of over lapping components due to lengths that are different in the two specifications. A 300# RFWN flange is longer than a 150# RFWN flange.

Creating a Base Profile

To create a base profile, the user can copy his AutoCAD icon or CADWorx Plant icon and give it a profile which will be based on the default AutoCAD profile.

1. Copy the AutoCAD or CADWorx Plant icon to a new icon on the desktop. This can also be copied into the start menu. Refer to the Windows documentation for example and procedures on copying.
2. Edit this icon by right clicking on it.
3. From the context menu, choose the Properties from the bottom.
4. Pick the Shortcut tab.
5. In the Target edit box, modify the profile at the end of the target. For example, "C:\Program File\Autodesk\AutoCAD 2011\acad.exe" /p UserBase (or any name which does not exist under the Profile tab in the Options dialog within AutoCAD). Make sure to place the profile command switch outside of the quotes as shown. The quotes will not be required if the name of the AutoCAD directory is not a long file name (spaces, etc.).
6. Pick the OK button.
7. Start AutoCAD with this icon.
8. AutoCAD will display a message indicating that it can not find this profile and will create it based on the AutoCAD defaults.

Modifying Specification and Data File Directories

CADWorx Plant uses a dynamic and flexible directory system for storing specifications and related data files. By default, all CADWorx specifications are stored in the Spec directory in the CADWorx Plant installation directory. Imperial data files are stored in the LIB_I directory, mixed-metric data files are stored in LIB_M, and pure metric data files are stored LIB_MM directories.

The user can relocate these directories to other drives on the same computer or to shared network drives. The specification directory i.e. the directory where CADWorx Plant looks for specification files (*.spc) can be changed by modifying the variable SpecificationDirectory (see page 63) stored in the current configuration file.

The path to individual data files is built dynamically by CADWorx Plant. The system was designed to allow the most flexibility in locating data files any where the user chooses. The LibraryDirectory (see page 59) variable in the configuration file stores the location of all data files. Individual specifications contain the library sub-directory (LibrarySubDirectory entry in the spec file) where that specification's data files are located.

Following is an example of how paths to data files are built dynamically by CADWorx Plant.

LibraryDirectory=C:\CADWorx Plant (Stored in the configuration file)

LibrarySubDirectory=LIB_I (Stored in the specification and can be accessed through the Info button in the Spec Editor)

DataFileSubDirectory=STD (Stored with each component in the spec)

DataFileName=PIP_W (Stored with each component in the spec)

Full data file path = LibraryDirectory + LibrarySubDirectory + DataFileSubDirectory + DataFileName.DataFileSubDirectory

Full data file path = C:\CADWorx Plant + LIB_I + STD + PIP_W.STD

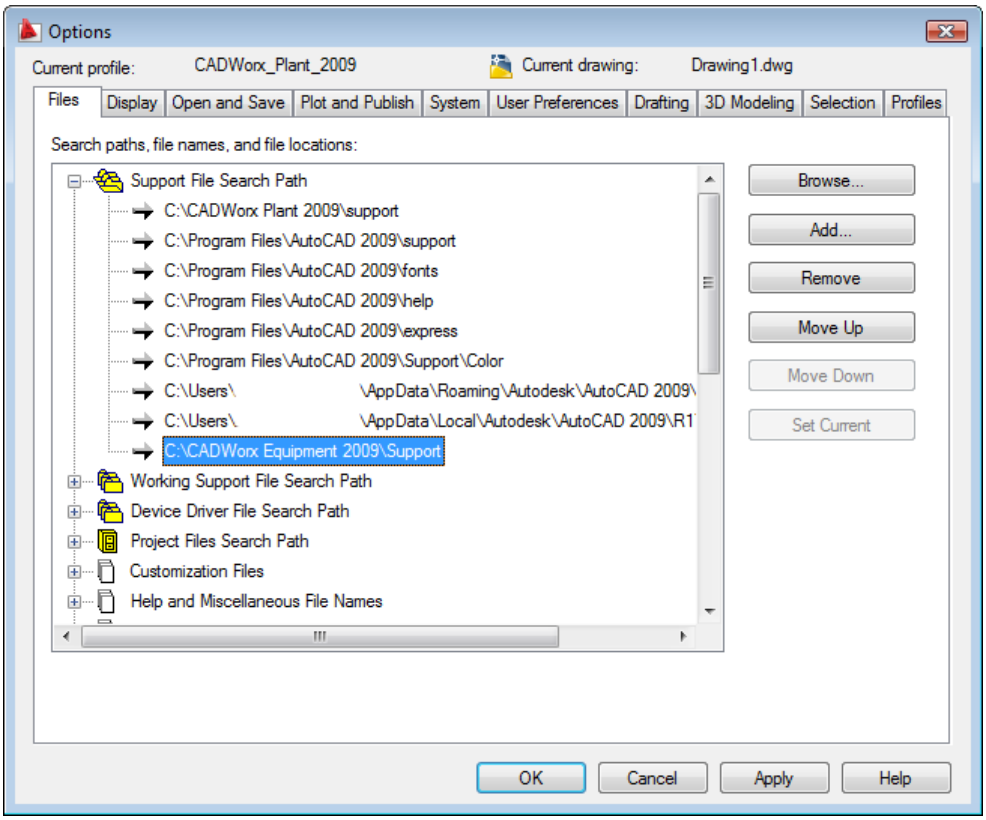
or C:\CADWorx Plant\LIB_I\STD\PIP_W.STD

Running CADWorx Equipment or PV Fabricator with CADWorx Plant

The following steps show how to run CADWorx Plant and CADWorx Equipment simultaneously within the same AutoCAD session. This will be necessary if equipment created with CADWorx Equipment needs to be modified while in the CADWorx Plant drawing session.

1. Start CADWorx Plant and go to AutoCAD Tools pull down menu -> Options.
2. In the Options dialog, select the Files tab and expand the Support File Search Path item.

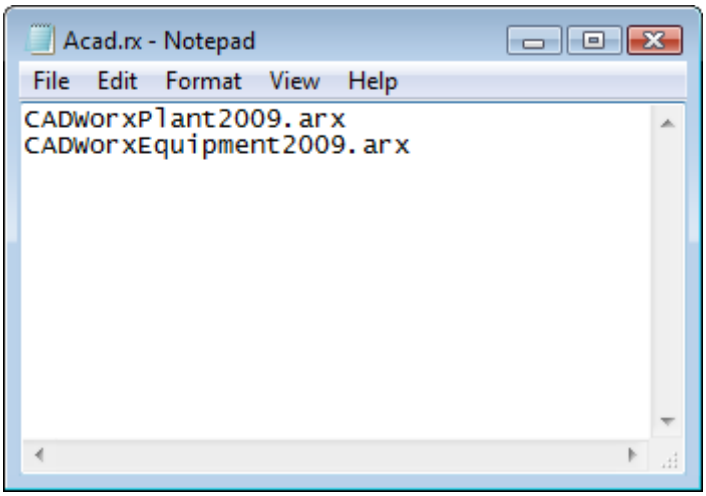
3. Click on Add button and browse to the CADWorx Equipment support directory (e.g. C:\CADWorx Equipment\Support) as shown below. Close CADWorx Plant.



4. Start Windows Explorer and browse to the CADWorx Plant support directory (e.g. C:\CADWorx Plant\Support).

5. Locate the file ACAD.RX and open it using NOTEPAD.EXE

6. Type CADWorxEquipment2010.arx into this file below the entry for CADWorxPlant2010.arx. If CADWorxEquipment2010.arx is above CADWorxPlant2010.arx, CADWorx Plant tool tips will not function. Save changes to this file and close the file.



7. Start CADWorx Plant. CADWorx Equipment will also start simultaneously. If the Equipment pull down menu does not load, run the command LOADMENUEQUIP.

Special Monitored Commands

Special Monitored Commands used with Live Database Features

This list of commands is monitored by CADWorx Plant for live external database synchronization. When any of the AutoCAD commands are executed, CADWorx Plant will automatically update a live external database accordingly.

They are divided into three groups:

“Copy Type Commands” that add new graphically entities into the model.

"Move Type Commands" that move the graphically entities within the model

“Erase/Unerase Type Commands” that remove/un-remove graphical entities.
“Erase/Unerase Type Commands” will mark rows in the database for deletion/un-deletion. If a CADWorx Plant graphical entity has been deleted from the drawing, the row from the database will be deleted when drawing file has been closed.

Erase/Un-Erase Type Commands

1. ERASE
2. U
3. UNDO
4. REDO
5. MREDO
6. OOPS
7. BLOCK
8. -WBLOCK
9. EXPLODE
10. CUTCLIP
11. GROUP

Copy Type Commands

1. COPY
2. BREAK
3. ARRAY
4. GRIP_COPY
5. GRIP_MIRROR
6. EXPLODE

7. MIRROR
8. PASTECLIP
9. PASTEORIG
10. DROPGEOM

Move Type Commands

1. MOVE
2. GRIP_STRETCH
3. STRETCH
4. DROPGEOM

APPENDIX A

Data Files

This Appendix discusses the data files used in CADWorx Plant. It also provides the criteria for making these files for each component.

In order for components to be dimensionally correct, CADWorx Plant must access a data file to determine the correct lengths and thickness of the components. When the user selects a component, CADWorx Plant checks the current specification to determine which file to access; then retrieves data from that file. It then uses the data to draw the component based on the current size.

The user can create new data files or modify existing files as long as the structure of the file is maintained. They can also use the Template feature in the Specification Editor (see page 254) to create templates for new data files that can then be easily completed to include the sizes the user requires. The parentheses and quotation marks are very important. See examples below.

The first line is a description (component, end connection, rating, manufacturer, and model no., when available). The user can put any description here.

```
BALL VALVE, 300LB FLG, LONG PATTERN (WKM)
```

The second line is the components' density.

```
0.283              DENSITY
```

The third line is intentionally left blank and the fourth line is the column header. This is a list of strings, that must start with an open parenthesis and end with a closing parenthesis. The actual column descriptions can be anything the user chooses, but must be enclosed in quotes.

```
("NOM.SIZE" "O.D." "FLG O.D." "FLG THK." "LENGTH" "WGT")
```

The fifth line is a line of equal signs that separate the actual data from the description and column header. If it is not present or is different, the program will not function properly.

```
==============================
```

The actual lines of data are a list and must be enclosed in parentheses. When the user selects a ball valve CADWorx Plant gets the flange outside diameter from the third column, the flange thickness from the forth column, etc. If the user rearranges the columns they may get strange looking components.

```
( 2.0000   2.3750   8.5000   1.7500  14.5000  125.00 )
```

Every data file needs a weight column. Weights are necessary for the center of gravity generator. If the user does not know the weight, they will need to put some fictitious numbers there. If they put a zero in the weight column, the program will just skip that component. The weights in CADWorx Plant data files were taken from the manufacturer listed in the description line. Where weights were unavailable, they were calculated and “EST WGT” will appear in the description line.

Long file names are permitted and any descriptive name the user wishes is allowed. The default file names are set up as follows:

```
BAL_F_L.900
Component (BALL valve)
End Connection (raised Face)
Other (Long pattern)
Rating (900 lb.)
```

The data file structure that is required for each component is listed in the <INSTALL-DIR>\SYSTEM\DATAFILE.TBL. This file is used in generating template data file when adding new components from within the Specification Editor (see page 254).

APPENDIX B

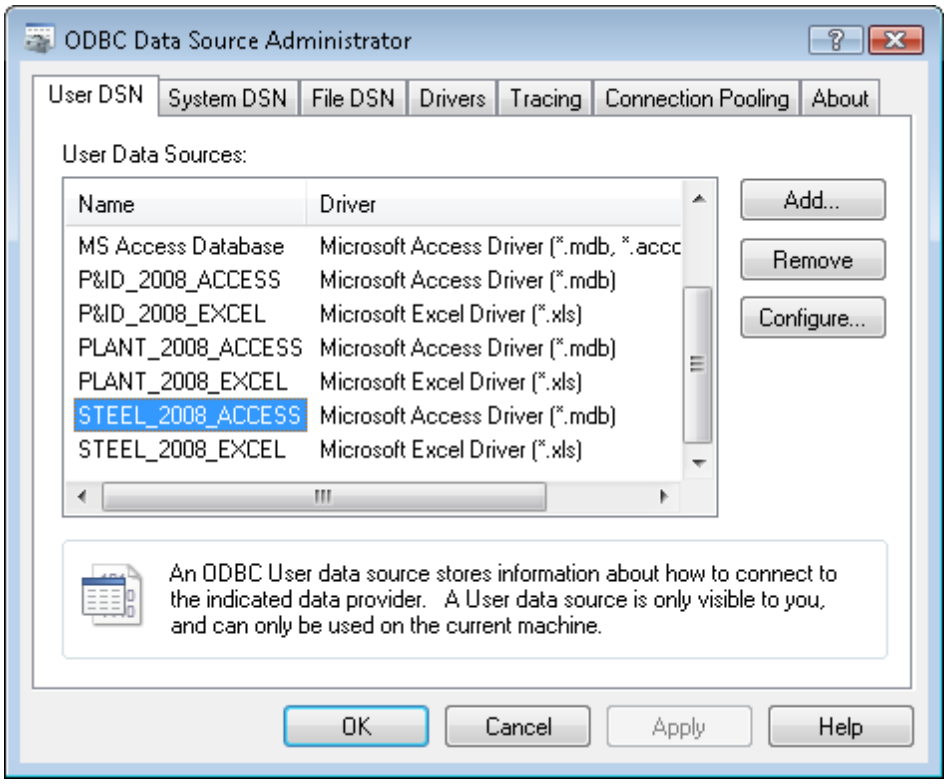
ODBC Drivers

This Appendix covers all the ODBC (Open DataBase Connectivity) information regarding the CADWorx products. Databases associated with CADWorx Plant will all use ODBC.

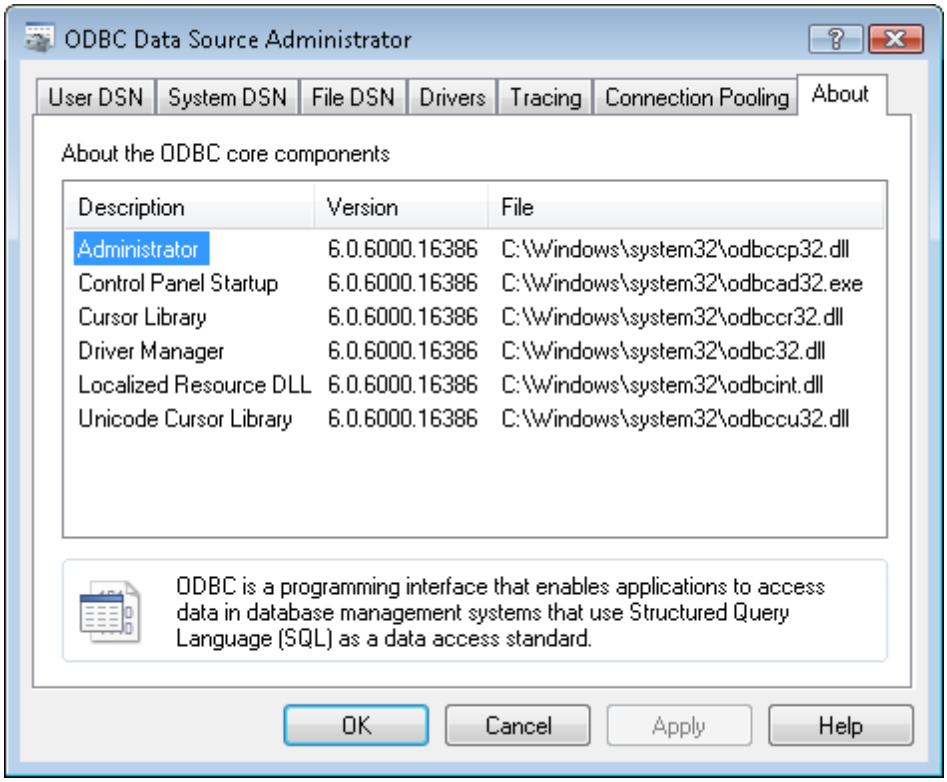
If the machine on which CADWorx Plant is going to be installed does not have the latest version of the ODBC drivers, these drivers should be obtained from the Microsoft website (http://www.microsoft.com/data).

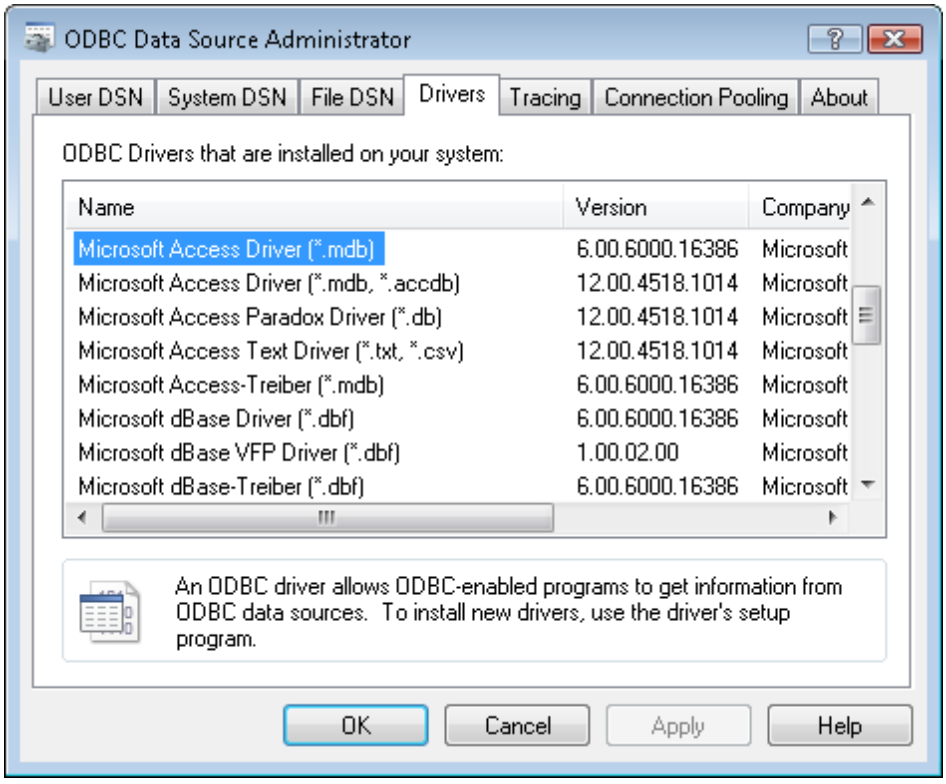
To determine what version of ODBC drivers are installed on the user's machine, pick the Windows Start Button -> Settings -> Control Panel -> Administrative Tools -> Data Sources (ODBC). Double click on the icon and then select the About tab. The versions installed on the machine should be 3.50 or greater. If the files on the user's machine are older, install the latest updates. The user MUST do this before installing CADWorx Plant.

After installing the program the CADWorx Plant User Data Sources will appear as shown below.

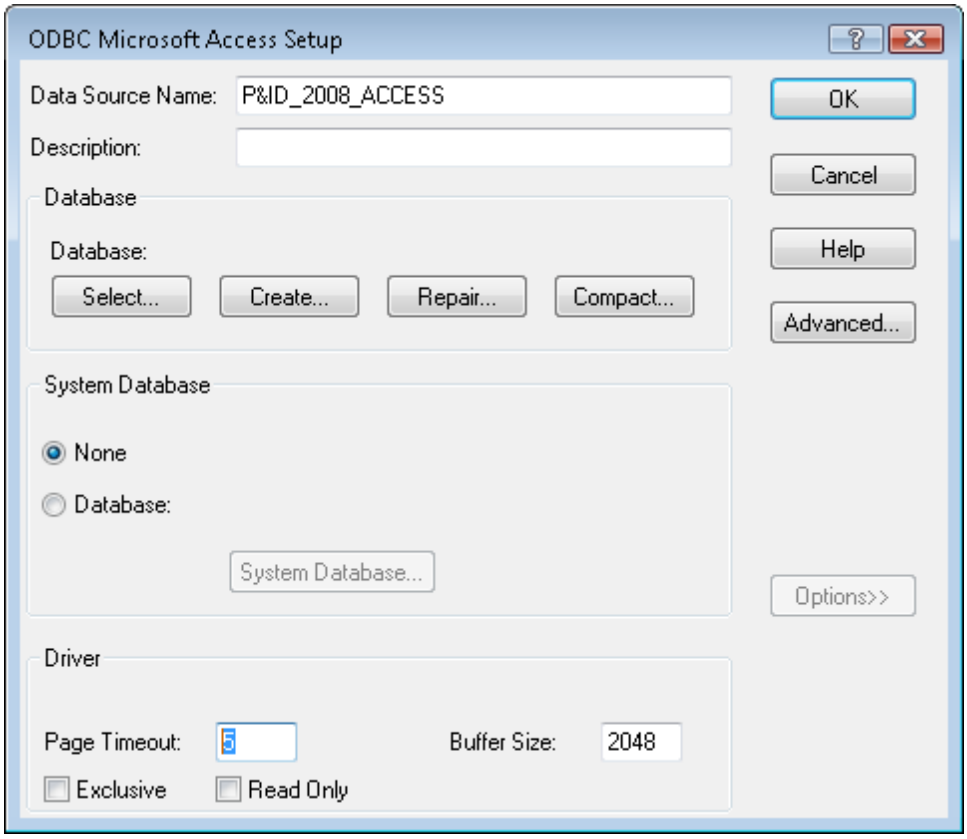


ODBC drivers configured correctly will insure that CADWorx Plant will perform correctly. The following figure shows the ODBC Driver information.

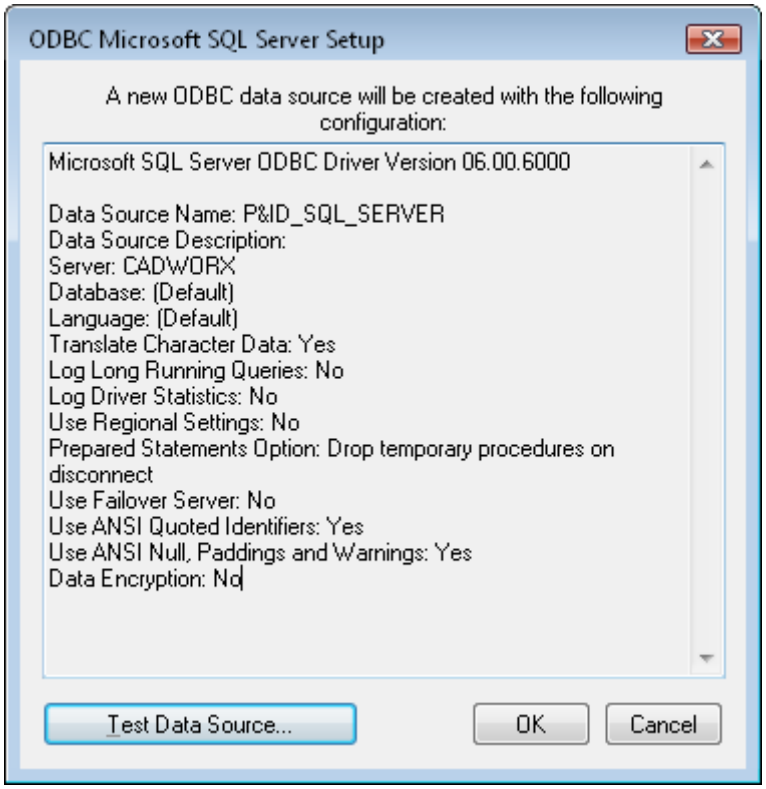




The following figure shows the Access data source.


ODBC Microsoft Access Setup
Data Source Name:
P&ID_2008_ACCESS
Description:
Database
Database:
Select...
Create...
Repair...
Compact...
System Database
None
Database:
System Database...
Driver
Page Timeout:
5
Buffer Size:
2048
Exclusive
Read Only
OK
Cancel
Help
Advanced...
Options>>

To setup a SQLServer Data Source Name, review the instructions in the file ..\CADWorx\System\SQLServer.txt. The following figure shows the SQL Server data source.



APPENDIX C

Misc. Support Files

This Appendix lists the miscellaneous support files used in CADWorx Plant. It also explains how these files are loaded and where they can be located.

IMPERIAL.CFG (and METRIC.CFG)

This default file is located in the \SYSTEM sub-directory of CADWorx Plant. This file can be located anywhere the user desires. At startup, CADWorx Plant locates this configuration file from the registry. Entering Setup (see page 18) will allow the user to save this file in any location. The user can have multiple configuration files saved in Project directories or any other location.

SETSIZE.TBL, SETSIZEA.TBL, and SETSIZEM.TBL

These files are located in the <INSTALL-DIR>\SYSTEM of CADWorx Plant. The files contain the nominal pipe sizes commonly used. SETSIZE.TBL is used for Imperial or English pipe sizes. SETSIZEM.TBL is for metric sizes and SETSIZEA.TBL is for metric sizes with English input. The user can add to these files if needed. Assuming the program was install on <INSTALL-DIR> listing the file <INSTALL-DIR>\SYSTEM\SETSIZE.TBL.

APPENDIX D

Component Groups

This Appendix lists all the common prompts used in the program. In these examples, a pipe will be addressed the same as a valve as the prompts are very similar for these components. An elbow will be the same as an angle or relief valve. The relief valve will have other options which is addressed in the appropriate toolbar topic.

The <last point> option allows the user to hit an enter which then finds the last component placed in the drawing. It will automatically connect the component to this previous placed component. If it is the start of a new drawing session or there are no other components in the drawing, this last point will not exist and the program prompts the user to pick a location.

Point not available. Please pick point:

The prompts shown in this Appendix are the default prompts only. For starting a component at its corner or center can be easily applied at the individual component command prompts. These prompts will only appear at the command line and it is suggested when working with CADWorx Plant that at least 3 visible command lines are showing.

Note: Commands listed here show general placement options and do not show the other options which might help place or position the component. These other options might allow for different lengths, starting locations, and others to be applied as the component is placed.

Type 1

This topic explains the correct procedure for placing components that have two point placements. Most components will prompt for a starting location and then a direction. Some, such as a pipe component, will ask for two exact points which will determine the length and direction.

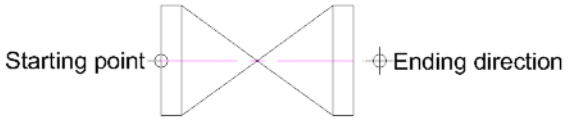
A typical pipe type component example...

Command: *<comand name>*
Pick start point <last point>: *pick a starting point or enter for last point*
Pick end point: *pick the ending point*



A typical valve type component example...

Command: *<comand name>*
Pick start point or [Center] <last point>: *pick a starting point or enter for last point, a center option is also available*
Pick direction: *pick direction*



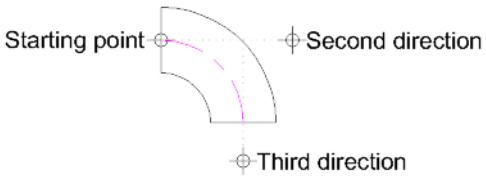
Type 2

This topic explains the correct procedure for placing components that have three point placements. Other components, such as a relief valves, have optional starting locations like the relief and pressure ends. Most components here will prompt for a starting location and then two directions. The directions do not have to be 90 degree apart as shown below -- therefore the user can work with Ortho off. If the two direction are entered along the same path, a message will appear that indicates both direction are colinear and the user will have to pick the directions again.

All points picked are colinear, please try again...

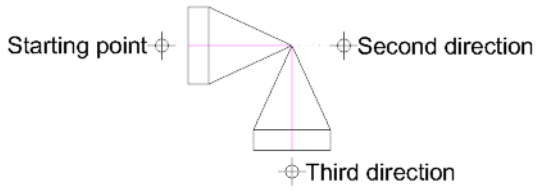
A typical 90 degree elbow type component example...

Command: *<comand name>*
Pick start point or [Corner] <last point>: *pick a starting point or enter for last point, a corner starting option is also available*
Pick corner direction: *pick the second direction toward corner*
Pick other direction: *pick the third direction*



A typical angle valve type component example...

Command: *<comand name>*
Pick start point or [Corner] <last point>: *pick a starting point or enter for last point, a corner starting option is also available*
Pick corner direction: *pick the second direction toward corner*
Pick other direction: *pick the third direction*



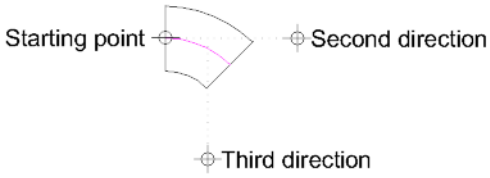
Type 3

This topic explains the correct procedure for placing components that have three point placements and are not exactly at a 90 degrees as discussed previously. Most components here will prompt for a starting location and then two directions. The directions do not have to be 90 degree apart as shown below -- therefore the user can work with Ortho off. If the the two direction are entered along the same path, a message will appear that indicates both direction are colinear and the user will have to pick the directions again.

All points picked are colinear, please try again...

A typical 45 degree elbow type component example...

Command: *<comand name>*
Pick start point or [Corner] <last point>: *pick a starting point or enter for last point, a corner starting option is also available*
Pick corner direction: *pick the second direction toward corner*
Pick other direction: *pick the third direction*



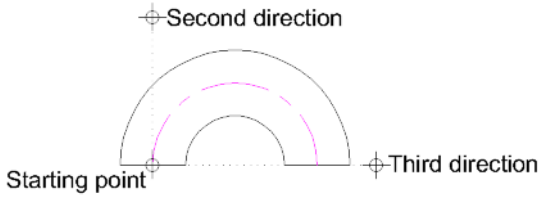
Type 4

This topic explains the correct procedure for placing components that have a different type three point placement. Most components here will prompt for a starting location and then two directions. The directions do not have to be 90 degree apart as shown below -- therefore the user can work with Ortho off. If the the two direction are entered along the same path, a message will appear that indicates both direction are colinear and the user will have to pick the directions again.

All points picked are colinear, please try again...

A typical return type component example...

Command: *<comand name>*
Pick start point or [Center] <last point>: *pick a starting point or enter for last point, a center starting option is also available*
Pick corner direction: *pick the second direction toward corner*
Pick other direction: *pick the third direction*



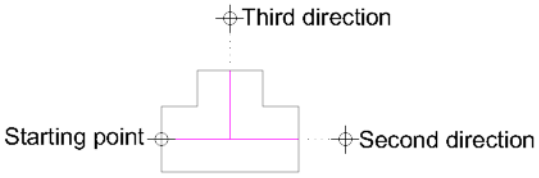
Type 5

This topic explains the correct procedure for placing components that have a branch type three point placement. Most components here will prompt for a starting location and then two directions. The directions do not have to be 90 degree apart as shown below -- therefore the user can work with Ortho off. If the the two direction are entered along the same path, a message will appear that indicates both direction are colinear and the user will have to pick the directions again.

All points picked are colinear, please try again...

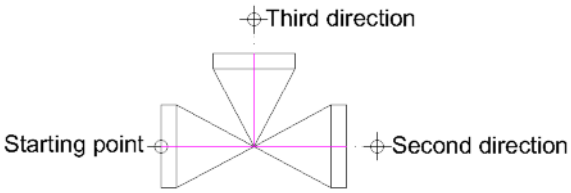
A typical tee type component example...

Command: *<comand name>*
Pick start point or [Branch/Center] <last point>: *pick a starting point or enter for last point, a center or branch starting option is also available*
Pick main end direction: *pick the second direction along the main direction*
Pick branch end direction: *pick the third direction toward branch*



A typical 3-way valve type component example...

Command: *<comand name>*
Pick start point or [Branch/Center] <last point>:*pick a starting point or enter for last point, a center or branch starting option is also available*
Pick main end direction: *pick the second direction along the main direction*
Pick branch end direction: *pick the third direction toward branch*

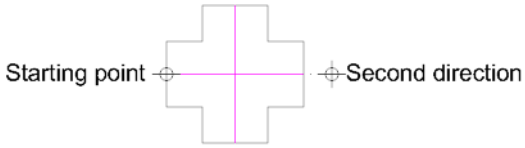


Type 6

This topic explains the correct procedure for placing components that have a branch type two point placement. Most components here will prompt for a starting location and then a single direction.

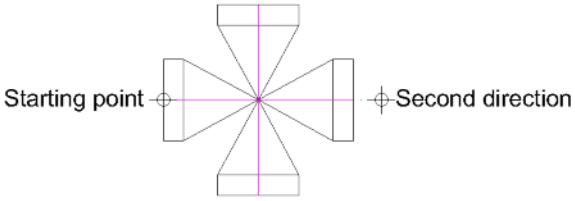
A typical cross type component example...

Command: *<comand name>*
Pick start point or [Center] <last point>: *pick a starting point or enter for last point, a center starting option is also available*
Pick main end direction: *pick the second direction along the main direction*



A typical 4-way valve type component example...

Command: *<comand name>*
Pick start point or [Center] <last point>: *pick a starting point or enter for last point, a center starting option is also available*
Pick main end direction: pick the second direction along the main direction



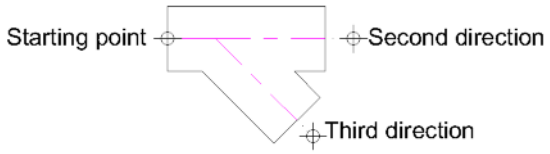
Type 7

This topic explains the correct procedure for placing components that have a branch type three point placement. Most components here will prompt for a starting location and then two directions. The directions do not have to be 45 degree apart as shown below. If the the two direction are entered along the same path, a message will appear that indicates both direction are colinear and the user will have to pick the directions again.

All points picked are colinear, please try again...

A typical lateral type component example...

Command: *<comand name>*
Pick start point or [Branch/Center] <last point>: *pick a starting point or enter for last point, a center or branch starting option is also available*
Pick main end direction: *pick the second direction along the main direction*
Pick branch end direction: *pick the third direction toward branch*



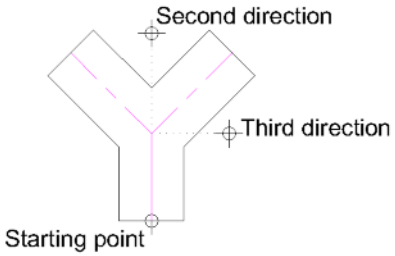
Type 8

This topic explains the correct procedure for placing components that have a wye type three point placement. Most components here will prompt for a starting location and then two directions. The directions do not have to be 45 or 90 degree apart as shown below. If the the two direction are entered along the same path, a message will appear that indicates both direction are colinear and the user will have to pick the directions again.

All points picked are colinear, please try again...

A typical wye type component example...

Command: *<comand name>*
Pick start point or [Branch/Center] <last point>: *pick a starting point or enter for last point, a center or branch starting option is also available*
Pick main end direction: *pick the second direction along the main direction*
Pick branch end direction: *pick the third direction toward branch*



APPENDIX E

Aliases and Command Names

All functions that are accessed from the pulldown menu or the toolbars have a command line equivalent function.

The following aliases are provided for CADWorx Plant. These aliases are copied from the <INSTALL-DIR>\SUPPORT\PIPE_ALT.PGP to the users ACAD.PGP field during installation. If desired, change any alias located here to avoid conflicts with aliases already defined in AutoCAD.

Command	Alias
3DFOLLOW	3DF
AUTOBOLT	AB
AUTOBREAK	ABR
ANCHOR	ANC
ANOTELEV	ANE
ARROWS	ARR
ANOTCOMP	ATC
ISOCFG	AUC
AUTODIM	AUD
AUTOGASKET	AUG
AUTOISO	AUI
BREAKDIST	BKD
BOMCUT	BMC
BOMDELETE	BMD
BOMEXPORT	BME
BOMIMPORT	BMI
BOMTOTAL	BMT
COMP2SPEC	C-S
C2ISO	C2I
C2MULTI	C2M
C2IN	C2N
C2OUT	C2O
C2TEXT	C2T

C2MEMORY	C2Y
COMPASS	CC
CONVERTCL	CCL
CONVERTDOUBLE	CDL
CEDIT	CE
CHANGELEV	CHE
CONVERTISO	CI
CONNECTTOG	CNT
CONTINUITY	CON
CONVERTSOLID	CS
CONVERTSINGLE	CSL
CURRENT	CU
DIMADJUST	DAJ
DBAUDIT	DBA
DBFGEN	DBF
DIMCOORD	DCD
DETAILS	DET
DIMTAILH	DTH
DIMTAILV	DTV
ELEV2PLAN	E-P
ENDBREAK	EB
ELBOWCL	ECL
EASTELEV	EEL
ELEMENT	ELE
ELEVATIONS	ELV
ENDFLG	ENF
ENDHIDDEN	ENH
ENDPIPE	ENP
ENDPLAIN	ENPL
EXIST	EXI
FITTINGWIDTH	FWD
GCEDIT	GCE
GEARELEV	GEE
GENERIC	GEN
GEARPLAN	GEP
GUIDE	GUI
INSTRUMENTS	IMT

INSERTMODE	INM
INSULATION	INS
ISOFLAG	ISF
ISOOUT	IX
LEVERELEV	LEE
LISTFILE	LF
LINEISOLATE	LIS
LINEPIPE	LIP
LISP	LL
LAYERRESTORE	LRS
LEVERPLAN	LVP
LINEWIDTH	LWD
LAYERCHANGE	LYC
LAYERDELETE	LYD
LAYEROFF	LYF
LAYERISOLATE	LYI
LAYERON	LYO
LAYERSET	LYS
LAYERUNISO	LYU
LINEISOLATE	LIS
MATCH	MAT
MODIFYLIB	MDL
TAGINSERT	TIN
TAGRELOCATE	TRL
TAGTOGGLE	TTO
NUMBER+	NB+
NUMBER-	NB-
NUMBERASSIGN	NBA
NUMBERANNOTATE	NBAN
NUMBERCOUNT	NBC
NUMBERSET	NBS
NUMBERSETUP	NBSH
NODEPLACE	NDP
NORTHELEV	NEL
NONSTDBOLT	NSB
OFFSETCL	OFC

OPERELEV	OPE
OPERPLAN	OPP
OSYELEV	OSE
OSYPLAN	OSP
OVERLAY	OVL
PREPARE	PAR
PLACEBOLT	PB
PCFIN	PCF
PIPEIN	PI
PLANS	PLA
PIPEOUT	PX
REINFPAD	RFP
REINFSAD	RFS
ROLLEDFLG	RLF
ROLLEDHIDDEN	RLH
ROLLEDPLAIN	RLP
ROTATIONAL	ROT
ROUTER	RT
ROUTETHRD	RTHD
ROUTEBWLR	RTLR
ROUTEBWSR	RTSR
ROUTESW	RTSW
SETANGLE	SA
SECTIONS	SECT
SOUTHELEV	SEL
MAIN	SM
SNUBBER	SNU
SPECCHG	SPCH
SPECEDIT	SPED
SPECOPTION	SPO
SPRING	SPR
RED	SR
SPEC	SS
SETUP	STP
TEXTHEIGHT	THT
TRANSLATIONAL	TRN
UCSNEXT	UNX

USER	USE
VIEWCLIP	VC
VIEWEAST	VE
VESSEL	VES
VIEWFLAT	VF
VIEWNORTH	VN
VESSELOD	VOD
VIEWSOUTH	VS
VIEWWEST	VW
WESTELEV	WEL
WELDGAP	WG
WELDSIZE	WS
WELDTOTAL	WT
XCEDIT	XCE
ZOOMF	ZF
ZOOMLOCK	ZL

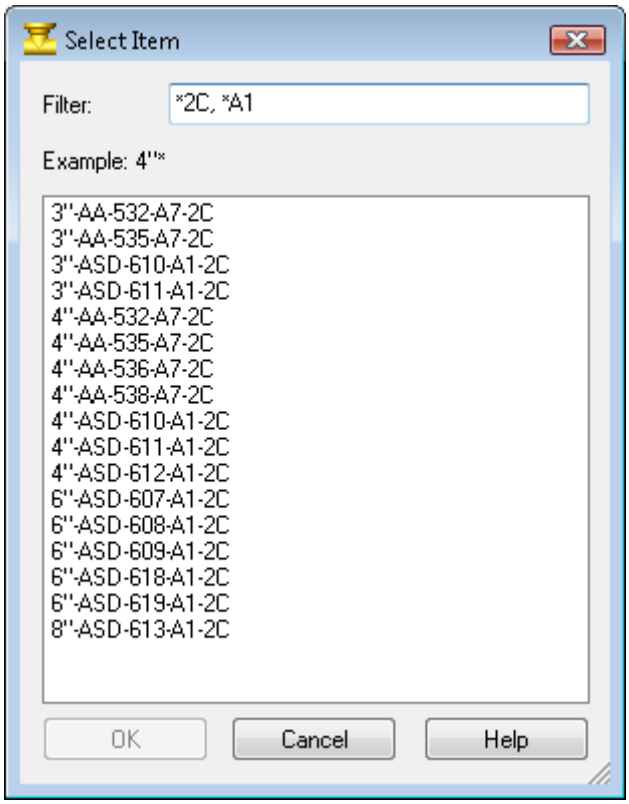
APPENDIX F

Common List Dialog

This appendix explains the use of a very common dialog. It's explanation is located here due to the multiple references to it throughout this user guide.

This dialog lists all the line number or specifications in the drawing (or database). It is available as an option in various commands and functions. Multiple items can be selected using the CTRL or SHIFT keys. A *filter* option is available as well to filter by any user criteria.

Note: The filter is case sensitive.



Wild-card characters	
Character	Definition
# (pound)	Matches any single numeric digit
@ (at)	Matches any single alphabetic character
. (period)	Matches any single nonalphanumeric character
* (asterisk)	Matches any character sequence, including an empty one, and it can be used anywhere in the search pattern: at the beginning, middle, or end
? (question mark)	Matches any single character
~ (tilde)	If it is the first character in the pattern, it matches anything except the pattern
[...]	Matches any one of the characters enclosed
[~...]	Matches any single character not enclosed
- (hyphen)	Used inside brackets to specify a range for a single character
, (comma)	Separates two patterns
` (reverse quote)	Escapes special characters (reads next character literally)

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