

PVFabricator

User's Guide



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Preface for PV Fabricator

This document provides command reference information and workflows for PVFabricator.

We welcome comments or suggestions about this documentation. You can send us an email at: PPMdoc@intergraph.com.

Documentation updates for supported software versions are available from *eCustomer* (<https://www.crmweb.intergraph.com>).

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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
[<i>Product Folder</i>]	The PVFabricator installation folder on your computer, for example, C:\PVFabricator. Substitute your installation folder anywhere this notation is used.
[<i>AutoCAD Folder</i>]	The AutoCAD installation folder on your computer, for example, C:\AutoCAD 2000i. Substitute your installation folder anywhere this notation is used.
[<i>CD-DRIVE</i>]	The CD-ROM drive on your computer. 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	User responses to command prompts.
Commands	PVFabricator and AutoCAD commands.
Options	Command line options.
CAPITALS	AutoCAD system variables or keywords.
Key + Key	Key combinations for which you must press and hold down one key and then press another, for example, CTRL+P or ALT+F4.

Technical Support

For the latest information on PVFabricator, visit *Intergraph CADWorx and Analysis Solutions* (<http://www.coade.com/>).

For current on-line support information, including knowledge base, technical tips, and downloads, visit *eCustomer* (https://crmweb.intergraph.com/ecustomer_enu).

You can contact Intergraph CAS Technical Support or Sales:

- *ICAS Dealer Support* (<http://www.coade.com/Support/Dealers.shtml>) or ICAS General Support (<http://support.intergraph.com/Default.asp>)
- **Technical Support E-mail:** ppmcrm@Intergraph.com
- **Phone:** 1-800-766-7701 (CADWorx Direct), 280-890-4566 (General)
- **Fax:** 281-890-3301
- **Sales E-mail:** sales.icas@intergraph.com

In order to provide technical support, Intergraph CAS needs to know what version of PVFabricator, AutoCAD, and Windows you are using. To determine this information, follow the steps outlined for the product you are using:

CADWorx Product	Steps
Plant, P&ID, fieldPipe, or Steel	<ul style="list-style-type: none">▪ Run the CADWORXABOUT command. A dialog box opens and displays the version information.▪ Close the dialog box to send the required information to the AutoCAD command line.▪ Copy the information from the command line, and email it to Intergraph CAS.
Equipment or PV Fabricator	<ul style="list-style-type: none">▪ Run the EQUIPABOUT command. A dialog box opens and displays the version information.▪ Close the dialog box to send the required information to the AutoCAD command line.▪ Copy the information from the command line, and email it to Intergraph CAS.
Internet Publisher	Version information is displayed at the AutoCAD command line when you start the software.
Datasheets	Click Help > About to display a dialog box that contains the required information.

In addition to the information above, the table below lists the additional files that are usually required to assist Intergraph CAS in resolving your technical issues:

CADWorx Product	Additional Files Required by Intergraph CAS Support
CADWorx Plant (Piping, Steel, HVAC)	<p>Email the AutoCAD model drawing (DWG) files.</p> <ul style="list-style-type: none"> ▪ If the model consists of several line numbers, indicate in the email which line numbers you are having issues with. ▪ If the model consists of many drawing files with dependent reference drawings (XREFs), use the AutoCAD ETRANSMIT command to create a ZIP file of all the drawings used in the master model.
CADWorx Plant (component data file creation)	Email information about the part's dimension from the manufacturer's catalog.
CADWorx ISOGEN	<p>Email the AutoCAD model drawing (DWG) files, any isometrics generated by ISOGEN, and the ISOGEN style files.</p> <ul style="list-style-type: none"> ▪ If the model consists of several line numbers, indicate in the email which line numbers you are having issues with. ▪ The ISOGEN.FLS file located in the ISOGEN style folder lists all the files used (*.OPL, *.MLS, *.FLS, *.DDF, *.POS, *.ALT, *.DWG*.DXF, *.FDF, *.ATT). Email all files listed in the ISOGEN.FLS file, including the ISOGEN.FLS file.
CADWorx Equipment	Email the AutoCAD model drawing (DWG) files.
CADWorx Equipment to PV Elite interface	Email the AutoCAD model drawing (DWG) file and PV Elite input file (*.PVI).
PV Fabricator	Email the AutoCAD model drawing (DWG) files and PV Elite input file (*.PVI).
CADWorx P&ID or CADWorx IP	<p>Email the complete P&ID project, which includes Project.CFG, Database.TBL, Map.TBL, and all the DWG, and MDB files. Usually, all of these files are located in one folder.</p> <ul style="list-style-type: none"> ▪ If the project has many P&ID drawing files, indicate which P&ID drawings and components you are having issues with. ▪ If you are experiencing database errors, include the file <i>[Product Folder]\P&ID\System\Sql_Error_Log.TXT</i>. ▪ If you are emailing MDB files, use a compression utility such as WinZIP. Most email servers have restrictions on transmitting MDB attachments.

CADWorx Steel	<p>Email the AutoCAD model drawing (DWG) files.</p> <ul style="list-style-type: none"> ▪ If the model consists of many components, indicate in the model which components or location you are having issues with.
CADWorx Datasheets	<p>Email the datasheets MDB file and any datasheet file generated (XLS).</p> <ul style="list-style-type: none"> ▪ If emailing MDB / XLS files, use a compression utility such as WinZIP. Most email servers have restrictions on transmitting MDB / XLS attachments.
CADWorx Plant to CAESAR II interface	<p>Email the AutoCAD model drawing (DWG) file and CAESAR II input file (*.A, *.C2).</p> <ul style="list-style-type: none"> ▪ Run the GETALLSPECDATAFILES command to copy all specifications and data files used in the model into sub-folders under where the model drawing file is located. ▪ Zip the contents of these folders and email the ZIP file to Intergraph CAS Support. ▪ If the model consists of several line numbers, indicate in the email which line numbers you are having issues with.
CADWorx Design Review or CADWorx Design Create	<p>Email the AutoCAD model drawing (DWG) files.</p> <ul style="list-style-type: none"> ▪ If the model consists of several line numbers, indicate in the email which line numbers you are having issues with. ▪ If the model consists of many drawing files with dependent reference drawings (XREFs), use the AutoCAD ETRANSMIT command to create a ZIP file of all drawings used in the master model.
CADWorx Design Viewer	<p>Email the AutoCAD model drawing (DWG) files and the *.CDR file.</p>

NOTES

- If you are attaching large or multiple files to your eCustomer log, use a compression utility such as WinZIP.
- If you cannot attach your drawing files, then try to duplicate the issue in a new drawing with as few components as possible. If the issue can be duplicated in a new drawing, attach the duplicate drawing to Intergraph CAS.
- For hardware lock issues, first verify that you have an Intergraph CAS hardware lock. CADWorx Plant Professional, CADWorx P&ID, CADWorx P&ID Professional, CADWorx Plant Ductile Iron, and CADWorx Plant Sanitary use the Intergraph CAS hardware lock for licensing. CADWorx Steel, CADWorx Steel Professional Plus do not use the Intergraph CAS hardware lock for licensing. For more information about hardware locks, see **CADWorx Hardware Lock Identification** in the *CADWorx Installation Guide*.

What's New in PVFabricator

Version 2014 (14.0)

This user's guide has been updated for this release with the following changes:

Items added and updated with PV Fabricator 2014:

Software Enhancements

- You can now create Tube Bundle details, in addition to the tubesheet and baffle details. You can generate, examine, and manipulate the details. For more information, see *Tubesheet Configuration* (on page 83), *Tube Bundle* (on page 90), *Configure a tubesheet, baffle, and tube bundle for detail* (on page 91), *Tube Bundle tab* (on page 94), and *Examine and Manipulate Tube Bundle details* (on page 102).
- You can now enter the nut properties for the tubesheet. For more information, see *Rod, Nut Properties* (on page 89).
- You can now remove or restore tube holes on a tubesheet. Tube holes can only be removed from tube sheets. For more information, see *Details Tab* (on page 79), *Remove Tube Hole* (on page 110), and *Restore Tube Hole* (on page 111).
- You can now change the hidden thread layer, color, and linetype in the **Section Settings**. For more information, see *Section* (on page 82), and *Examine detail settings* (on page 96).
- You can now update the detail BOM. For more information, see *Update Detail BOM* (on page 105).

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- Updated the rod, nut, and tube hole properties. The rod properties in the software were moved and combined into the **Rod, Nut Properties** tab. For more information, see *Rod, Nut Properties* (on page 89).
- Added more instruction on generating a support detail. For more information, see *Generate a support detail* (on page 95).

Items added and updated with PV Fabricator 2013:

Software Enhancements

- You can now create tubesheet and baffle configurations, generate the details, and examine and manipulate the details. For more information, see *Tubesheet Configuration* (on page 83), *Generate Tubesheet or Baffle details* (see "Generate Tubesheet, Baffle, or Tube Bundle details" on page 96), *Examine and Manipulate Tubesheet details* (on page 100), and *Examine and Manipulate Baffle details* (on page 101). The manual has been updated accordingly.
- You can now change the **Hatch Settings** for a tubesheet when examining the details. For more information, see *Examine detail settings* (on page 96).
- You can now create a bill of materials by **All**, **Selection**, or **append to Existing BOM**. For more information, see *Generate Detail BOM* (on page 103), *Generate BOM* (on page 51), and *Add a bill of materials to the drawing* (on page 52).
- You can now change a tube to a rod in a tubesheet. For more information, see *Change Tube To Rod* (on page 108).

- You can now change a rod to a tube in a tubesheet. For more information, see *Change Rod To Tube* (on page 108).
- You can now add a rod in a tubesheet. For more information, see *Add Rod* (on page 109).
- You can now remove a rod in a tubesheet. For more information, see *Remove Rod* (on page 109).
- You can now add a groove to a tubesheet. For more information, see *Make Tubesheet Groove* (on page 110).
- You can now remove a groove in a tubesheet. For more information, see *Remove Tubesheet Groove* (on page 110).
- You can now setup hatch settings in **Common Settings**. For more information, see *Hatch* (on page 83) and *Common Settings* (on page 81).
- You can now setup tag details for a baffle. For more information, see *Common Settings* (on page 81).

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- Updated the setup for the **Details** tab. For more information, see *Common Settings* (on page 81).
- Updated examining detail topic to include new settings for the Tubesheet. For more information, see *Examine detail settings* (on page 96).
- Updated all images and any command names that changed.

Items added and updated with PV Fabricator 2012:

- You can now examine, tag, and annotate the detail of supports for equipment. For more information, see *Support Detail Tab* (see "*Details Tab*" on page 79).
- You can now generate the support detail of a piece of equipment. For more information, see *Generate Detail* (on page 95).
- You can now examine and change generated detail by selecting it after generation. For more information, see *Examine detail settings* (on page 96).
- You can now update generated detail in the drawing. For more information, see *Update Detail* (on page 103).
- You can now generate a detail bill of materials for supports after generation. For more information, see *Generate Detail BOM* (on page 103).
- You can now generate tags to tag generated support detail in the drawing. For more information, see *Generate Tag* (see "*Generate Detail Tag*" on page 105).
- You can now change the tag settings of generated tags by selecting them after generation. For more information, see *Tag setting* (on page 105).
- You can now generate automatic tags to tag generated support detail in the drawing. For more information, see *Generate Automatic Tag* (see "*Generate Automatic Detail Tag*" on page 106).
- You can now place linear dimensions on a generated support detail. For more information, see *Linear* (see "*Add Linear Dimensions*" on page 106).
- You can now place aligned dimensions on a generated support detail. For more information, see *Aligned* (see "*Add Aligned Dimension*" on page 106).

- You can now place radius dimensions on a generated support detail. For more information, see *Radius* (see "Add Radius Dimension" on page 107).
- You can now place jogged dimensions on a generated support detail. For more information, see *Jogged* (see "Add Jogged Dimension" on page 107).
- You can now place angle dimensions on a generated support detail. For more information, see *Angle* (see "Add Angle Dimension" on page 107).
- You can now reset settings for all information in your generated support details. For more information, see *Reset Settings* (on page 113).

SECTION 1

Introduction

Intergraph PV Fabricator has advanced tools for the accurate and speedy production of pressure vessel fabrication deliverables. When linked to Intergraph PV Elite™, PV Fabricator greatly improves the workflow between engineer and designer to increase productivity and reduce the re-work and errors that cause costly overruns and penalties. By producing drawings quickly and more accurately, firms can increase throughput and customer satisfaction, thereby winning repeat business.

- **Setup** - Provides an overview about setup. For more information, see *PVFabricator Setup* (on page 25).
- **Vessel General Panel** - Provides descriptions of the commands on the **Vessel General** panel. For more information, see *Vessel General Panel* (on page 35).
- **BOM and Nozzle Schedule Panel** - Provides descriptions of the commands on the **BOM and Nozzle Schedule** panel. For more information, see *BOM and Nozzle Schedule Panel* (on page 51).
- **Tagging Panel** - Provides descriptions of the commands on the **Tagging** panel. For more information, see *Tagging Panel* (on page 55).
- **Details Panel** - Provides descriptions of the commands on the **Details** panel. For more information, see *Details Panel* (on page 59).
- **Annotation Panel** - Provides descriptions of the commands on the **Annotation** panel. For more information, see *Annotation Panel* (on page 67).
- **Dimensioning Panel** - Provides descriptions of the commands on the **Dimensioning** panel. For more information, see *Dimensioning Panel* (on page 73).
- **Fabricator Console** - Provides description of the **Fabricator** tab. For more information, see *Fabricator Console* (on page 75).
- **Details Tab** - Provides description of the **Details** tab. For more information, see *Details Tab* (on page 79).
- **Equipment Console** - Provides description of the **Equipment** tab. For more information, see *Equipment Console* (on page 115).

Please contact Technical Support for any questions/comments.

About PVFabricator

 Equipment Menu: **About**

 Command line: **EQUIPABOUT**

Displays the following information about the current version of the software:

- PVFabricator version and build date
- PVFabricator installation folder
- Windows version
- AutoCAD version
- Company name
- Hardware lock (ESL) number

SECTION 2

PVFabricator Workflow

The following steps represent a typical workflow using PVFabricator.


Import a model from PVElite

1. Click the **Placement** tab on the **Equipment** console.
2. Click **Import**.

*The **Select PVElite File** dialog box displays.*


3. Select the PVElite file to import.
The model imports.

Generate a 2D outline

1. Click a paper space tab.
2. Click **Generate 2D Outline** .
3. Select a viewport or press ENTER for **Automatic**.
4. Select **Yes** to turn off viewports or **No** to leave them active.


The software adds the outline graphics to your existing viewports.

Insert a title block

1. Click a paper space tab.
2. Click **Insert Title Block** .
3. Select the title block to place from the **Library** list.
4. Click **Insert**.


The title block displays in the layout.

Insert a weld detail

1. Click a paper space tab.
2. Click **Insert Weld Detail** .
3. Double click the weld detail to place from the **Library** list.
*The weld detail appears under **Selected**.*
4. Select the weld detail to place from the **Selected** list.
5. Click **Insert**.
6. Type **A** to insert weld details in an area, or type **I** to insert individual weld details.
7. Select a first corner point.
8. Select a second corner point.
9. Select the direction for the details.


The weld details display in the layout.

Insert other details

1. Click a paper space tab.
2. Click **Insert Other Detail** .
- The **Insert Other Details** dialog box displays.*
3. Select the details to place from the **Library** list, and drag them to the **Selected** list.
4. Click **Insert**.
5. Type **A** to insert the details in an area, or type **I** to insert individual details.
6. Specify first corner point.
7. Specify second corner point.
8. Specify the directions for the details.


The details display in the layout.

Insert a name plate

1. Click a paper space tab.
2. Click **Insert Name Plate** .
- The **Insert Name Plate** dialog box displays.*
3. Select the name plate to place from the **Library** list.
4. Click **Insert**.
- Pick a point.*
5. Specify the location for the name plate.


The name plate displays in the layout.

Insert a note


1. Click a paper space tab.
2. Click **Insert Note** .
- The **Insert Notes** dialog box displays.*
3. Double-click the note from the **Library** list.
- The note displays in the **Preview** area.*
4. Click **Insert**.
- Specify first corner.*
5. Select the first point for the note on the layout.
- Specify opposite corner.*
6. Select the second point for the note on the layout.

The note displays.

Insert a weld symbol


1. Click a paper space tab.
2. Click **Insert Weld Symbol** .
*The **Insert Weld Symbol** dialog box displays.*
3. Select the weld symbol from the **Library** list.
*The symbol displays in the **Preview** area.*
4. Click **Insert**.
Pick first point.
5. Select the first point for the weld symbol on the layout.
Pick second point.
6. Select the second point for the weld symbol on the layout.
The weld symbol displays.

Insert a callout

1. Click a paper space tab.
2. Click **Insert Callout** .
*The **Insert Callouts** dialog box displays.*
3. Select the callout from the **Library** list.
*The callout displays in the **Preview** area.*
4. Click **Insert**.
Pick first point.
5. Select the point for the leader line.
Pick second point.
6. Select the point for the callout.
The callout displays.

SECTION 3

PVFabricator Setup

 Equipment Tab: 

 PVFabricator Menu: **Setup**

 Command line: **EQUIPSETUP**

Specifies setup parameters for PVFabricator.

Layers - Displays the **Layer Control** dialog box.

Edit Config File - Displays the **Configuration File** dialog box.


Layer Control Dialog Box (on page [27](#))

Configuration File Dialog Box (on page [29](#))


What do you want to do?

- *Add a layer* (on page [25](#))
 - *Rename a layer* (on page [25](#))
 - *Delete a layer* (on page [26](#))
 - *Change the line type of a layer* (on page [26](#))
 - *Change the color of a layer* (on page [26](#))
 - *Edit a configuration variable* (on page [26](#))
 - *Restore a configuration file* (on page [27](#))
 - *Save a configuration file to a new name* (on page [27](#))
-


Add a layer

1. Click **Setup**  on the **Equipment** tab.
2. Click **Layers** on the **PVFabricator Setup** dialog box.
*The **Layer Control** dialog box displays.*
3. Type the name of the new layer in the **Name** box.
4. Click **Add**.

Rename a layer


1. Click **Setup**  on the **Equipment** tab.
*The **PVFabricator Setup** dialog box displays.*
2. Click **Layers** on the **PVFabricator Setup** dialog box.
*The **Layer Control** dialog box displays.*
3. Select the layer to rename.
4. Type the new name for the layer in the **Name** box.
5. Click **Update**.

Delete a layer


1. Click **Setup**  on the **Equipment** tab.
*The **PVFabricator Setup** dialog box displays.*
2. Click **Layers** on the **PVFabricator Setup** dialog box.
*The **Layer Control** dialog box displays.*
3. Select the layer to delete.
4. Click **Delete**.

NOTE If a layer is required, **Delete** is not available.


Change the line type of a layer

1. Click **Setup**  on the **Equipment** tab.
*The **PVFabricator Setup** dialog box displays.*
2. Click **Layers** on the **PVFabricator Setup** dialog box.
*The **Layer Control** dialog box displays.*
3. Click **Linetype**.
*The software displays the **Select Linetype** dialog box with all of the line types defined in the CFM.LIN and ACAD.LIN files found in the AutoCAD search path*
4. Select the line type to associate with the layer.
5. Click **OK** on the **Select Linetype** dialog box.


Change the color of a layer

1. Click **Setup**  on the **Equipment** tab.
*The **PVFabricator Setup** dialog box displays.*
2. Click **Layers** on the **PVFabricator Setup** dialog box.
*The **Layer Control** dialog box displays.*
3. Click **Color**.
*The **Select Color** dialog box displays.*
4. Select the color to associate with the layer.
5. Click **OK** on the **Select Color** dialog box.


Edit a configuration variable

1. Click **Setup**  on the **Equipment** tab.
*The **PVFabricator Setup** dialog box displays.*
2. Click **Edit Config File** on the **PVFabricator Setup** dialog box.
*The **Configuration File** dialog box displays.*
3. Select the variable to edit from the list.
4. Type the new value for the variable. For more information, see Startup Variables.
5. Click **Save**.

Restore a configuration file

1. Click **Setup**  on the **Equipment** tab.
*The **PVFabricator Setup** dialog box displays.*
2. Click **Edit Config File** on the **PVFabricator Setup** dialog box.
*The **Configuration File** dialog box displays.*
3. Click **Restore**.
*The **Select a Configuration to Use** dialog box displays.*
4. Select the configuration file to restore.
5. Click **Open** on the **Select a Configuration to Use** dialog box.

Save a configuration file to a new name

1. Click **Setup**  on the **Equipment** tab.
*The **PVFabricator Setup** dialog box displays.*
2. Click **Edit Config File** on the **PVFabricator Setup** dialog box.
*The **Configuration File** dialog box displays.*
3. Click **Save As**.
*The **Save Configuration As** dialog box displays.*
4. Specify the location and file name for the configuration file.
5. Click **Save**.

Layer Control Dialog Box

Specifies drawing layers and the default names, line types, and colors of the layers. Default layers are provided. You can add and delete layers.

NOTES

- The default layers are mandatory for use with PVFabricator and cannot be deleted. See the tables below.
- Layer changes made in this dialog box affect the current drawing session similar to the AutoCAD **LAYER** command.
- The commands in this dialog box are inactive if the configuration file permission is set to read-only.

Name - Specifies the layer name. The layer name can be anything that is allowed by AutoCAD. Wild card characters and spaces are not allowed. Click **Update** to change the name.

Add - Adds a new layer with a default name. New layers are created when the drawing is initialized with PVFabricator.

Update - Updates the selected layer to use the value in the **Name** box.

Delete - Removes the selected layer from the list. You cannot delete default layers.

Linetype - Specifies the line type for the selected layer. Select a value in the **Select Linetype** dialog box.

NOTE Line types are defined in the *[Product Folder]\PVFabricator\Support\Cfm.lin* and *[AutoCAD Product Folder]\UserDataCache\Support\acad.lin* files.

Color - Displays the **Select Color** dialog box for selecting the color of the layer.

CADWorx Plant, P&ID, PV Fabricator Layers

Layer#	Default Name	Layer is used for
Layer1	System	Non-graphical components (Gaskets, Bolts, TAP, OTAP)
Layer2	Border	Border drawing
Layer3	BOM	BOM graphics
Layer4	CI	Centerline of piping components
Layer5	Dim1	Dimensions
Layer6	Dim2	Dimensions
Layer7	Equip	PVPFabricator components
Layer8	Exist	Existing components
Layer9	Graph	Graphics annotations (Arrows, Section, Plan, Detail Labels)
Layer10	Steel	Steel components
Layer11	Text	Text annotations
Layer12	View1	Viewports
Layer13	CI_Steel	Centerline of steel components
Layer14	HVAC	HVAC components
Layer15	CI_HVAC	Centerline of HVAC components
Layer16	TopWorks	Valve TopWorks solid
Layer17	Clash	Clash entity
Layer18	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 is 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	PVFabricator
Layer12	Text	Text
Layer13	Graph	Graphics
Layer14	Exist	Existing
Layer15 to Layer100	(User Defined)	(User Defined)

NOTE If the **CI** and **CI_Steel** layers are frozen or turned off, some CADWorx commands might not function because data is stored on the centerline that is on these layers.

Configuration File Dialog Box

Specifies values for the selected configuration file.

Drawing Prototype - Specifies the drawing environment type.

- **English/Inch** - Restores the *[Product Folder]\PVFabricator\System\Imperial.cfg* file.
- **Metric/Inch** - Restores the *[Product Folder]\PVFabricator\System\Metric.cfg* file.
- **Metric/Metric** - Restores the *[Product Folder]\PVFabricator\System\Metric.cfg* file.

NOTES

- The default drawing template, Imperial.dwt (Metric.dwt for metric) is also stored in the registry. Each time the software is started, the default template is used.
- The selected drawing prototype only changes the setting for the default configuration files (Imperial.cfg or Metric.cfg). If you are using a different configuration file, use the SystemMeasure variable to change the drawing prototype setting.

List - Displays variables to change. Change variable values in the text box below the list. For more information, see Startup Variables.

Browse or **Settings** - Opens an additional dialog box to select a file or additional settings for the selected variable.

Save - Saves the current configuration file to its present location.

Save As - Saves the current configuration file to another location.

Restore - Restores a saved configuration file to the current drawing environment. This places the configuration file location in the registry so that it is used the next time PVFabricator is started.

Cancel - Cancels the operation.

NOTE If the configuration file permission is set to read-only, only the **Cancel** button is available.

Startup Defaults

When starting PVFabricator for the first time, you are prompted for the system of measurement to use in the drawing. This dialog box only displays the first time you start PVFabricator. After that, the configuration file name and location is stored in the registry. The following options are available:

- **Imperial Measurements - Imperial Pipe Sizes** - Restores the *[Product Folder]\PVFabricator\System\Imperial.cfg* file.
- **Metric Measurements - Imperial Pipe Sizes** - Restores the *[Product Folder]\PVFabricator\System\Metric.cfg* file.
- **Metric Measurements - Metric Pipe Sizes** - Restores the *[Product Folder]\PVFabricator\System\Metric.cfg* file.

Startup Variables

Provides default drawing and component values. You can modify the variables in the configuration file as needed. For more information, see *Configuration File Dialog Box* (on page 29). The most recent configuration file is stored in the registry and is used the next time you start PVFabricator.

AlphaSizeControl

Default value: 0

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

Controls the number of decimal places used in **Alpha Size**. Setting this variable to zero places no decimals in **Alpha Size**. Setting the variable to a value from 1 to 9 places 1 to 9 decimal places in **Alpha Size**. Setting this variable to 10 suppresses trailing zeros in **Alpha Size**.

Example with a value 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 a value 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 a value 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 recommended setting is **10**.

BomScheduleDirection

Default value: 1

Allowed values: 0 or 1

Specifies the direction of the bill of materials schedule.

- **0** - Places the bill of materials schedule in the upward direction, with the column headings at the bottom.
- **1** - Places the bill of materials schedule in the downward direction, with the column headings at the top.

BomTagRadiusFactor

Default value: 1.25

Allowed values: Positive real

Controls the radius of tag bubbles. Bubbles are sized using this factor multiplied by the AutoCAD setvar **TEXTSIZE**.

BomTagSpacingFactor

Default value: 1.25

Allowed values: Positive real

Controls how far bill of materials bubbles are spaced from the model or components within the drawing.

EquipmentToolTipSetting

Default value: 1

Allowed values: Positive integer

Controls the items you see in the tool tip display when the cursor hovers over an equipment component.

- **0** - Turns off all tool tips.
- **1** - Displays the overall equipment name and component description in the tool tip. For nozzles, the diameter and flange rating are also displayed.
- **2** - Displays extended tool tips.

NOTE A setting of **2** has the following restrictions:

- If CADWorx Equipment is loaded with CADWorx Plant, the extended tool tips do not display.
- If the CADWorx Equipment component is in an XREF drawing, the extended tool tips do not display.

LanguageFile

Default value: English.dic

Allowed values: A valid and properly formatted language file

Specifies the language of all PVFabricator dialog boxes and prompts. The specified file must be present in the *[Product Folder]\PVFabricator\Support* folder for proper operation of PVFabricator.

MaterialFile

Default value: MaterialDivision1.TXT

Allowed values: Valid and properly formatted material file

Specifies the active material file. This allows you to use a different material file for each project.

NOTE This file must exist in the *[Product Folder]\PVFabricator\System* folder.

PlatformHandRailFile

Default value: Default_I.RAIL (Default_M.RAIL for metric)

Allowed values: Valid and properly formatted hand rail configuration file

Specifies the active hand rail configuration file. This option allows you to use a different hand rail configuration file for each project.

NOTE This file must exist in the *[Product Folder]\PVFabricator\System* folder.

PlatformLadderFile

Default value: Default_I.LADDER (Default_M.LADDER for metric)

Allowed values: Valid and properly formatted ladder configuration file

Specifies the active ladder configuration file. This option allows you to use a different ladder configuration file for each project.

NOTE This file must exist in the *[Product Folder]\PVFabricator\System* folder.

ReferenceLine

Default value: 0

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

Specifies the placement of certain components such as nozzles, saddles, platforms, and so on.

0 - Positions components with reference to the parent element's insertion location.

1 - Positions components with reference to the location point.

SaveLastProfileUsed

Default value: 1

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

Specifies whether PVFabricator saves the last profile used on exit.

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.

0 - Prevents PVFabricator from saving the last profile used.

1 - Allows PVFabricator to save the last profile used.

SteelLibraryDirectory

Default value: *[Product Folder]\PVFabricator\Steel_I* (*[Product Folder]\PVFabricator\Steel_M* for metric)

Allowed values: Valid folder name

Locates the data file used for drawing Steel. The data files can be located anywhere on the computer, local network, or wide area network.

NOTE This folder can be set as read-only.

SystemMeasure

Default value: 1

Allowed values: 0, 1, or 2

Specifies the unit system:

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

UpgradedLayersSection

Specifies whether the layer section of the configuration file has been updated with the new layers used with PVPFabricator new features.

This setting should not be changed. The software automatically manages this setting.

1 - The configuration file has been updated.

UserShapeDirectory

Default value: *[Product Folder]\PVPFabricator\UserShape*


Allowed values: Valid folder name






Specifies the directory where user shapes are stored.

If you change the directory, all files in the **UserShape** folder must be copied to this path to function correctly.

SECTION 4



Vessel General Panel

 Fabricator tab: **Vessel General**

	Command Name	 Command Line
	Generate 2D Outline - Generates the outline drawings of selected viewports in the layout or paper space. For more information, see <i>Generate 2D Outline</i> (on page 35).	G2D
	Update 2D Outline - Synchronizes the 2D outline with changes from the 3D model. For more information, see <i>Update 2D Outline</i> (on page 36).	U2D
	Setup - Specifies the defaults for generating drawings. For more information, see <i>Configuration Settings</i> (on page 36).	VDSET
	Block Manager - Adds user-defined blocks to a library used in the drawing. For more information, see <i>Block Manager</i> (on page 44).	BLKMAN

Generate 2D Outline

 Fabricator tab: **Vessel General > Generate 2D Outline** 

 Vessel General toolbar: **Generate 2D Outline** 


 Command line: **G2D**

Generates the outline drawings of selected viewports in the layout or paper space. Before using this command, you must position your viewports in your layout drawing, configure your settings, and add your blocks to the library.

CTRL+right-click a viewport to control its appearance. The available options include:

- **Export to BOM**
- **Export to NS**
- **Hide All**
- **Show All**
- **Hide All Hidden Lines**
- **Show All Hidden Lines**
- **Hide All Center Lines**
- **Show All Center Lines**

Generate a 2D outline


1. Click a paper space tab.
2. Click **Generate 2D Outline** .
3. Select a viewport or press ENTER for **Automatic**.
4. Select **Yes** to turn off viewports or **No** to leave them active.

The software adds the outline graphics to your existing viewports.

Update 2D Outline


 Fabricator tab: **Vessel General > Update 2D Outline** 

 Vessel General toolbar: **Update 2D Outline** 

 Command line: **U2D**

Synchronizes the 2D outline with changes from the 3D model.

Update a 2D outline

1. Click a paper space tab.
2. Click **Update 2D Outline** .

The software updates the outline to reflect any changes in the model.

Configuration Settings

 Fabricator tab: **Configuration Settings** 

 Vessel General panel: **Setup** 

 Vessel General toolbar: **Fabricator Setup** 

 Command line: **VDSET**

Displays the **Configuration Settings** dialog box so that you can specify the defaults for generating drawings.

Configuration Settings Dialog Box

Specifies parameters for drawing generation.

Configuration Type - Specifies the object type for which you are setting the configuration. This selection controls the options available on the rest of the dialog box. The available options are:

- **2D Outline** (on page 40)
- **Annotation Notes** (on page 40)
- **Annotation Callouts** (on page 40)
- **Annotation Weld Symbols** (on page 41)
- **Dimensions** (on page 41)

- **BOM Properties** (on page 42)
- **BOM Tag** (on page 42)
- **Nozzle Schedule Properties** (on page 43)
- **Nozzle Schedule Tag** (on page 44)

Save - Writes the current settings to a file so that you can use them on another project.


Load - Imports a settings file.

Apply - Applies the current settings to the active drawing.


What do you want to do?

- *Configure 2D outlines* (on page [37](#))
 - *Configure annotation notes* (on page 37)
 - *Configure annotation callouts* (on page 38)
 - *Configure annotation weld symbols* (on page 38)
 - *Configure dimensions* (on page 38)
 - *Configure bill of materials properties* (on page 38)
 - *Configure bill of materials tags* (on page 38)
 - *Configure nozzle schedule properties* (on page 39)
 - *Configure nozzle schedule tags* (on page 39)
 - *Save configuration settings to a file* (on page 39)
 - *Load configuration settings from a file* (on page 39)
-


Configure 2D outlines

1. Click **Setup** .
*The **Configuration Settings** dialog box displays.*
2. Click **2D Outline** in the tree.
The 2D outline configuration settings display.
3. Set up the configuration settings to meet your needs. For more information, see *2D Outline* (on page 40).


Configure annotation notes

1. Click **Setup** .
*The **Configuration Settings** dialog box displays.*
2. Click **Annotation > Notes** in the tree.
The annotation note configuration settings display.
3. Set up the configuration settings to meet your needs. For more information, see *Annotation Notes* (on page 40).


Configure annotation callouts

1. Click **Setup** .
*The **Configuration Settings** dialog box displays.*
2. Click **Annotation > Callouts** in the tree.
The annotation callout configuration settings display.
3. Set up the configuration settings to meet your needs. For more information, see *Annotation Callouts* (on page 40).


Configure annotation weld symbols

1. Click **Setup** .
*The **Configuration Settings** dialog box displays.*
2. Click **Annotation > Weld Symbols** in the tree.
The annotation weld symbols configuration settings display.
3. Set up the configuration settings to meet your needs. For more information, see *Annotation Weld Symbols* (on page 41).


Configure dimensions

1. Click **Setup** .
*The **Configuration Settings** dialog box displays.*
2. Click **Dimensions** in the tree.
The dimension configuration settings display.
3. Set up the configuration settings to meet your needs. For more information, see *Dimensions* (on page 41).


Configure bill of materials properties

1. Click **Setup** .
*The **Configuration Settings** dialog box displays.*
2. Click **BOM > Properties** in the tree.
The bill of materials property configuration settings display.
3. Set up the configuration settings to meet your needs. For more information, see *BOM Properties* (on page 42).


Configure bill of materials tags

1. Click **Setup** .
*The **Configuration Settings** dialog box displays.*
2. Click **BOM > Tag** in the tree.
The bill of materials tag configuration settings display.
3. Set up the configuration settings to meet your needs. For more information, see *BOM Tag* (on page 42).


Configure nozzle schedule properties

1. Click **Setup** .
*The **Configuration Settings** dialog box displays.*
2. Click **Nozzle schedule > Properties** in the tree.
The nozzle schedule property configuration settings display.
3. Set up the configuration settings to meet your needs. For more information, see *Nozzle Schedule Properties* (on page 43).


Configure nozzle schedule tags

1. Click **Setup** .
*The **Configuration Settings** dialog box displays.*
2. Click **Nozzle schedule > Tag** in the tree.
The nozzle schedule tag configuration settings display.
3. Set up the configuration settings to meet your needs. For more information, see *Nozzle Schedule Tag* (on page 44).

Save configuration settings to a file

1. Click **Setup** .
*The **Configuration Settings** dialog box displays.*
2. Click **Save**.
*The **Save to file** dialog box displays*
3. Specify the location and file name for the configuration settings.
4. Click **Save**.

Load configuration settings from a file

1. Click **Setup** .
*The **Configuration Settings** dialog box displays.*
2. Click **Load**.
*The **Load from file** dialog box displays.*
3. Specify the location and file name for the configuration settings.
4. Click **Open**.

2D Outline

Controls parameters for viewport outline drawings.

Main Layer - Specifies the 2D outline layer.

Main Color - Specifies the 2D outline color.

Main Linetype - Specifies the 2D outline linetype.

Hidden Layer - Specifies the hidden line layer.

Hidden Color - Specifies the hidden line color.

Hidden Linetype - Specifies the hidden line linetype.

Center Layer - Specifies the centerline layer.

Center Color - Specifies the center line color.

Center Linetype - Specifies the center line linetype.

Annotation Notes

Controls settings for notes.

Note Line Setting - Specifies the note line settings with the following:

- **Layer** - Specifies the note layer.
- **Color** - Specifies the note color.
- **Text Style** - Specifies the note text style.
- **Text height** - Specifies the note text height.

Title Setting - Specifies the title settings for notes with the following:

- **Text** - Specifies the note title.
- **Layer** - Specifies the note title layer.
- **Text style** - Specifies the note title text style.
- **Color** - Specifies the note title color.
- **Text height** - Specifies the note title text height.
- **Justification** - Specifies the note title justification.
- **Gap Distance** - Specifies the note title gap distance.

Paragraph Setting - Specifies the paragraph settings for notes with the following:

- **Tabs** - Specifies the tabs distance for notes.
- **Line spacing** - Specifies the line spacing for notes.

Annotation Callouts

Controls settings for annotation callouts.

Layer - Specifies the annotation callout layer.

Text Style - Specifies the annotation callout text style.

Color - Specifies the annotation callout color.

Height - Specifies the annotation callout height.

Leader - Indicates whether the annotation callout includes a leader line.

Arrow - Specifies the leader line arrow.

Border - Specifies the annotation callout border.

Annotation Weld Symbols

Controls settings for annotation weld symbols.

Layer - Specifies the annotation weld symbol layer.

Arrow - Specifies the annotation weld symbol arrow.

Dimensions

Controls settings for automatic dimensioning.

Dimension Style - Specifies the automatic dimensioning style.

Text Height - Specifies the automatic dimensioning text height.

Dimension Type - Specifies the automatic dimensioning type.

BOM

Controls settings for the bill of materials columns. These columns are based on the imported PV Elite data fields.

Data Column - Lists the available data items. Drag items from the **Data Column** list to the **Selected Column** list to add them to the bill of materials.

Selected Column - Lists the selected data items. You can add any number of columns. Drag items in the list to reorder them. Drop items on top of other items to concatenate those database fields into a single column on the bill of materials. The data from these new fields must be entered manually. Use the **SEPARATOR** field to add a separator type to a concatenated column.

Table Format

Row Height - Specifies the row height on the bill of materials.

Grow Direction - Specifies the direction in which the bill of materials table grows.

Sort Order - Displays the **Sort Order** dialog box.

Sort Type List - Displays the **Sort Type List** dialog box.

Column Format

Header Title - Specifies the column heading for the item selected in the **Selected Column** list.

Column Width - Specifies the column width for the item selected in the **Selected Column** list.

Alignment - Specifies the column alignment for the item selected in the **Selected Column** list.

Sort Order Dialog Box

Controls sorting parameters.

Level 1 - Specifies the data type and sorting order for the first level of sorting.

Level 2 - Specifies the data type and sorting order for the second level of sorting.

Level 3 - Specifies the data type and sorting order for the third level of sorting.

Sort Type List Dialog Box

Displays the list of data types and indicates the order in which they are sorted. Drag and drop items in the list to change the order.

BOM Properties

Controls settings for bill of materials properties.

Layer - Specifies the bill of materials properties layer.

Line Color - Specifies the bill of materials properties line color.

Linetype - Specifies the bill of materials properties linetype.

Header Text Style - Specifies the bill of materials properties header text style.

Header Text Color - Specifies the bill of materials properties header text color.

Header Text Height - Specifies the bill of materials properties header text height.

Text Style - Specifies the bill of materials properties text style.

Text Color - Specifies the bill of materials properties text color.

Text Height - Specifies the bill of materials properties text height.

Warning Color - Specifies the bill of materials properties warning color.

Title - Specifies the bill of materials properties title text.

BOM Tag

Controls settings for bill of materials tags. This dialog box also controls the numbering style of the bill of materials and associated tags.

Layer - Specifies the bill of materials tag layer.

Balloon Color - Specifies the bill of materials tag balloon color.

Balloon Linetype - Specifies the bill of materials tag balloon linetype.

Balloon Size - Specifies the bill of materials tag balloon size.

Text Style - Specifies the bill of materials tag text style.

Text Color - Specifies the bill of materials tag text color.

Text Height - Specifies the bill of materials tag text height.

Warning Color - Specifies the bill of materials tag warning color.

Warning Text - Specifies the bill of materials tag warning text.

Tag Start - Specifies the bill of materials tag starting letter.

Tag Inc - Specifies the bill of materials tag increment.

Nozzle Schedule

Controls settings for the bill of materials columns. These columns are based on the imported PV Elite data fields associated with the nozzles.

Data Column - Lists the available data items. Drag items from the **Data Column** list to the **Selected Column** list to add them to the nozzle schedule.

Selected Column - Lists the selected data items. You can add any number of columns. Drag items in the list to reorder them. Drop items on top of other items to concatenate those database fields into a single column on the nozzle schedule. The data from these new fields must be entered manually. Use the **SEPARATOR** field to add a separator type to a concatenated column.

Table Format

Row Height - Specifies the row height on the nozzle schedule.

Grow Direction - Specifies the direction in which the nozzle schedule table grows.

Sort Order - Displays the **Sort Order** dialog box.

Sort Type List - displays the **Sort Type List** dialog box.

Column Format

Header Title - Specifies the column heading for the item selected in the **Selected Column** list.

Column Width - Specifies the column width for the item selected in the **Selected Column** list.

Alignment - Specifies the column alignment for the item selected in the **Selected Column** list.

Sort Order Dialog Box

Controls sorting parameters.

Level 1 - Specifies the data type and sorting order for the first level of sorting.

Level 2 - Specifies the data type and sorting order for the second level of sorting.

Level 3 - Specifies the data type and sorting order for the third level of sorting.

Sort Type List Dialog Box

Displays the list of data types and indicates the order in which they are sorted. Drag and drop items in the list to change the order.

Nozzle Schedule Properties

Controls settings for nozzle schedule properties.

Layer - Specifies the nozzle schedule properties layer.

Line Color - Specifies the nozzle schedule properties line color.

Linetype - Specifies the nozzle schedule properties linetype.

Header Text Style - Specifies the nozzle schedule properties header text style.

Header Text Color - Specifies the nozzle schedule properties header text color.

Header Text Height - Specifies the nozzle schedule properties header text height.

Text Style - Specifies the nozzle schedule properties text style.

Text Color - Specifies the nozzle schedule properties text color.

Text Height - Specifies the nozzle schedule properties text height.

Warning Color - Specifies the nozzle schedule properties warning color.

Title - Specifies the nozzle schedule properties title text.

Nozzle Schedule Tag

Controls settings for nozzle schedule tags. This dialog box also controls the numbering style of the nozzle schedule and associated tags.

Layer - Specifies the nozzle schedule tag layer.

Balloon Color - Specifies the nozzle schedule tag balloon color.

Balloon Linetype - Specifies the nozzle schedule tag balloon linetype.

Balloon Size - Specifies the nozzle schedule tag balloon size.

Text Style - Specifies the nozzle schedule tag text style.

Text Color - Specifies the nozzle schedule tag text color.

Text Height - Specifies the nozzle schedule tag text height.

Warning Color - Specifies the nozzle schedule tag warning color.

Warning Text - Specifies the nozzle schedule tag warning text.

Tag Start - Indicates whether the tag numbering is automatic. If this value is set to **Automatic**, the description from PV Elite is the number used for the tags and nozzle schedule items.


Tag Inc - Specifies the nozzle schedule tag increment.

Combine Nozzles - Indicates whether identical nozzles are combined into one line item. This option is only available if **Tag Start** is set to **Manual**.

Block Manager

 Fabricator tab: **Vessel General > Block Manager** 

 Vessel General toolbar: **Block Manager** 

 Command line: **BLKMAN**

Adds user-defined blocks to a library used in the drawing. This library is made up of different types of blocks, and each type is used in a different way. The manager also controls the notes and callouts.

Save - Writes the current library to a block manager file (*.BLKCFG).

Load - Reads a block manager file (*.BLKCFG).


- *Title Block Tab* (on page 46)
- *Name Plate Tab* (on page 47)
- *Weld Detail Tab* (on page 47)
- *Other Detail Tab* (on page 47)

- *Notes Tab* (on page 48)
 - *Callouts Tab* (on page 48)
 - *Weld Symbols Tab* (on page 49)
 - *Data Mapping Tab* (on page 49)
-


What do you want to do?

- *Add title blocks to the library* (on page 45)
 - *Add name plates to the library* (on page 45)
 - *Add weld details to the library* (on page 45)
 - *Add other details to the library* (on page 46)
 - *Add notes to the library* (on page 46)
 - *Add weld symbols to the library* (on page 46)
 - *Set up data mapping* (on page 46)
-


Add title blocks to the library

1. Click **Block Manager** .
*The **Block Manager** dialog box displays.*
2. Click the **Title Block** tab.
3. Set up the parameters to meet your requirements. For more information, see *Title Block Tab* (on page 46).

Add name plates to the library

1. Click **Block Manager** .
*The **Block Manager** dialog box displays.*
2. Click the **Name Plate** tab.
3. Set up the parameters to meet your requirements. For more information, see *Name Plate Tab* (on page 47).

Add weld details to the library

1. Click **Block Manager** .
*The **Block Manager** dialog box displays.*
2. Click the **Weld Detail** tab.
3. Set up the parameters to meet your requirements. For more information, see *Weld Detail Tab* (on page 47).

Add other details to the library

1. Click **Block Manager** .

*The **Block Manager** dialog box displays.*

2. Click the **Other Detail** tab.
3. Set up the parameters to meet your requirements. For more information, see *Other Detail Tab* (on page 47).

Add notes to the library

1. Click **Block Manager** .

*The **Block Manager** dialog box displays.*

2. Click the **Notes** tab.
3. Set up the parameters to meet your requirements. For more information, see *Notes Tab* (on page 48).

Set up data mapping

1. Click **Block Manager** .

*The **Block Manager** dialog box displays.*

2. Click the **Data Mapping** tab.
3. Set up the parameters to meet your requirements. For more information, see *Data Mapping Tab* (on page 49).

Add weld symbols to the library

1. Click **Block Manager** .

*The **Block Manager** dialog box displays.*

2. Click the **Weld Symbols** tab.
3. Set up the parameters to meet your requirements. For more information, see *Weld Symbols Tab* (on page 49).

Title Block Tab

Adds user-defined title blocks to the library. You can add any number of title blocks. This includes any embedded blocks defined in the drawings.

Library Path - Specifies the file location. Type the path to the file, or click **Browse** to locate the file.

Library - Displays the items currently in the library.

Add - Displays the **Select Drawing File** dialog box so that you can select drawings to add to the library. After you select a file, the **Select <item>** dialog box displays. For more information, see *Select <item> Dialog Box* (on page 50).

Remove - Deletes the selected items from the **Library** list.

Preview - Displays a preview of the selected item in the **Library** list.

Name Plate Tab

Adds name plate drawings to the library. You can add any number of name plates. This includes any embedded blocks defined in the drawings.

Library Path - Specifies the file location. Type the path to the file, or click **Browse** to locate the file.

Library - Displays the items currently in the library.

Add - Displays the **Select Drawing File** dialog box so that you can select drawings to add to the library. After you select a file, the **Select <item>** dialog box displays. For more information, see *Select <item> Dialog Box* (on page 50).

Remove - Deletes the selected items from the **Library** list.

Preview - Displays a preview of the selected item in the **Library** list.

Weld Detail Tab

Adds weld detail blocks to the library. You can add any number of weld details. This includes any embedded blocks defined in the drawings.

Library Path - Specifies the file location. Type the path to the file, or click **Browse** to locate the file.

Library - Displays the items currently in the library.

Add - Displays the **Select Drawing File** dialog box so that you can select drawings to add to the library. After you select a file, the **Select <item>** dialog box displays. For more information, see *Select <item> Dialog Box* (on page 50).

Remove - Deletes the selected items from the **Library** list.

Preview - Displays a preview of the selected item in the **Library** list.

Other Detail Tab

Adds other detail blocks to the library. You can add any number of details. This includes any embedded blocks defined in the drawings.

Library Path - Specifies the file location. Type the path to the file, or click **Browse** to locate the file.

Library - Displays the items currently in the library.

Add - Displays the **Select Drawing File** dialog box so that you can select drawings to add to the library. After you select a file, the **Select <item>** dialog box displays. For more information, see *Select <item> Dialog Box* (on page 50).

Remove - Deletes the selected items from the **Library** list.

Preview - Displays a preview of the selected item in the **Library** list.

Notes Tab

Defines notes to use in the drawing. You can define any number of notes.

Library - Displays the items currently in the library.

Add - Displays the **Add Note Line** dialog box so that you can define the individual notes added to the library.

Remove - Deletes the selected items from the **Library** list.

Preview - Displays a preview of the selected item in the **Library** list.

Add Note Line Dialog Box

Defines the individual notes added to the library. These settings control the appearance of the notes. They are saved with each note that you define.

Use global settings - Indicates whether global settings are applied to the note. You cannot specify the color or text height of notes if this option is selected.

Color - Specifies the color of the note. This option is only available if **Use global settings** is cleared.

Text height - Specifies the text height of the note. This option is only available if **Use global settings** is cleared.

Content - Type the text of the note.

Callouts Tab

Defines callouts to be used in the drawing. You can define any number of callouts.

Library - Displays the items currently in the library.

Add - Displays the **Add Callout** dialog box so that you can define the individual callouts added to the library.

Remove - Deletes the selected items from the **Library** list.

Preview - Displays a preview of the selected item in the **Library** list.

Add Callout Dialog Box

Defines the individual callouts added to the library. These settings control the appearance of the callouts. They are saved with each callout that you define.

Use global settings - Indicates whether global settings are applied to the callout. You cannot specify the **Member setting** options of callouts if this option is selected.

Member setting

Layer - Specifies the layer of the callout. This option is only available if **Use global settings** is cleared.

Color - Specifies the color of the callout. This option is only available if **Use global settings** is cleared.

Text Style - Specifies the text style of the callout. This option is only available if **Use global settings** is cleared.

Height - Specifies the height of the callout. This option is only available if **Use global settings** is cleared.

Leader - Indicates whether the callout includes a leader. This option is only available if **Use global settings** is cleared.

Arrow - Specifies the arrow associated with the callout. This option is only available if **Leader** is set to **Yes**.

Border - Specifies the border associated with the callout. This option is only available if **Use global settings** is cleared.

Content - Type the text of the callout.

Weld Symbols Tab

Adds weld symbol blocks to the library. You can add any number of weld symbol blocks. This includes any embedded blocks defined in the drawings.

Library Path - Specifies the file location. Type the path to the file, or click **Browse** to locate the file.

Library - Displays the items currently in the library.

Add - Displays the **Select Drawing File** dialog box so that you can select drawings to add to the library. After you select a file, the **Select <item>** dialog box displays. For more information, see *Select <item> Dialog Box* (on page 50).

Remove - Deletes the selected items from the **Library** list.

Preview - Displays a preview of the selected item in the **Library** list.

Data Mapping Tab

Maps PV Elite data to attributes in the library blocks. This mapping depends on the block type. After the attributes are mapped, the PV Elite data automatically fills in the defined attribute for a given block type. **Title Block**, **Name Plate**, and **Other Detail** all map the data to their respective attributes.

Title Block - Loads **Block List** with title block objects.

Other Detail - Loads **Block List** with other detail objects.

Name Plate - Loads **Block List** with name plate objects.

Block List - Indicates the active library block.

PV Elite Data - Displays the available PV Elite data. Drag items from this list to the **Block Attributes** list to create the mapping.

Block Attributes - Displays the attributes assigned to the blocks. Drag items from the **PV Elite Data** list and drop them on items in this list to map the attributes.

Prev - Displays the **Block Attributes** for the previous item in the **Block List**.

Next - Displays the **Block Attributes** for the next item in the **Block List**.

Clear - Removes the mapping for the current set of **Block Attributes**.

Select <item> Dialog Box

Specifies parameters for adding items to the **Library** list on the **Block Manager** dialog box.

Browse - Displays the **Select Drawing File** dialog box so that you can select drawings to add to the list.

File list - Lists the blocks in the drawing file. Select the check boxes next to the file name to add the blocks.

Preview - Displays a preview of the highlighted block.

Check All - Selects the check boxes for all of the blocks in the list.

Uncheck All - Clears the check boxes for all of the blocks in the list.






Filter - Enter a filter.

Add - Adds the selected blocks.

SECTION 5

BOM and Nozzle Schedule Panel


 Fabricator tab: **BOM and Nozzle Schedule**

	Command Name	 Command Line
	Generate BOM - Adds a bill of materials to the drawing. For more information, see <i>Generate BOM</i> (on page 51).	GBOM
	Generate Nozzle Schedule - Adds a nozzle schedule to the drawing. For more information, see <i>Generate Nozzle Schedule</i> (on page 52).	GNS
	Insert BOM Item - Adds an item to the bill of materials. For more information, see <i>Insert BOM Item</i> (on page 53).	IBI
	Insert Nozzle Schedule Item - Adds an item to the nozzle schedule. For more information, see <i>Insert Nozzle Schedule Item</i> (on page 53).	INI

Generate BOM

 Fabricator tab: **BOM and Nozzle Schedule > Generate BOM** 

 BOM and Nozzle Schedule toolbar: **Generate BOM** 

 Command line: **GBOM**

Adds a bill of materials to the drawing. The columns that display, and the format of the table, are controlled by the BOM configuration settings. For more information, see *BOM* (on page 41).

All - Generates a bill of material (BOM) for all the objects in the paper space.

Selection - Generates a bill of material for the selected object in the paper space.

append to Existing BOM - Adds a bill of material to an existing BOM in the paper space.


Current Layout - Generates a detail BOM for the currently selected paper space.

Drawing - Generates a BOM for the entire drawing. This includes all paper spaces.

NOTE For the **Drawing** option to appear differently from that of the **Current Layout** option you must have more than one paper space.

Add a bill of materials to the drawing

Generate All BOMs

1. Click a paper space tab.
2. Click **Generate BOM** .
3. Select **All**, or type **ALL** on the command line, and then press ENTER.
4. Select **Current Layout**, or type **C** on the command line, and then press ENTER.

The software generates the BOM for the currently selected paper space.

OR


Select **Drawing**, or type **D** on the command line, and then press ENTER.

The software generates the BOM for all available paper spaces.

5. Click in paper space to place the bill of materials.


The bill of materials is placed.

Generate Selection BOMs

1. Click a paper space tab.
2. Click **Generate BOM** .
3. Select **Selection**, or type **S** on the command line, and then press ENTER.
4. Select the model to be included in the BOM, and then press ENTER to finish.
5. Click in paper space to place the bill of materials.

The bill of materials is placed.

Append Existing BOMs

1. Click a paper space tab.
2. Click **Generate BOM** .
3. Select **append to Existing BOM**, or type **E** on the command line, and then press ENTER.
4. Select the BOM to append to.
5. Select the model data to add to the BOM, and then press ENTER to finish.

The bill of materials is updated with the new information.

Generate Nozzle Schedule


 Fabricator tab: **BOM and Nozzle Schedule > Generate Nozzle Schedule** 

 BOM and Nozzle Schedule toolbar: **Generate Nozzle Schedule** 






 Command line: **GNS**

Adds a nozzle schedule to the drawing. The columns that display, and the format of the table, are controlled by the nozzle schedule configuration settings. For more information, see *Nozzle Schedule* (on page 43).

Add a nozzle schedule to the drawing

1. Click a paper space tab.
 2. Click **Generate Nozzle Schedule** .
 3. Select the origin point for the nozzle schedule.
- Pick an origin point or [topLeft/Topright/Bottomleft/bottomRight].*
- The nozzle schedule displays in the drawing.*

Insert BOM Item

-  Fabricator tab: **BOM and Nozzle Schedule > Insert BOM Item** 
-  BOM and Nozzle Schedule toolbar: **Insert BOM Item** 
-  Command line: **IBI**

Adds an item to the bill of materials. The format of bill of materials tags is controlled by the BOM configuration settings. For more information, see *BOM Tag* (on page 42).

Add BOM Item Dialog Box


Controls parameters for adding an item to the bill of materials.

ITEM - Specifies the tag for the item. The next tag in the sequence displays by default. The tag start point and increment are controlled by the bill of materials tag configuration settings. For more information, see *BOM Tag* (on page 42).

TYPE - Specifies the type of the item.






QTY - Specifies the quantity of the item.

Insert an item into a bill of materials

1. Click a paper space tab.
2. Click **Insert BOM Item** .
- The **Add BOM Item** dialog box displays.*
3. Specify the field values to meet your needs.
4. Click **OK**.

The software adds the item to the bill of materials.

Insert Nozzle Schedule Item

-  Fabricator tab: **BOM and Nozzle Schedule > Insert Nozzle Schedule Item** 
-  BOM and Nozzle Schedule toolbar: **Insert Nozzle Schedule Item** 
-  Command line: **INI**

Adds an item to the nozzle schedule. The format of nozzle schedule tags is controlled by the nozzle schedule configuration settings. For more information, see *Nozzle Schedule Tag* (on page 44).


Add Nozzle Schedule Item Dialog Box

Controls parameters for adding an item to the nozzle schedule.

MARK - Specifies the mark for the nozzle schedule item.


REPAD_SIZE - Specifies the repad size for the nozzle schedule item.








Insert an item into a nozzle schedule

1. Click a paper space tab.
2. Click **Insert Nozzle Schedule Item** .
- The **Add Nozzle Schedule Item** dialog box displays.*
3. Specify the field values to meet your needs.
4. Click **OK**.
- The software adds the item to the nozzle schedule.*

SECTION 6



Tagging Panel

 Fabricator tab: **Tagging**

	Command Name	 Command Line
	Generate Tag - Generates tags for the items that you select. For more information, see <i>Generate Tag</i> (on page 55).	G TAG
	Generate BOM Tag - Generates bill of material tags for the items that you select. For more information, see <i>Generate BOM Tag</i> (on page 56).	GBT
	Generate Nozzle Schedule Tag - Generates nozzle schedule tags for the items that you select. For more information, see <i>Generate Nozzle Schedule Tag</i> (on page 56).	GNT
	Move Tag - Moves a tag. For more information, see <i>Move Tag</i> (on page 56).	TMV
	Remove Tag Line - Deletes the leader line from a tag. For more information, see <i>Remove Tag Line</i> (on page 57).	TRTL
	Reset Tag - Resets a tag. For more information, see <i>Reset Tag</i> (on page 57).	TRS

Generate Tag


 Fabricator tab: **Tagging > Generate Tag** 

 Tagging toolbar: **Generate Tag** 

 Command line: **G TAG**

Generates tags for the items that you select.

Generate a tag

1. Click a paper space tab.
2. Click **Generate Tag** .






Select a component or a viewport.
3. Select the items to tag.

Pick a point to insert tag for <item>.
4. Select the end point for the leader line.

Pick another point.
5. Select the point for the tag.


The tag displays in the drawing.

Generate BOM Tag






-  Fabricator tab: **Tagging > Generate BOM Tag** 
-  Tagging toolbar: **Generate BOM Tag** 
-  Command line: **GBT**

Generates bill of materials tags for the items that you select.

Generate bill of materials tags


1. Click a paper space tab.
2. Click **Generate BOM Tag** .
Select a viewport or components.
3. Select the items to tag.
Pick a point to insert tag for <item>.
4. Select the end point for the leader line.
Pick another point.
5. Select the point for the tag.
The tag displays in the drawing.

Generate Nozzle Schedule Tag






-  Fabricator tab: **Tagging > Generate Nozzle Schedule Tag** 
-  Tagging toolbar: **Generate Nozzle Schedule Tag** 
-  Command line: **GNT**

Generates nozzle schedule tags for the items that you select.

Generate nozzle schedule tags


1. Click a paper space tab.
2. Click **Generate Nozzle Schedule Tag** .
Select a viewport or components.
3. Select the items to tag.
Pick a point to insert tag for <item>.
4. Select the end point for the leader line.
Pick another point.
5. Select the point for the tag.
The tag displays in the drawing.

Move Tag


-  Fabricator tab: **Tagging > Move Tag** 
-  Tagging toolbar: **Move Tag** 
-  Command line: **TMV**

Moves a tag.

Move a tag

1. Click a paper space tab.
2. Click **Move Tag** .
Select tag to move, 1 group.
3. Select the tag to move.
Pick another point.
4. Select the point for the tag.
The tag displays in the new location.

Remove Tag Line


 Fabricator tab: **Tagging > Remove Tag Line** 

 Tagging toolbar: **Remove Tag Line** 

 Command line: **TRTL**

Deletes the leader line from a tag.

Remove a tag leader line

1. Click a paper space tab.
2. Click **Remove Tag Line** .
Select tag to remove tag line, 1 group.
3. Select the tag to remove the leader line.

Reset Tag


 Fabricator tab: **Tagging > Reset Tag** 

 Tagging toolbar: **Reset Tag** 

 Command line: **TRS**


Resets a tag.










Reset a tag

1. Click a paper space tab.
2. Click **Reset Tag** .
Select tag to reset, 1 group.
3. Select the tag to reset.
Pick a point to insert.
4. Select the end point for the leader line.
Pick another point.
5. Select the point for the tag.
The tag displays in the new location.

SECTION 7

Details Panel

 Fabricator tab: **Details**

	Command Name	 Command Line
	Insert Title Block - Inserts a title block in the current layout. For more information, see <i>Insert Title Block</i> (on page 59).	ITB
	Edit Title Block - Edits the title block in the current layout. For more information, see <i>Edit Title Block</i> (on page 60).	ETB
	Insert Weld Detail - Inserts weld details. For more information, see <i>Insert Weld Detail</i> (on page 61).	IWD
	Edit Weld Detail - Edits weld details. For more information, see <i>Edit Weld Detail</i> (on page 62).	EWD
	Insert Other Detail - Inserts other details. For more information, see <i>Insert Other Detail</i> (on page 62).	IOD
	Edit Other Detail - Edits other details. For more information, see <i>Edit Other Detail</i> (on page 63).	EOD
	Insert Name Plate - Inserts a name plate in the current layout. For more information, see <i>Insert Name Plate</i> (on page 64).	INP
	Edit Name Plate - Edits a name plate in the current layout. For more information, see <i>Edit Name Plate</i> (on page 65).	ENP

Insert Title Block

 Fabricator tab: **Details > Insert Title Block** 

 Details toolbar: **Insert Title Block** 

 Command line: **ITB**

Displays the **Insert Title Block** dialog box so that you can select a title block to add to the layout. The active title block library is controlled by the **Title Block** tab on the **Block Manager** dialog box. For more information, see *Title Block Tab* (on page 46).

Insert Title Block Dialog Box

Controls parameters for inserting title blocks into the layout.


Library - Displays the items currently in the library. The items on this list are controlled by the **Title Block** tab on the **Block Manager** dialog box. For more information, see *Title Block Tab* (on page 46).

Preview - Displays a preview of the selected item in the **Library** list.

Scale - Specifies the scale of the item.

Insert - Inserts the selected symbol into the layout.

Insert a title block

1. Click a paper space tab.
2. Click **Insert Title Block** .
*The **Insert Title Block** dialog box displays.*
3. Select the title block to place from the **Library** list.
4. Click **Insert**.
The title block displays in the layout.

Edit Title Block

 Fabricator tab: **Details > Edit Title Block** 

 Details toolbar: **Edit Title Block** 

 Command line: **ETB**

Displays the **Edit Title Block** dialog box so that you can edit a title block.

The active title block library is controlled by the **Title Block** tab on the **Block Manager** dialog box. For more information, see *Title Block Tab* (on page 46).

Edit Title Block Dialog Box


Controls parameters for editing a title block.

Library - Displays the items currently in the library. The items in this list are controlled by the **Title Block** tab on the **Block Manager** dialog box. For more information, see *Title Block Tab* (on page 46).

Preview - Displays a preview of the selected item in the **Library** list.

Scale - Specifies the scale of the item.


Edit a title block

1. Click a paper space tab.
2. Click **Edit Title Block** .
Select title block.
3. Select the title block to edit.
*The **Edit Title Block** dialog box displays.*
4. Make the changes as necessary.

Insert Weld Detail

 Fabricator tab: **Details > Insert Weld Detail** 

 Details toolbar: **Insert Weld Detail** 

 Command line: **IWD**

Displays the **Insert Weld Details** dialog box so that you can select weld details to add to the layout.

The active weld detail library is controlled by the **Weld Detail** tab on the **Block Manager** dialog box. For more information, see *Weld Detail Tab* (on page 47).

Insert Weld Details Dialog Box

Controls properties for inserting weld details into the layout.

Library - Displays the items currently in the library. The items on this list are controlled by the **Weld Detail** tab on the **Block Manager** dialog box. For more information, see *Weld Detail Tab* (on page 47).

Selected - Displays the items that you have chosen from the library. Drag items from the **Library** list to add them to this list.


Preview - Displays a preview of the selected item in the **Library** list.

Scale - Specifies the scale of the item.

Spacing - Specifies the spacing between the items.



Insert - Inserts the selected symbol into the layout.


Insert a weld detail

1. Click a paper space tab.
2. Click **Insert Weld Detail** .
*The **Insert Weld Details** dialog box displays.*
3. Double click the weld detail to place from the **Library** list.
*The weld detail appears under **Selected**.*
4. Select the weld detail to place from the **Selected** list.
5. Click **Insert**.
6. Type **A** to insert weld details in an area, or type **I** to insert individual weld details.
7. Select a first corner point.
8. Select a second corner point.
9. Select the direction for the details.
The weld details display in the layout.

Edit Weld Detail

 Fabricator tab: **Details > Edit Weld Detail** 

 Details toolbar: **Edit Weld Detail** 

 Command line: **EWD**

Displays the **Edit Weld Details** dialog box so that you can edit the weld details.

The active weld detail library is controlled by the **Weld Detail** tab on the **Block Manager** dialog box. For more information, see *Weld Detail Tab* (on page 47).

Edit Weld Details Dialog Box

Controls parameters for editing weld details.

Library - Displays the items currently in the library. The items in this list are controlled by the **Weld Details** tab on the **Block Manager** dialog box. For more information, see *Weld Detail Tab* (on page 47).


Selected - Displays the items that you have chosen from the library. Drag items from the **Library** list to add them to this list.

Preview - Displays a preview of the selected item in the **Library** list.

Scale - Specifies the scale of the item.

Spacing - Specifies the spacing between the items.

Edit a weld detail

1. Click a paper space tab.
2. Click **Edit Weld Detail** 
Select a weld detail.
3. Select the weld detail to edit.
*The **Edit Weld Detail** dialog box displays.*
4. Make the changes as necessary.

Insert Other Detail

 Fabricator tab: **Details > Insert Other Detail** 

 Details toolbar: **Insert Other Detail** 

 Command line: **IOD**

Displays the **Insert Other Details** dialog box so that you can insert other details into the layout.

The active other detail library is controlled by the **Other Detail** tab on the **Block Manager** dialog box. For more information, see *Other Detail Tab* (on page 47).

Insert Other Details Dialog Box

Controls parameters for inserting other details in the layout.

Library - Displays the items currently in the library. The items in this list are controlled by the **Other Detail** tab on the **Block Manager** dialog box. For more information, see *Other Detail Tab* (on page 47).

Selected - Displays the items that you have chosen from the library. Drag items from the **Library** list to add them to this list.


Preview - Displays a preview of the selected item in the **Library** list.

Scale - Specifies the scale of the item.

Spacing - Specifies the spacing between the items.

Insert - Inserts the selected symbol into the layout.

Insert other details

1. Click a paper space tab.
2. Click **Insert Other Detail** .
- The **Insert Other Details** dialog box displays.*
3. Select the details to place from the **Library** list, and drag them to the **Selected** list.
4. Click **Insert**.
5. Type **A** to insert the details in an area, or type **I** to insert individual details.
6. Specify first corner point.
7. Specify second corner point.
8. Specify the directions for the details.

The details display in the layout.

Edit Other Detail

 Fabricator tab: **Details > Edit Other Detail** 

 Details toolbar: **Edit Other Detail** 

 Command line: **EOD**

Displays the **Edit Other Details** dialog box so that you can edit other details in the layout.

The active other detail library is controlled by the **Other Detail** tab on the **Block Manager** dialog box. For more information, see *Other Detail Tab* (on page 47).

Edit Other Details Dialog Box

Controls parameters for inserting other details in the layout.

Library - Displays the items currently in the library. The items in this list are controlled by the **Other Detail** tab on the **Block Manager** dialog box. For more information, see *Other Detail Tab* (on page 47).


Selected - Displays the items that you have chosen from the library. Drag items from the **Library** list to add them to this list.

Preview - Displays a preview of the selected item in the **Library** list.

Scale - Specifies the scale of the item.

Spacing - Specifies the spacing between the items.

Edit other details

1. Click a paper space tab.
2. Click **Edit Other Detail** .
Select a detail.
3. Select the detail to edit.
*The **Edit Other Details** dialog box displays.*
4. Make the changes as necessary.

Insert Name Plate

 Fabricator tab: **Details > Insert Name Plate** 

 Details toolbar: **Insert Name Plate** 

 Command line: **INP**

Displays the **Insert Name Plate** dialog box so that you can insert a name plate in the current layout.

The active name plate library is controlled by the **Name Plate** tab on the **Block Manager** dialog box. For more information, see *Name Plate Tab* (on page 47).

Insert Name Plate Dialog Box

Controls parameters for inserting name plates into the layout.


Library - Displays the items currently in the library. The items on this list are controlled by the **Name Plate** tab on the **Block Manager** dialog box. For more information, see *Name Plate Tab* (on page 47).

Preview - Displays a preview of the selected item in the **Library** list.

Scale - Specifies the scale of the item.

Insert - Inserts the selected symbol into the layout.

Insert a name plate

1. Click a paper space tab.
2. Click **Insert Name Plate** .
*The **Insert Name Plate** dialog box displays.*
3. Select the name plate to place from the **Library** list.
4. Click **Insert**.
Pick a point.
5. Specify the location for the name plate.
The name plate displays in the layout.

Edit Name Plate

 Fabricator tab: **Details > Edit Name Plate** 

 Details toolbar: **Edit Name Plate** 

 Command line: **ENP**

Displays the **Edit Name Plate** dialog box so that you can edit a name plate in the current layout.

The active name plate library is controlled by the **Name Plate** tab on the **Block Manager** dialog box. For more information, see *Name Plate Tab* (on page 47).

Edit Name Plate Dialog Box


Controls parameters for editing name plates.

Library - Displays the items currently in the library. The items on this list are controlled by the **Name Plate** tab on the **Block Manager** dialog box. For more information, see *Name Plate Tab* (on page 47).

Preview - Displays a preview of the selected item in the **Library** list.


Scale - Specifies the scale of the item.








Edit a name plate

1. Click a paper space tab.
2. Click **Edit Name Plate** .
Select a name plate.
3. Select the name plate to edit.
*The **Edit Name Plate** dialog box displays.*
4. Make the changes as necessary.

SECTION 8

Annotation Panel

 Fabricator tab: **Annotation**

	Command Name	 Command Line
	Insert Weld Symbol - Inserts a weld symbol in the current layout. For more information, see <i>Insert Weld Symbol</i> (on page 67).	IWS
	Edit Weld Symbol - Edits an existing weld symbol. For more information, see <i>Edit Weld Symbol</i> (on page 68).	EWS
	Insert Note - Inserts a note in the current layout. For more information, see <i>Insert Note</i> (on page 69).	INT
	Edit Note - Edits an existing note. For more information, see <i>Edit Note</i> (on page 70).	ENT
	Insert Callout - Inserts a callout in the current layout. For more information, see <i>Insert Callout</i> (on page 71).	ICO
	Edit Callout - Edits an existing callout. For more information, see <i>Edit Callout</i> (on page 72).	ECO

Insert Weld Symbol

 Fabricator tab: **Annotation > Insert Weld Symbol** 

 Annotation toolbar: **Insert Weld Symbol** 

 Command line: **IWS**

Displays the **Insert Weld Symbol** dialog box so that you can select a weld symbol to insert in the current layout.

The default settings for weld symbols are controlled by the **Annotation > Weld Symbols** section of the **Configuration Settings** dialog box. For more information, see *Annotation Weld Symbols* (on page 41).

The active weld symbol library is controlled by the **Weld Symbols** tab on the **Block Manager** dialog box. For more information, see *Weld Symbols Tab* (on page 49).

Insert Weld Symbol Dialog Box

Controls parameters for inserting weld symbols into the layout.


Library - Displays the items currently in the library. The items in the list are controlled by the **Weld Symbols** tab on the **Block Manager** dialog box. For more information, see *Weld Symbols Tab* (on page 49).

Preview - Displays a preview of the selected item in the **Library** list.





Scale - Specifies the scale of the symbol.

Insert - Inserts the selected symbol into the layout.

Insert a weld symbol

1. Click a paper space tab.
2. Click **Insert Weld Symbol** .
*The **Insert Weld Symbol** dialog box displays.*
3. Select the weld symbol from the **Library** list.
*The symbol displays in the **Preview** area.*
4. Click **Insert**.
Pick first point.
5. Select the first point for the weld symbol on the layout.
Pick second point.
6. Select the second point for the weld symbol on the layout.
The weld symbol displays.

Edit Weld Symbol

-  Fabricator tab: **Annotation > Edit Weld Symbol** 
-  Annotation toolbar: **Edit Weld Symbol** 
-  Command line: **EWS**

Displays the **Edit Weld Symbol** dialog box so that you can change the parameters.

The default settings for weld symbols are controlled by the **Annotation > Weld Symbols** section of the **Configuration Settings** dialog box. For more information, see *Annotation Weld Symbols* (on page 41).

The active weld symbol library is controlled by the **Weld Symbols** tab on the **Block Manager** dialog box. For more information, see *Weld Symbols Tab* (on page 49).

Edit Weld Symbol Dialog Box

Controls parameters for editing weld symbols.


Use global settings - Indicates whether global settings are applied to the weld symbol. You cannot specify the **Member setting** options if this option is selected.

Member setting

Layer - Specifies the annotation weld symbol layer.

Arrow - Specifies the annotation weld symbol arrow.

Edit a weld symbol

1. Click a paper space tab.
2. Click **Edit Weld Symbol** .
Select a weld symbol.
3. Select the weld symbol to edit.
*The **Edit Weld Symbol** dialog box displays.*
4. Set up the parameters to meet your requirements.

Insert Note

 Fabricator tab: **Annotation > Insert Note** 

 Annotation toolbar: **Insert Note** 

 Command line: **INT**

Displays the **Insert Notes** dialog box so that you can select a note to insert in the current layout.

The default settings for notes are controlled by the **Annotation > Notes** section of the **Configuration Settings** dialog box. For more information, see *Annotation Notes* (on page 40).

The active note libraries are controlled by the **Notes** tab on the **Block Manager** dialog box. For more information, see *Notes Tab* (on page 48).

Insert Notes Dialog Box

Controls parameters for inserting notes in the layout.

Library - Displays the items currently in the library. The items on this list are controlled by the **Notes** tab on the **Block Manager** dialog box. For more information, see *Notes Tab* (on page 48).

Selected - Displays the items that you have chosen from the library. Drag items from the **Library** list to add them to this list.

Preview - Displays a preview of the selected item in the **Library** list.

Number style - Specifies the numbering style for the note.

Start number - Specifies the starting number for notes.

Orientation - Specifies the note orientation.

Details - Displays the **Note detail** dialog box.

Insert - Inserts the selected symbol into the layout.

Note Detail Dialog Box

Controls parameters for notes.

Use global settings - Indicates whether global settings are applied to the note. You cannot specify the detail options of the note if this option is selected.

Layer - Specifies the layer for the note.

Text Style - Specifies the text style for the note.

Title - Specifies the title for the note.

Title layer - Specifies the title layer for the note.

Title color - Specifies the title color for the note.

Gap distance - Specifies the distance between the title and the note.

Title justification - Specifies the title justification for the note.


Title text style - Specifies the title text style for the note.

Title text height - Specifies the title text height for the note.

Tabs - Specifies the tab settings for the note.

Line spacing - Specifies the line spacing for the note.

Insert a note

1. Click a paper space tab.
2. Click **Insert Note** .
*The **Insert Notes** dialog box displays.*
3. Double-click the note from the **Library** list.
*The note displays in the **Preview** area.*
4. Click **Insert**.
Specify first corner.
5. Select the first point for the note on the layout.
Specify opposite corner.
6. Select the second point for the note on the layout.
The note displays.

Edit Note

 Fabricator tab: **Annotation > Edit Note** 

 Annotation toolbar: **Edit Note** 

 Command line: **ENT**

Displays the **Edit Notes** dialog box so that you can edit a note.

The default settings for notes are controlled by the **Annotation > Notes** section of the **Configuration Settings** dialog box. For more information, see *Annotation Notes* (on page 40).

The active note libraries are controlled by the **Notes** tab on the **Block Manager** dialog box. For more information, see *Notes Tab* (on page 48).

Edit Notes Dialog Box

Controls parameters for editing notes.

Library - Displays the items currently in the library. The items on this list are controlled by the **Notes** tab on the **Block Manager** dialog box. For more information, see *Notes Tab* (on page 48).

Selected - Displays the items that you have chosen from the library. Drag items from the **Library** list to add them to this list.

Preview - Displays a preview of the selected item in the **Library** list.

Number style - Specifies the numbering style for the note.

Start number - Specifies the starting number for notes.

Orientation - Specifies the note orientation.

Details - Displays the **Note detail** dialog box.

Note Detail Dialog Box

Controls parameters for notes.

Use global settings - Indicates whether global settings are applied to the note. You cannot specify the detail options of the note if this option is selected.

Layer - Specifies the layer for the note.

Text Style - Specifies the text style for the note.

Title - Specifies the title for the note.

Title layer - Specifies the title layer for the note.

Title color - Specifies the title color for the note.

Gap distance - Specifies the distance between the title and the note.

Title justification - Specifies the title justification for the note.


Title text style - Specifies the title text style for the note.

Title text height - Specifies the title text height for the note.

Tabs - Specifies the tab settings for the note.

Line spacing - Specifies the line spacing for the note.

Edit a note

1. Click a paper space tab.
2. Click **Edit Note**  .
Select notes.
3. Select the note to edit.
*The **Edit Notes** dialog box displays.*
4. Set up the parameters to meet your requirements.

Insert Callout

 Fabricator tab: **Annotation > Insert Callout** 

 Annotation toolbar: **Insert Callout** 

 Command line: **ICO**

Displays the **Insert Callout** dialog box so that you can select a callout to insert in the current layout.

The default settings for callouts are controlled by the **Annotation > Callouts** section of the **Configuration Settings** dialog box. For more information, see *Annotation Callouts* (on page 40).

The active callout libraries are controlled by the **Callouts** tab on the **Block Manager** dialog box. For more information, see *Callouts Tab* (on page 48).

Insert Callout Dialog Box


Controls parameters for inserting callouts in the layout.

Library - Displays the items currently in the library. The items on this list are controlled by the **Callouts** tab on the **Block Manager** dialog box. For more information, see *Callouts Tab* (on page 48).

Preview - Displays a preview of the selected item in the **Library** list.

Insert - Inserts the selected symbol into the layout.

Insert a callout

1. Click a paper space tab.
2. Click **Insert Callout** .
*The **Insert Callouts** dialog box displays.*
3. Select the callout from the **Library** list.
*The callout displays in the **Preview** area.*
4. Click **Insert**.
Pick first point.
5. Select the point for the leader line.
Pick second point.
6. Select the point for the callout.
The callout displays.

Edit Callout

 Fabricator tab: **Annotation > Edit Callout** 

 Annotation toolbar: **Edit Callout** 

 Command line: **ECO**

Displays the **Edit Callout** dialog box so that you can edit a callout.

The default settings for callouts are controlled by the **Annotation > Callouts** section of the **Configuration Settings** dialog box. For more information, see *Annotation Callouts* (on page 40).

The active callout libraries are controlled by the **Callouts** tab on the **Block Manager** dialog box. For more information, see *Callouts Tab* (on page 48).

Edit Callout Dialog Box

Controls parameters for callouts.

Use global settings - Indicates whether global settings are applied to the callout. You cannot specify the detail options of the callout if this option is selected.

Layer - Specifies the layer for the callout.

Color - Specifies the color for the callout.

Text Style - Specifies the text style for the callout.

Height - Specifies the height of the callout.


Leader - Indicates whether the callout includes a leader line.

Arrow - Specifies the type of arrow for the leader line.

Border - Specifies the border for the callout.


Content - Specifies the callout text.



Edit a callout

1. Click a paper space tab.
2. Click **Edit Callout** .
Select a callout.
3. Select the callout to edit.
*The **Edit Callouts** dialog box displays.*
4. Set up the parameters to meet your requirements.

SECTION 9



Dimensioning Panel

 Fabricator tab: **Dimensioning**

	Command Name	 Command Line
	Generate Dimensions - Generates dimensions for outlines. For more information, see <i>Generate Dimensions</i> (on page 73).	GDIM

Generate Dimensions

 Fabricator tab: **Dimensioning > Generate Dimensions** 

 Dimensioning toolbar: **Generate Dimensions** 


 Command line: **GDIM**

Generates dimensions for 2D outlines.

What do you want to do?


- Generate a viewport
 - *Generate dimensions for a 2D drawing* (on page 73)
-

Generate dimensions for a 2D drawing

1. Click a paper space tab.
2. Click **Generate Dimensions** .
3. Select the viewport in which to generate the dimensions.
4. Select a point in the view, or select a component.
5. Select a point in the view, or select a component.
6. Specify the point for the dimensions.
7. Type **Y** to create the reference line, or type **N** to continue without the reference line.
8. Select the next point in the view.
9. Steps 4 through 9 repeat and you must press ENTER four times to finish. Alternatively, press ESC twice to finish.

SECTION 10

Fabricator Console













 PVFabricator menu: **Fabricator Palette**



















 Command line: **CADWORXEQUIPMENT** (also **EM**)





Displays the **PVFabricator Fabricator** console.

The icons at the top of the console represent commands that are also available on the panels and toolbars.

You can import a PV Elite file directly into the PV Fabricator environment to generate a complete fabrication drawing set. For more information, see *Import a model from PVElite* (on page 118).

	Setup - Specifies the defaults for generating drawings. For more information, see <i>Configuration Settings</i> (on page 36).
	Block Manager - Adds user-defined blocks to a library used in the drawing. For more information, see <i>Block Manager</i> (on page 44).
	Generate 2D Outline - Generates the outline drawings of selected viewports in the layout or paper space. For more information, see <i>Generate 2D Outline</i> (on page 35).
	Update the 2D Outline - Synchronizes the 2D outline with changes from the 3D model. For more information, see <i>Update 2D Outline</i> (on page 36).
	Generate BOM - Adds a bill of materials to the drawing. For more information, see <i>Generate BOM</i> (on page 51).
	Generate Nozzle Schedule - Adds a nozzle schedule to the drawing. For more information, see <i>Generate Nozzle Schedule</i> (on page 52).
	Insert BOM Item - Adds an item to the bill of materials. For more information, see <i>Insert BOM Item</i> (on page 53).
	Insert Nozzle Schedule Item - Adds an item to the nozzle schedule. For more information, see <i>Insert Nozzle Schedule Item</i> (on page 53).
	Generate Tag - Generates tags for the items that you select. For more information, see <i>Generate Tag</i> (on page 55).
	Generate BOM Tag - Generates bill of material tags for the items that you select. For more information, see <i>Generate BOM Tag</i> (on page 56).
	Generate Nozzle Schedule Tag - Generates nozzle schedule tags for the items that you select. For more information, see <i>Generate Nozzle Schedule Tag</i> (on page 56).
	Generate Automatic Tag - Automatically creates tags. For more information, see <i>Generate Automatic Tag</i> (on page 77).


	Move Tag - Moves a tag. For more information, see <i>Move Tag</i> (on page 56).
	Reset Tag - Resets a tag. For more information, see <i>Reset Tag</i> (on page 57).
	Remove Tag Line - Deletes the leader line from a tag. For more information, see <i>Remove Tag Line</i> (on page 57).
	Update Tag Description - Updates the description of the selected tag. For more information, see <i>Update Tag Description</i> (on page 78).
	Insert Title Block - Inserts a title block in the current layout. For more information, see <i>Insert Title Block</i> (on page 59).
	Edit Title Block - Edits the title block in the current layout. For more information, see <i>Edit Title Block</i> (on page 60).
	Insert Weld Details - Inserts weld details. For more information, see <i>Insert Weld Detail</i> (on page 61).
	Edit Weld Details - Edits weld details. For more information, see <i>Edit Weld Detail</i> (on page 62).
	Insert Other Details - Inserts other details. For more information, see <i>Insert Other Detail</i> (on page 62).
	Edit Other Details - Edits other details. For more information, see <i>Edit Other Detail</i> (on page 63).
	Insert Name Plate - Inserts a name plate in the current layout. For more information, see <i>Insert Name Plate</i> (on page 64).
	Edit Name Plate - Edits a name plate in the current layout. For more information, see <i>Edit Name Plate</i> (on page 65).
	Insert Notes - Inserts a note in the current layout. For more information, see <i>Insert Note</i> (on page 69).
	Edit Notes - Edits an existing note. For more information, see <i>Edit Note</i> (on page 70).
	Insert Weld Symbol - Inserts a weld symbol in the current layout. For more information, see <i>Insert Weld Symbol</i> (on page 67).
	Edit Weld Symbol - Edits an existing weld symbol. For more information, see <i>Edit Weld Symbol</i> (on page 68).
	Insert Callout - Inserts a callout in the current layout. For more information, see <i>Insert Callout</i> (on page 71).
	Edit Callout - Edits an existing callout. For more information, see <i>Edit Callout</i> (on page 72).

	Generate Dimensions - Generates dimensions for outlines. For more information, see <i>Generate Dimensions</i> (on page 73).
	Select Objects - Selects objects to edit or manipulate. For more information, see <i>Select Objects</i> .
	Match - Makes one object look like another selected objects. For more information, see <i>Match</i> (on page 77).
	Reset - Resets object properties to the default. For more information, see <i>Reset</i> (on page 77).

When you select items in the paper or model space, the parameters for those items display on the **Fabricator** console.

Generate Automatic Tag

 Fabricator console: **Generate Automatic Tag** 


 Command line: **GATAG**

Automatically creates tags.

BOM - Creates tags for the vessel based on items in the bill of materials.


Nozzle - Creates tags for the nozzles.

Combine - Creates tags for both the items in the bill of materials and the nozzles.

1. Click **Generate Automatic Tag** .
2. Select the type of tag.
3. Select a component or viewport.
Tags are automatically created.
4. Press ENTER to finish.

Match

 Fabricator console: **Match** 

 Command line: **MATSET**


Makes one object look like another selected object.

Reset

 Fabricator console: **Reset** 

 Command line: **RESSET**

Resets the object to the default.


1. Click **Reset** .
2. Select the component to reset.
The component resets.

Update Tag Description

 Fabricator console: **Update Tag Description** 

 Command line: **TUD**

Enables you to update tag descriptions.

1. Click **Update Tag Description** .
2. Select a component.
Tags descriptions are updated.
3. Press ENTER to finish.














SECTION 11










Details Tab

 PVFabricator menu: **Fabricator Palette > Details**

 Command line: **CADWORXEQUIPMENT** (also **EM**)

Displays the **PVFabricator Fabricator** console.

	Common Settings - Specifies the defaults for generating drawings. For more information, see <i>Common Settings</i> (on page 81).	DTSET
	Tubesheet Config - Displays the Tubesheet Configuration dialog box where you can create a tubesheet layout for detail. For more information, see <i>Tubesheet Configuration</i> (on page 83).	TSCFG
	Generate Detail - Generates a detailed drawing of a selected component. For more information, see <i>Generate Detail</i> (on page 95).	GND
	Update Detail - Updates the changes to the selected detail component. For more information, see <i>Update Detail</i> (on page 103).	UPD
	Generate Detail BOM - Adds a detailed bill of materials to the drawing. For more information, see <i>Generate Detail BOM</i> (on page 103).	GENDTBOM
	Update Detail BOM - Updates the detailed bill of materials in the drawing. For more information, see <i>Update Detail BOM</i> (on page 105).	DTBOMUPD
	Generate Detail Tag - Generates tags for the items that you select. For more information, see <i>Generate Detail Tag</i> (on page 105).	GDTAG
	Generate Automatic Detail Tag - Automatically creates tags. For more information, see <i>Generate Automatic Detail Tag</i> (on page 106).	GADTAG
	Add Linear Dimension - Places linear dimensions for a selected detail. For more information, see <i>Add Linear Dimensions</i> (on page 106).	LINEARDIM
	Add Aligned Dimension - Places aligned dimensions for a selected detail. For more information, see <i>Add Aligned Dimension</i> (on page 106).	ALIGNEDDIM
	Add Radius Dimension - Places the radius value for the selected detail. For more information, see <i>Add Radius Dimension</i> (on page 107).	RADIUSDIM
	Add Jogged Dimension - Places the jogged radius for the selected detail. For more information, see <i>Add Jogged Dimension</i> (on page 107).	JOGGEDDIM
	Add Angle Dimension - Places the angle dimension for the selected detail. For more information, see <i>Add Angle Dimension</i> (on page 107).	ANGLEDIM

	Change Tube To Rod - Changes a tube in the tubesheet to a rod. For more information, see <i>Change Tube To Rod</i> (on page 108).	CTTR
	Change Rod to Tube - Changes a rod in a tubesheet to a tube. For more information, see <i>Change Rod to Tube</i> (on page 108).	CRTT
	Add Rod - Adds a rod to a tubesheet. For more information, see <i>Add Rod</i> (on page 109).	ADDROD
	Remove Rod - Removes a rod from a tubesheet. For more information see <i>Remove Rod</i> (on page 109).	RMROD
	Make Tubesheet Groove - Adds a groove to a partition. For more information, see <i>Make Tubesheet Groove</i> (on page 110).	MTSGR
	Remove Tubesheet Groove - Removes a groove to a partition. For more information, see <i>Remove Tubesheet Groove</i> (on page 110).	RTSGR
	Remove Tube Hole - Removes a tube hole on the tubesheet. For more information, see <i>Remove Tube Hole</i> (on page 110).	RMTSHOLE
	Restore Tube Hole - Restores a tube hole on the tubesheet. For more information, see <i>Restore Tube Hole</i> (on page 111).	RSTSHOLE
	Reset Settings - Resets the tags on the selected detail in the drawing. For more information, see <i>Reset Settings</i> (on page 113).	RESDTSET

None Selection drop down - Specifies the components from which you can select to generate details. This list shows **None Selected** when there is no item selected, or shows the name, such as **Skirt** of the item when that item is selected.

No Item selected / General Settings - Describes the selected item. **No Item selected** displays only when an item has not been selected.

- **Parts Selection** - Specifies the parts to view of the detail.
- **View** - Specifies the views available for the detail.
 - **All View** - Displays the detail from all views.
 - **Top View** - Displays the detail in a top view.
 - **Elevation View** - Displays the detail from an elevation view.
 - **Front View** - Displays the detail from a front view.


NOTE Views vary by the detail parts.

- **Scale** - Specifies the scale of the detail.
- **Sub Scale** - Specifies the scale of the sub details. These are extra details that can be extracted from the main detail. For more information, see *Examine and Manipulate Tubesheet details* (on page 100) and *Examine and Manipulate Baffle details* (on page 101).

NOTE These settings only apply to the detail before it is inserted in the drawing. After inserting the detail, click it, and the details display in the **Details** tab. For more information, see *Examine detail settings* (on page 96)

Common Settings

 Fabricator console: **Support Detail > Common Settings** 

 Command line: **DTSET**

Specifies parameters for drawing generation.

Type of Configuration Setting - Specifies the type of configuration setting you want to make changes to. This selection controls the options available on the rest of the dialog box. The available options are:

- **All Configuration** - Lists the main common settings. These are detailed below.
- **Saddle** - Specifies the **Tag** configuration settings for a Saddle.
- **Skirt** - Specifies the **Tag** configuration settings for a Skirt.
- **Lugs** - Specifies the **Tag** configuration settings for a Lug.
- **Legs** - Specifies the **Tag** configuration settings for a Leg.
- **Tubesheet** - Specifies the **Tag** configuration settings for a Tubesheet.
- **Baffle** - Specifies the **Tag** configuration for a Baffle.
- **Tube** - Specifies the **Tag** configuration for the Tube.
- **Rod** - Specifies the **Tag** configuration for the Rod.
- **Spacer** - Specifies the **Tag** configuration for the Spacer.
- **Impingement Plate** - Specifies the **Tag** configuration for the Impingement Plate.
- **Nut of Rod** - Specifies the **Tag** configuration for the Nut of Rod.
 - **Tag** - The following are the common settings for the **Tags** of the supports.
 - **Prefix** - Specifies the abbreviation of the **Tag** type. Example: **SK** for Skirt.
 - **Tag Start** - Specifies the starting point for a tag. The default is **A**.
 - **Tag Inc** - Specifies the tag increment. The default is **1**.

All Configuration - Specifies the object type for which you are setting the configuration. This selection controls the options available on the rest of the dialog box. The available options are:

- *2D Outline* (on page 40)
- *Dimensions* (on page 82)
- *Tag* (on page 82)
- *Section* (on page 82)
- *Hatch* (on page 83)
- *Text* (on page 83)

Save - Writes the current settings to a file so that you can use them on another project.

Load - Imports a settings file.

Apply - Applies the current settings to the active drawing.

Dimensions

Controls settings for automatic dimensioning.

Dimension Style - Allows you to select the automatic dimensioning style from the list. The default is **Standard**.

Text Height - Allows you to type the automatic dimensioning text height. The default is **0.5000**.

Tag

Controls settings for support detail tag. These can also be changed by selecting the detail.

- **Layer** - Allows you to select the layer from the list. Default is **0**.
- **Balloon Type** - Allows you to select the balloon type from the list. The default is **Circle**.
- **Balloon Color** - Allows you to select the balloon color from the list. The default is **ByLayer**.
- **Balloon Linetype** - Allows you to select the balloon linetype from the list. The default is **ByLayer**.
- **Balloon Size** - Allows you to type the balloon size. The default is **0.5000**.
- **Text Style** - Allows you to select the text style from the list. The default is **Standard**.
- **Text Color** - Allows you to select the text color from the list. The default is **ByLayer**.
- **Text Height** - Allows you to type the text height. The default is **0.125**.

Section

Controls settings for support detail section.

- **Divide Line Layer** - Allows you to select the divide line layer from the list. The default is **0**.
- **Divide Line Color** - Allows you to select the divide line color from the list. The default is **Red**.
- **Divide Line Linetype** - Allows you to select the divide line linetype from the list. The default is **Divide**.
- **Thread Line Layer** - Allows you to select the thread line layer from the list. The default is **0**.
- **Thread Line Color** - Allows you to select the thread line color from the list. The default is **Color 115**.
- **Thread Line Linetype** - Allows you to select the thread line linetype from the list. The default is **ByLayer**.
- **Hidden Thread Line Layer** - Allows you to select the hidden thread line layer from the list. The default is **0**.
- **Hidden Thread Line Color** - Allows you to select the hidden thread line color from the list. The default is **Color 117**.
- **Hidden Thread Line Linetype** - Allows you to select the hidden thread line linetype from the list. The default is **HIDDEN**.
- **Wave Line Layer** - Allows you to select the wave line layer from the list. The default is **0**.
- **Wave Line Color** - Allows you to select the wave line color from the list. The default is **Color 116**.
- **Wave Line Linetype** - Allows you to select the wave line linetype from the list. The default is **ByLayer**.

Hatch

Controls settings for support detail hatch.

- **Hatch Layer** - Allows you to select the hatch layer from the list. The default is **0**.
- **Hatch Color** - Allows you to select the hatch color from the list. The default is **Color 13**.
- **Hatch Linetype** - Allows you to select the hatch linetype from the list. The default is **ByLayer**.
- **Hatch Space** - Allows you to type hatch space. The default is **1**.
- **Hatch Angle** - Allows you to type hatch angle. The default is **0**.

Text

Controls settings for support detail text. These can also be changed by selecting the detail.

- **Text Title Layer** - Allows you to select the text title layer from the list. The default is **0**.
- **Text Title Color** - Allows you to select the text title color from the list. The default is **Green**.
- **Title Text Height** - Allows you to type the title text height. The default is **0.125**.
- **Title Text Style** - Allows you to select the title text style from the list. The default is **Standard**.
- **Text Note Layer** - Allows you to select the text note layer from the list. The default is **0**.
- **Text Note Color** - Allows you to select the text note color from the list. The default is **White**.
- **Text Note Height** - Allows you to type the text note height. The default is **0.125**.
- **Text Note Style** - Allows you to select the text note style from the list. The default is **Standard**.

Tubesheet Configuration

 Fabricator console: **Details > Tubesheet Config** 

 Command line: **TSCFG**

Enables you to specify the **Tubesheet Properties**, **Tube Layout Data**, **Tube Hole Properties**, create details for **Baffle/Tube Supports**, **Rod, Nut Properties**, and **Tube Bundle**.

If you update any of these properties the tubesheet appearance changes in the layout.

See Also

- *Tubesheet Properties* (on page 84)
- *Tube Layout Data* (on page 85)
- *Tube Hole Properties* (on page 86)
- *Baffle/Tube Supports* (on page 87)
- *Rod, Nut Properties* (on page 89)
- *Tube Bundle* (on page 90)
- *Configure a tubesheet, baffle, and tube bundle for detail* (on page 91)
- *Generate Tubesheet, Baffle, or Tube Bundle details* (on page 96)
- *Examine detail settings* (on page 96)

Tubesheet Properties

General Exchanger Data

Tubesheet Analysis Method - Specifies the rules used for the tubesheet analysis. There are four available methods: **TEMA**, **PD 5500**, **ASME UHX**, and **EN 13445**.

Exchanger Type - Specifies the type of the exchange where the tubesheet is located. There are three available types: **Fixed**, **U-Tube**, and **Floating**.

TEMA

NOTE This is disabled for all **Tubesheet Analysis Methods**, except **TEMA**.

TEMA Exchange Notation - Specifies the three letter notation of the TEMA exchanger.

TEMA Exchange Class - Specifies the letter representing the class for the TEMA exchanger.

Tubesheet Location

Front - Specifies the front properties for a tubesheet. Select this to enter the **Front** property sets as listed below.

Rear - Specifies the rear properties for a tubesheet. Select this to enter the **Rear** property sets as listed below.

Description - Enables you to enter a description of the tubesheet. This description displays in the **Support Details** tab in the **None Selected** list.

Front/Rear Tubesheet Properties

Tubesheet Type - Specifies the tubesheet type based on the selection of the tubesheet method.

Outside Diameter - Specifies the outside diameter of the tubesheet.

Tubesheet Thickness - Specifies the thickness of the tubesheet.

Depth of Groove in Tubesheet - Specifies the depth of the groove in the tubesheet.

Weld Leg at back Tubesheet - Specifies the distance of the weld leg at the back on the tubesheet.

Tubesheet Extend as Flange - Specifies if the tubesheet extends as a flange. This is unavailable unless needed based on the **Tubesheet Type**.

Tubesheet of Extended portion - Specifies the length of the extended portion. This is unavailable unless needed based on the **Tubesheet Type**.

Material - Specifies the material for the tubesheet. Select a material type from the list.

Front/Rear Flange Properties

NOTE This is available only when required by the **Tubesheet Type**.

Number Of Bolts - Specifies the number of bolts located on the flange of the tubesheet.

Bolt Rotation Angle - Specifies the bolt rotation on the flange of the tubesheet.

Bolt Circle Diameter - Specifies the diameter of the bolt circle on the flange of the tubesheet.

Bolt Hole OD - Specifies the outside diameter of the bolt hole on the flange of the tubesheet.

Front/Rear Backing Ring

NOTE This is available only when required by the **Tubesheet Type**.

Backing Ring Thickness - Specifies the thickness of the backing ring on the tubesheet.

Backing Ring ID - Specifies the inside diameter of the backing ring on the tubesheet.

Backing Ring OD - Specifies the outside diameter of the backing ring on the tubesheet.

G Dimension for Backing Ring - Specifies the G dimension of the backing ring on the tubesheet.

Material - Specifies the material for the backing ring. Select a material type from the list.

Misc

Raised Face OD - Specifies the outside diameter of the raised face.

Raised Face - Specifies the size of the raised face.

Raised Face Shell Side OD - Specifies the outside diameter of the raised face shell side.

Raised Face Shell Side - Specifies the size of the raised face shell side.


Channel Inside Diameter - Specifies the inside diameter of the channel for the tubesheet.


NOTES


- If a value entered is invalid, a message appears at the bottom middle of the **Tubesheet Configuration** dialog box stating that this value is invalid and providing information for a solution.
- If you click **OK** before moving on to *Tube Layout Data* (on page 85), the system accepts information in all the other tabs.


Tube Layout Data

Partition Layout Type

Type A  - Specifies the partition layout type based on the number of partitions. The number of partitions on the tubesheet can be viewed in **Preview**.

Type B  - Specifies the partition layout type based on the number of partitions. The number of partitions on the tubesheet can be viewed in **Preview**.

Type C  - Specifies the partition layout type based on the number of partitions. The number of partitions on the tubesheet can be viewed in **Preview**.

Type D  - Specifies the partition layout type based on the number of partitions. The number of partitions on the tubesheet can be viewed in **Preview**.

Number Of Horizontal Partitions - Specifies the amount of horizontal partitions in the tubesheet. This setting has a maximum value of **8** and a minimum value of **0** for **Type A**. For **Type B** and **C** it has a maximum value of **8** and a minimum value of **2**.

Partitions Thickness - Specifies the thickness of the partitions. All partitions change thickness according to this setting. There is no individual partition thickness setting.

Partitions Clearance - Specifies the clearance of the partitions. All partitions change clearance according to this setting. There is no individual partition clearance setting.

Tube Information

Number Of Tubes - Displays the number of tubes in the tubesheet. This value is also imported when you import a model, with tubesheets, from PV Elite.

Tube Pattern - Specifies the pattern of the tubes inside the tubesheet. There are four patterns: **Triangle**, **Rotated Triangle**, **Square**, and **Rotated Square**.

Tube Boundary (OTL) - Specifies the boundary of the tubesheet. This setting works in conjunction with the **Shell Outside Diameter** setting.

Tube Outer Diameter - Specifies the outside diameter of the tubes inside the tubesheet.

Tube Pitch - Specifies the spacing distance between the tubes on the inside of the tubesheet.

Material - Specifies the material for the tube information. Select a material type from the list.

Nozzle Clearance

Nozzle Clearance - Top - Specifies the clearance of the tubes from the top nozzle.

Nozzle Clearance - Bottom - Specifies the clearance of the tubes from the bottom nozzle.

Nozzle Clearance - Left - Specifies the clearance of the tubes from the left nozzle.

Nozzle Clearance - Right - Specifies the clearance of the tubes from the right nozzle.

Shell

Shell Inside Diameter - Specifies the inside diameter of the shell. This is the inner circle of the tubesheet. This setting works in conjunction with the **Tube Boundary (OTL)** setting.

Preview

Displays a preview of the appearance of the tubesheet and the total number of tubes. As you enter the data in the fields you can view the changes in this box.

NOTES

- If a value entered is invalid, a message appears at the bottom middle of the **Tubesheet Configuration** dialog box stating that this value is invalid and providing information for a solution.
- If you click **OK** before moving on to *Tube Hole Properties* (on page 86), the system accepts the information in all the other tabs.

Tube Hole Properties

Tube Hole properties

Include Tube Hole Groove - Includes the tube hole groove on the tubesheet. If you do not want to include the tube hole groove, clear this check box, making the information unavailable.

(A) Groove Depth - Specifies the depth of groove for the tube hole. See the image below.

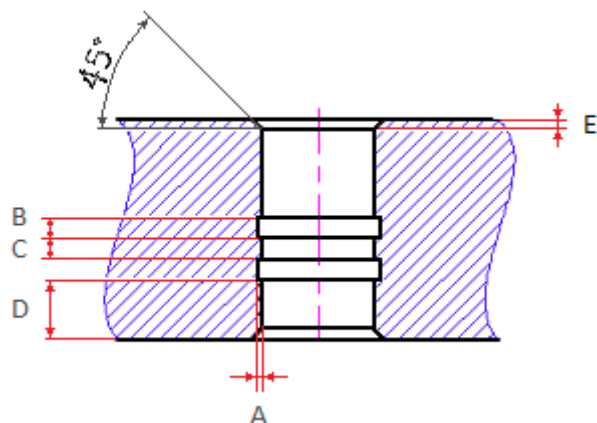
(B) Groove Width - Specifies the width of the groove for the tube hole. See the image below.

(C) Space Between Grooves - Specifies the space between the grooves of the tube hole. See the image below.

(D) Grooves Distance Edge - Specifies the distance of the groove from the edge of the tube hole. See the image below.

(E) Hole Ream - Specifies the size of the hole ream for the tube hole. See the image below.

Num of Groove in Hole - Specifies the amount of grooves in the hole. This is not displayed in the image.



NOTES

- If a value entered is invalid, a message appears at the bottom middle of the **Tubesheet Configuration** dialog box stating that this value is invalid and providing information for a solution.
- If you click **OK** before moving on to *Baffle/Tube Supports* (on page 87), the system accepts the information in all the other tabs.

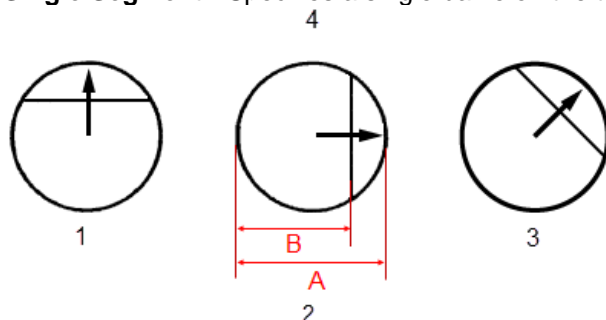
Baffle/Tube Supports

Include Baffle - Includes the baffle on the tubesheet. If you do not want a baffle with the tubesheet clear this check box, making the information unavailable.

Baffle Information

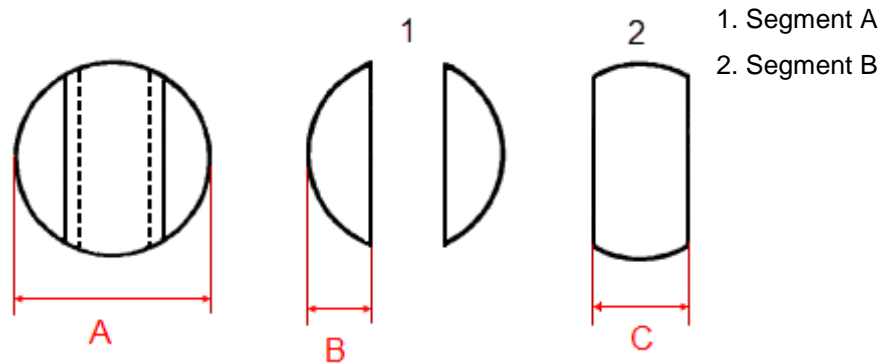
Segment Type - Enables you to select the between three different types of baffle segments.

Single Segment - Specifies a single baffle on the tubesheet.

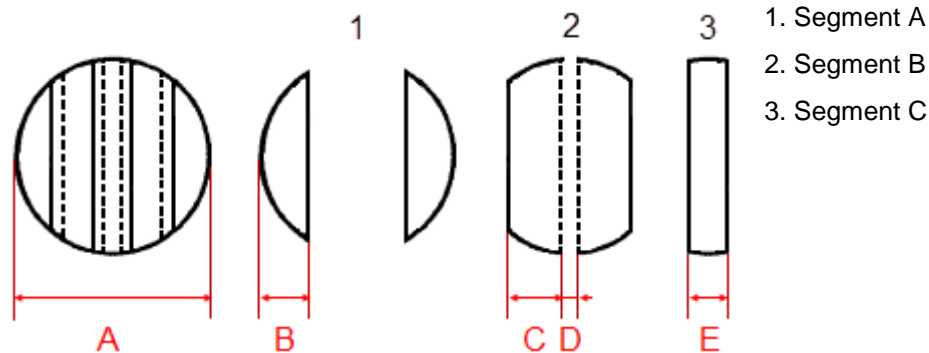


1. Horizontal
2. Vertical
3. Rotated
4. Segment A

- **Double Segment** - Specifies a double baffle on the tubesheet.



- **Triple Segment** - Specifies a triple baffle on the tubesheet.



NOTE These selections work in conjunction with the **Orientation Type**.

(A) Baffle OD - Specifies the outside diameter of the baffle. See letter **A** in the images above.

(B) Segment A Width - Specifies the width of the baffle for **Segment A**. See letter **B** in the images above.

(C) Segment B Width - Specifies the double width for a double or triple segment baffle for **Segment B**. See letter **C** in the images above.

(D) Segment B Gap - Specifies the distance gap between a triple segment baffle for **Segment B**. See letter **D** in the **Triple Segment** image above.

(E) Segment C Width - Specifies the width of the triple segment baffle for **Segment C**. See letter **E** in the **Triple Segment** image above.

Baffle Thickness - Specifies the thickness of the baffle.

Orientation Type - Specifies the orientation of the baffle.

- **Horizontal** - Specifies a horizontal baffle.
- **Vertical** - Specifies a vertical baffle.
- **Rotated** - Specifies a rotated baffle.

Rotation Angle - Specifies the rotation of the baffle when the **Rotated** option is selected for **Orientation Type**.

Material - Specifies the material for the baffle. Select a material type from the list.

Number of Segments - Specifies the number of segments for the baffle. The number of segments is the average of this value and priority according to the order of appearance of the segment.

NOTES

- If a value entered is invalid, a message appears at the bottom middle of the **Tubesheet Configuration** dialog box stating that this value is invalid and providing information for a solution.
- If you click **OK** before moving on to *Rod, Nut Properties* (on page 89), the system accepts the information in all the other tabs.

Rod, Nut Properties

Include Rod - Sets the software to include the rod in the tubesheet. If you do not want a rod with the tubesheet clear this check box, making the information unavailable.

Rod Properties

Rod Diameter - Specifies the diameter for the rod.

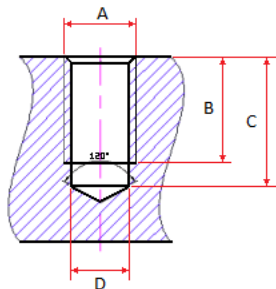
(A) Rod UNC - Specifies the diameter of the rod for the threads. See the image below.

(B) Rod depth - Specifies the depth of the rod. See the image below.

(C) Rod drill depth - Specifies the depth of the drill. See the image below.

(D) Rod drill size - Specifies the size of the actual drill. See the image below.

Material - Specifies the material for the rod. Select a material type from the list.



Nut Properties

Number of Nut - Specifies the number of nuts on a rod. See the image below.

(S) Nut UNC - Specifies the diameter of the threads for the nut. See the image below.

Pitch of Nut UNC - Specifies the pitch of the nut for the threads. See the image below.

(W) Width Across Flats - Specifies the width of the nut across the flats. See the image below.

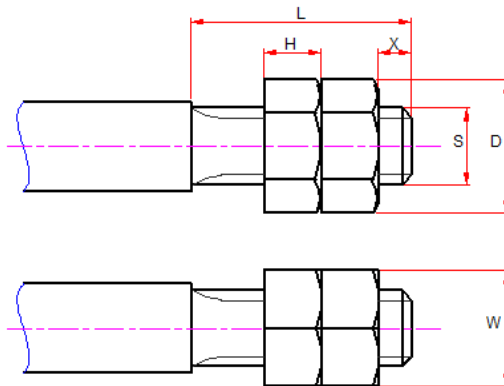
(D) Width Across Corners - Specifies the width of the nut across the corners. See the image below.

(H) Thickness of Nut - Specifies the thickness of the nut. See the image below.

(X) Thread Extension Length - Specifies the thread extension link above the nut. See the image below.

(L) Thread Length - Specifies the thread length on the rod. See the image below.

Material - Specifies the material for nut of the rod. Select a material type from the list.



NOTES

- If a value entered is invalid, a message appears at the bottom middle of the **Tubesheet Configuration** dialog box stating that this value is invalid and providing information for a solution.
- If you click **OK** before moving on to *Tube Bundle* (on page 90), the system accepts the information in all the other tabs.

Tube Bundle

Spacer Info

Values are available when **Include Rod** on the **Rod, Nut Properties** tab is selected.

Spacer Outer Diameter - Specifies the outside diameter for the spacer in the tube bundle.

Material - Specifies the material for the tube information. Select a material type from the list.

Impingement Plate Info

Include Impingement Plate - Includes the impingement plate on the tube bundle. If you do not want to include the impingement plate, clear this check box, making the information unavailable.

Diameter - Specifies the diameter of the impingement plate.

Thickness - Specifies the thickness of the impingement plate.

Material - Specifies the material for the tube information. Select a material type from the list.

Distance to Tubesheet - Specifies the distance from the impingement plate to the tubesheet.

Tube Length Info


Straight Tube Length - Specifies the length of the tube.

Straight Tube Length measured between - Specifies the length of the tube as measured between the **Inner Faces** or the **Outer Faces**.

Extended Tube Length - Specifies the length of the extended tube.

Dist. from Tubesheet to Baffle - Specifies the distance from the tubesheet to the baffle.

Configure a tubesheet, baffle, and tube bundle for detail

1. Click **Tubesheet Config** . Alternatively, type **TSCFG** on the command line, and then press ENTER.

The Tubesheet Configuration dialog box displays.

NOTE When importing from PV Elite some if not all of the information in the **Tubesheet Configuration** is automatically populated and you can just accept it by clicking **OK**. The following steps are provided so that you might change information contained in a Tubesheet, Baffle, Tube Bundle, or create details from scratch. If a box is unavailable, then the option does not apply to the selected tubesheet type.

Tubesheet Properties tab

1. Under **General Exchanger Data**, do the following:
 - a. Select a **Tubesheet Analysis Method**. There are four methods: **TEMA**, **PD 5500**, **ASME UHX**, and **EN 13445**. Your options vary depending on which of these methods you select. For more information, see *Tubesheet Properties* (on page 84).
 - b. Select an **Exchanger Type**. There are three types: **U-tube**, **Fixed**, and **Floating**.
2. Under **TEMA**, do the following only when you use the **TEMA** method. This option is unavailable for all other methods.
 - a. Select the 3 types of **TEMA Exchange Notation(s)** using the 3 separate lists.
 - b. Select the **TEMA Exchange Class**.
3. Under **Tubesheet Location**, select **Front** or **Rear**. Selecting **Front** enables you to enter data for the **Front Tubesheet Properties**, **Front Backing Ring**, and **Front Flange Properties** for the tubesheet. Selecting **Rear** enables you to enter data for the rear options of these settings.
4. Type a description of the tubesheet in the **Description** box.
5. Under **Front/Rear Tubesheet Properties**, do the following:
 - a. Select a **Tubesheet Type**.
 - b. Type an outside diameter in the **Outside Diameter** box.
 - c. Type a tubesheet thickness in the **Tubesheet Thickness** box.
 - d. Type the depth of the groove in the tubesheet in the **Depth of Groove in Tubesheet** box.
 - e. Type a value for the weld leg at the back of the tubesheet in the **Weld Leg at back Tubesheet** box.
 - f. Select **Tubesheet Extend as Flange** when available and applicable.
 - g. Type a thickness of the extended portion of the tubesheet in the **Thickness of Extended portion** box.
 - h. Select **Material** for the tubesheet.
6. Under **Front/Rear Backing Ring**, do the following when available and applicable:
 - a. Type a thickness for the backing ring in the **Backing Ring Thickness** box.
 - b. Type an inside diameter for the backing ring in the **Backing Ring ID** box.
 - c. Type an outside diameter for the backing ring in the **Backing Ring OD** box.
 - d. Type a G dimension for the backing ring in the **G Dimension for Backing Ring** box.
 - e. Select **Material** for the backing ring.

7. Under **Front/Rear Flange Properties**, do the following when available and applicable:
 - a. Type the number of bolts on the flange in the **Number Of Bolts** box.
 - b. Type a value for the bolt rotation in the **Bolt Rotation Angle** box.
 - c. Type a radius for the bolt circle in the **Bolt Circle Diameter** box.
 - d. Type an outside diameter in the **Bolt Hole OD** box.
8. Under **Misc**, do the following:
 - a. Type an outside diameter of the raised face attached to the tubesheet in the **Raised Face OD** box.
 - b. Type a value for the raised face in the **Raised Face** box.
 - c. Type an outside diameter of the raised face shell side attached to the tubesheet in the **Raised Face Shell Side OD** box.
 - d. Type a value for the raised face shell side in the **Raised Face Shell Side** box.
 - e. Type a value for the inside diameter of the channel in the **Channel Inside Diameter** box.
9. Select the *Tube Layout Data* tab (on page 92).

Tube Layout Data tab

NOTE The information entered here affects the preview to the right. Some of the fields correlate with one another so that each field you change affects the appearance of the tubesheet. When making a change, you may receive an **Invalid value** message based on how you laid out the boundaries of your tubesheet. For more information, see *Tube Layout Data* (on page 85).

1. Under **Partition Layout Type**, do the following:
 - a. Select a **Type A**, **Type B**, **Type C**, or **Type D** for the partition type. This determines how many lanes are in the tube sheet. For more information, see *Tube Layout Data* (on page 85).
 - b. Type a value of **0** through **8** or use the arrows to determine the amount of horizontal partitions in the **Number of Horizontal Partitions** box. If you select B or C partition types you are not able to have less than two partitions.
 - c. Type the thickness of the partitions in the **Partitions Thickness** box.
 - d. Type the clearance of the partitions in relation to the tubes, boundary, and shell in the **Partitions Clearance** box.
2. Under **Tube Information**, do the following:
 - a. Select a tube pattern in the **Tube Pattern** list.
 - b. Type the boundary of the tube in the **Tube Boundary (OTL)** box.
 - c. Type the outside diameter of the tube in the **Tube Outer Diameter** box.
 - d. Type the pitch of the tubes in the **Tube Pitch** box. You can examine the pitch of the tubes when you generate the detail. For more information, see *Examine and Manipulate Tubesheet details* (on page 100).
 - e. Select **Material** for the tubes.

3. Under **Nozzle Clearance**, do the following:
 - a. Type the top clearance of the tubes from the top nozzle in the **Nozzle Clearance - Top** box.
 - b. Type the bottom clearance of the tubes from the bottom nozzle in the **Nozzle Clearance - Bottom** box.
 - c. Type the left clearance of the tubes from the left nozzle in the **Nozzle Clearance - Left** box.
 - d. Type the right clearance of the tubes from the right nozzle in the **Nozzle Clearance - Right** box.
4. Type the inside diameter of the shell in the **Shell Inside Diameter** box.
5. Select the *Tube Hole Properties* tab (on page 93).

***Tube Hole Properties* tab**

NOTE The letters in front of the properties correspond to the letters in the diagram to the right. For more information, see *Tube Hole Properties* (on page 86).

1. Under **Tube Hole Properties**, do the following:
 - a. Type the depth of the groove in the **(A) Groove Depth** box.
 - b. Type the width of the groove in the **(B) Groove Width** box.
 - c. Type the amount of space between the grooves in the **(C) Space Between Grooves** box.
 - d. Type the distance to the edge of the groove in the **(D) Grooves Distance Edge** box.
 - e. Type the size of the hole ream in the **(E) Hole Ream** box.
 - f. Type the number of grooves in the hole in the **Num of Groove in Hole** box.
2. Select the *Baffle/Tube Supports* tab (on page 93).

***Baffle/Tube Supports* tab**

NOTE The letters in front of the properties correspond to the letters in the diagram to the right. For more information, see *Baffle/Tube Supports* (on page 87).

1. Under **Baffle Information**, do the following:
 - a. Select a segment type from the **Segment Type** list. For more information, see *Baffle/Tube Supports* (on page 87).
 - b. Type the outside diameter of the baffle in the **(A) Baffle OD** box.
 - c. Type the width of the baffle in the **(B) Baffle Width** box.
 - d. Type the double width of the baffle segment in the **(C) Double Width** box. This option is only available with **Double** and **Triple** segments. For more information, see *Baffle/Tube Supports* (on page 87).
 - e. Type the distance of the spacing for the double gap in the **(D) Double Gap** box. This option is only available with a **Triple** segment. For more information, see *Baffle/Tube Supports* (on page 87).
 - f. Type the triple width of the baffle in the **(E) Triple Width** box. This option is only available with a **Triple** segment. For more information, see *Baffle/Tube Supports* (on page 87).

- g. Type the thickness of the baffle in the **Baffle Thickness** box.
- h. Select the orientation of the baffle from the **Orientation Type** list. For more information, see *Baffle/Tube Supports* (on page 87).
- i. Type the rotation angle of the baffle in the **Rotation Angle** box. This option is only available when the **Orientation Type** of **Rotated** is selected.
- j. Select **Material** for the baffle.

NOTE If a value entered is invalid, a message appears at the bottom middle of the **Tubesheet Configuration** dialog box stating that this value is invalid and providing information for a solution.

- 2. Select the *Rod, Nut Properties tab* (on page 94).

Rod, Nut Properties tab

- 1. Under **Rod Properties**, do the following:
 - a. Type the diameter of the rod in the **Rod Diameter** box.
 - b. Type the diameter of the rod for the threads in the **(A) Rod UNC** box.
 - c. Type the depth of the rod in the **(B) Rod depth** box.
 - d. Type the depth of the drill in the **(C) Rod drill depth** box.
 - e. Type the size of the rod drill in the **(D) Rod drill size** box.
 - f. Select **Material** for the rods.
- 2. Under **Nut Properties**, do the following:
 - a. Type the number of nuts in the **Number of Nut** box. Alternatively, click the arrow buttons to increase or decrease the number of nuts. The maximum value allowed is **8**. If you enter a value greater than 8, the software returns the box to the previous value.
 - b. Type the diameter of the threads for the nut in the **(S) Nut UNC** box.
 - c. Type the pitch of the nut for the threads in the **Pitch of Nut UNC** box.
 - d. Type the width of the nut across the flats in the **(W) Width Across Flats** box.
 - e. **(D) Width Across Corners** updates automatically from the value entered in the **(W) Width Across Flats** box.
 - f. Type the thickness of the nut in the **(H) Thickness of Nut** box.
 - g. Type extension length of the thread for the nut in the **(X) Thread Extension Length** box.
 - h. Type the length of the thread on the rod in the **(L) Thread Length** box.
 - i. Select **Material** for the nuts.
- 3. Select the *Tube Bundle tab* (on page 94).

Tube Bundle tab

- 1. Under **Spacer Info**, do the following:
 - a. Type the outer diameter of the spacer in the **Spacer Outer Diameter**.
 - b. Select **Material** for the spacer information.

2. Under **Impingement Plant Info**, do the following:
 - a. Type the diameter of the impingement plate in the **Diameter** box.
 - b. Type the thickness of the impingement plate in the **Thickness** box.
 - c. Select **Material** for the impingement plate.
 - d. Type the value of the distance from the impingement plate to the tubesheet in the **Distance to Tubesheet** box.
3. Under **Tube Length Info**, do the following:
 - a. Type the length of the straight tube in the **Straight Tube Length** box.
 - b. Select **Inner Faces** or **Outer Faces** to determine whether the straight tube length is measured between the inner faces or the outer faces of the tube bundle.
 - c. Type the length of the extended tube in the **Extended Tube Length** box.
 - d. Type a value for the distance from the tubesheet to the baffle in the **Dist. from Tubesheet to Baffle**.


NOTE If a value entered is invalid, a message appears at the bottom middle of the **Tubesheet Configuration** dialog box stating that this value is invalid and providing information for a solution.

4. Click **OK**.

NOTE To generate the tubesheet, baffle, or tube bundle that you just configured, see *Generate Tubesheet, Baffle, or Tube Bundle details* (on page 96).


Generate Detail

 Fabricator console: **Details > Generate Detail** 

 Command line: **GND**



Generates a detailed drawing.

Generate a support detail

1. Click a paper space tab.
2. Select the component to generate detail. These are components such as **Skirt**, **Legs**, **Lugs**, and **Saddle**.
3. In the **General Setting** section, select the **View** list, and click an available view to display.
4. Click **Generate Detail** .
5. Click in the paper space to place the detail drawing.
The detail displays in the paper space.


Create Sketch for extra details from a support detail

It is possible to examine the extra details from a support detail.


1. In the drawing, select the support detail.
Pink grip points  display where detail can be pulled from the support detail.
 2. Click a pink grip point , and drag it into the drawing to view the detail.
- The detail displays.

Generate Tubesheet, Baffle, or Tube Bundle details

 Fabricator console: **Details > Generate Detail** 

 Command line: **GND**

Generates a detailed drawing.

1. Click a paper space tab.
2. Select the **Tubesheet, Baffle, or Tube Bundle** in the list to generate the detail.
3. In the **General Setting** section, select the **View** list, and click an available view to display.
4. Set the **Scale** and **Sub Scale**, if needed. The default value of **Scale** is **0.1**, and the default value of **Sub Scale** is **0.5**.
5. Click **Generate Detail** .
6. Click in the paper space to place the detail drawing.

The detail displays in the paper space.

See Also

- *Examine and Manipulate Tubesheet details* (on page 100)
- *Examine and Manipulate Baffle details* (on page 101)
- *Examine and Manipulate Tube Bundle details* (on page 102)

Examine detail settings

After the detail is generated, you can change the **2D Outlines, Tag settings, Text Settings, Section Settings, Hatch Settings, and BOM Settings**. The default settings are retrieved from the **Common Settings**. The settings can be changed at the part level (double-clicking the part itself and making changes), the detail level (selecting the part and making changes to the **Tag setting**), and in the *Common Settings* (on page 81).

- To change the detail settings, select the detail, and then select the options displayed in the **Details** tab. Some of the settings below can also be changed in **Common Setting**. **Common Setting** in the **Detail** tab can be accessed by selecting the part. To access the remaining settings listed below, select the **COMMON SETTING** list, and then select the part name, such as **Front Tubesheet**.

2D Outline - Lists the following detail options:

- **Main Layer** - Allows you to select the main layer from the list. The default is **0**.
- **Main Color** - Allows you to select the main color from the list. The default is **Green**.
- **Main Linetype** - Allows you to select the main linetype from the list. The default is **ByLayer**.
- **Hidden Layer** - Allows you to select the hidden layer from the list. The default is **0**.
- **Hidden Color** - Allows you to select the hidden color from the list. The default is **Cyan**.
- **Hidden Linetype** - Allows you to select the hidden linetype from the list. The default is **HIDDEN**.
- **Center Layer** - Allows you to select the center layer from the list. The default is **0**.

- **Center Color** - Allows you to select the center color from the list. The default is **Yellow**.
- **Center Linetype** - Allows you to select the center linetype from the list. The default is **CENTER**.
- **Scale** - Allows you to select the scale from the list. This value determines the size of the main detail when inserted in the drawing. The default is **0.1000**.
- **Sub Details Scale** - Specifies the scale of the sub details. This value determines the size of the extra detail when inserted in the drawing. The default is **0.2000**. These are extra details that can be extracted from the main detail. For more information, see *Examine and Manipulate Tubesheet details* (on page 100) and *Examine and Manipulate Baffle details* (on page 101).

NOTE The lists are read-only. Select the required item from the list to change the setting.

Tag setting - Lists the following tag settings of the detail:

- **Layer** - Allows you to select the layer from the list. The default is **0**.
- **Balloon Type** - Allows you to select the balloon type from the list. The default is **Circle**.
- **Balloon Color** - Allows you to select the balloon color from the list. The default is **ByLayer**.
- **Balloon Linetype** - Allows you to select the balloon linetype from the list. The default is **ByLayer**.
- **Balloon Size** - Allows you to type the balloon size. The default is **0.5000**.
- **Text Style** - Allows you to select the text style from the list. The default is **Standard**.
- **Text Color** - Allows you to select the text color from the list. The default is **ByLayer**.
- **Text Height** - Allows you to type the text height. The default is **0.1250**.
- **Tag prefix** - Allows you to type the tag prefix. The default is the detail currently available.
- **Tag start** - Allows you to type the tag start. The default is **A**.
- **Tag inc** - Allows you to type the tag increment. The default is **1**.

NOTE You can only enter information into the **Balloon Size**, **Text Height**, **Tag prefix**, **Tag start**, and **Tag inc** settings in this section. For the remaining settings, you must select an item from the list.

Text Settings - Lists the following text settings of the detail:

- **Title Layer** - Allows you to select the title layer from the list. The default is **0**.
- **Text Title Color** - Allows you to select the title color from the list. The default is **Green**.
- **Title Text Style** - Allows you to select the title text style from the list. The default is **Standard**.
- **Title Text Height** - Allows you to type the title text height. The default is **0.1250**.
- **Note Layer** - Allows you to select the note layer from the list. The default is **0**.
- **Note Text Color** - Allows you to select the note text color from the list. The default is **White**.
- **Note Text Style** - Allows you to select the note text style from the list. The default is **Standard**.
- **Note Text Height** - Allows you to type the note text height. The default is **0.1250**.

NOTE You can only enter information into the **Title Text Height** and the **Note Text Height** settings in this section. For the remaining settings, you must select an item from the list.

Section Settings - Lists the following section settings of the detail.

- **Divide Line Layer** - Allows you to select the divide line layer from the list. The default is **0**.
- **Divide Line Color** - Allows you to select the divide line color from the list. The default is **Red**.
- **Divide Line Linetype** - Allows you to select the divide line linetype from the list. The default is **Divide**.
- **Thread Line Layer** - Allows you to select the thread line layer from the list. The default is **0**.
- **Thread Line Color** - Allows you to select the thread line color from the list. The default is **Color 115**.
- **Hidden Thread Line Layer** - Allows you to select the hidden thread line layer from the list. The default is **0**.
- **Hidden Thread Line Color** - Allows you to select the hidden thread line color from the list. The default is **Color 117**.
- **Hidden Thread Line Linetype** - Allows you to select the hidden thread line linetype from the list. The default is **HIDDEN**.
- **Thread Line Linetype** - Allows you to select the thread line linetype from the list. The default is **ByLayer**.
- **Wave Line Layer** - Allows you to select the wave line layer from the list. The default is **0**.
- **Wave Line Color** - Allows you to select the wave line color from the list. The default is **Color 116**.
- **Wave Line Linetype** - Allows you to select the wave line linetype from the list. The default is **ByLayer**.

Hatch Settings - Lists the following hatch settings of the tubesheet and baffle details.

NOTE Only **Hatch Space** and **Hatch Angle** can be entered under **Common Settings**. You must select a detail to edit the other hatch settings.


- **Hatch Layer** - Allows you to select the hatch layer from the list. The default is **0**.
- **Hatch Color** - Allows you to select the hatch color from the list. The default is **Color 13**.
- **Hatch Linetype** - Allows you to select the hatch linetype from the list. The default is **ByLayer**.
- **Hatch Space** - Allows you to type hatch space. The default is **1.0000**.
- **Hatch Angle** - Allows you to type hatch angle. The default is **0**.



BOM Setting - Lists the following bill of material settings:

- **Title** - Allows you to type the header title in the box. The default is the name of the bill of materials list selected.
- **Column Detail** - Allows you to set the column detail in the **Column Detail** dialog box. The default is three dots. For more information, see *Column Detail Dialog Box* (on page 99).
- **Layer** - Allows you to select the layer from the list. The default is **0**.
- **Line Color** - Allows you to select the line color from the list. The default is **ByLayer**.
- **Linetype** - Allows you to select the linetype from the list. The default is **ByLayer**.
- **Title Text Style** - Allows you to select the text style for the title from the list. The default is **STANDARD**.

- **Title Text Color** - Allows you to select the text color for the title from the list. The default is **ByLayer**.
- **Title Text Height** - Allows you to type the height of the title text in the box. The default is **1/2"**.
- **Title Row Height** - Allows you to type the height of the title row in the box. The default is **1/2"**.
- **Header Text Style** - Allows you to select the text style of the header from the list. The default is **STANDARD**.
- **Header Text Color** - Allows you to select the text color for the header from the list. The default is **ByLayer**.
- **Header Text Height** - Allows you to type the height of the header text in the box. The default is **1/2"**.
- **Header Row Height** - Allows you to type the height of the header row in the box. The default is **1/2"**.
- **Text Style** - Allows you to select the text style of all other text from the list. The default is **STANDARD**.
- **Text Color** - Allows you to select the text color of all other text from the list. The default is **ByLayer**.
- **Text Height** - Allows you to type the height of all other text in the box. The default is **1/2"**.
- **Row Height** - Allows you to type the height of all other rows in the box. The default is **1/2"**.

Create Sketch for extra details



In some detail you can it is possible to examine extra details of certain items. When the detail with extra detail is selected, a pink grip point  displays indicating the origin of the extra detail.

1. In the drawing select the detail.
Pink grip points  display at the origin of the detail.
2. Click a pink grip point , and drag it into the drawing to view the detail.
The detail displays.

NOTE **Sub Scale Details** enables you to change the size of the extra details in the drawing by selecting a size from the list.

Remove Sketch for extra details

In the tubesheet, it is possible to remove the extra details sketch from the drawing.

1. In the drawing, select the tubesheet.
Blue grip points  display on the extra detail.
2. Select a blue grip point , and drag it into the tubesheet to remove the extra detail.
The extra detail removes from the drawing.

Column Detail Dialog Box

Specifies the column detail settings for a bill of material **BOM settings**.

Data Column - Lists the available items that can be placed in a bill of materials.

Selected Column - Lists the items selected for appearance in a bill of materials.

Table Format

Row Height - Specifies the height of a row for the table format of a bill of material

Grow Direction - Specifies the grow direction of the table for the bill of materials.

Column Format

Header Title - Specifies the title of the specific column.

Column Width - Specifies the width of a specific column.

Alignment - Specifies the alignment of a specific column.

Examine and Manipulate Tubesheet details

Create Sketch for extra details

In the tubesheet, it is possible to examine the extra details of the tubesheet such as the pitch or **Tube Hole properties**.

1. In the drawing select the tubesheet.

Pink grip points  display where detail can be pulled from the tubesheet.

2. Click a pink grip point , and drag it into the drawing to view the detail.

The detail displays.


TIP If you want to edit an existing tubesheet detail, double-click it to open the **Tubesheet Configuration** dialog box.

Remove Sketch for extra details

In the tubesheet, it is possible to remove the extra details sketch from the drawing.

1. In the drawing, select the tubesheet.

Blue grip points  display on the extra detail.

2. Select a blue grip point , and drag it into the tubesheet to remove the extra detail.


The extra detail removes from the drawing.

Partition adjustment

In the tubesheet, it is possible to move the partitions around to adjust the number of tubes in a partition section.

1. In the drawing select the tubesheet.

Blue grip points  display on the partitions.

2. Select a blue grip point , and drag it in the direction you want to move the partition.

The partition changes location and adjusts the tubes accordingly.


TIPS

- Partitions can only be moved along the center vertical partition and cannot be moved if there is only one row of tubes between partitions.
- If you want to change the appearance of the detail settings, you can do this in the **Support Detail** tab. The following settings appear when you select a Tubesheet: **2D Outline**, **Tag setting**, **Text Settings**, and **Hatch Settings**. For more information on these settings, see *Examine detail settings* (on page 96).


Nozzle adjustment

In the tubesheet, it is possible to adjust the nozzles and limit the number of tubes in a section.

1. In the drawing select the tubesheet.

Blue grip points  display on the nozzle.

NOTE These are the four singular lines at the top, bottom, left, and right on the circle.

2. Select a blue grip point , and drag it in the direction you want to move the nozzle.

The nozzle changes location and adjusts the tubes accordingly.

Change Tube to Rod

Changes a tube in the tubesheet to a rod. For more information, see *Change Tube To Rod* (on page 108).

Change Rod to Tube

Changes a rod in the tubesheet to a tube. For more information, see *Change Rod to Tube* (on page 108).

Add a Rod

Adds a rod to the tubesheet. For more information, see *Add Rod* (on page 109).


Remove a Rod


Removes a rod from the tubesheet. For more information, see *Remove Rod* (on page 109).

Examine and Manipulate Baffle details

Baffle orientation adjustment for single segment baffle

1. In the drawing, select the baffle.

A yellow grip point  displays on the flat side of the baffle on the inside of the gap.

2. Select the yellow grip point , and drag the point in the direction you want to rotate the orientation.

The orientation adjusts.


3. Click any point to finish.

NOTE You can change the appearance of the detail settings in the **Support Detail** tab. The following settings appear when you select a Tubesheet: **2D Outline**, **Tag setting**, **Text Settings**, and **Hatch Settings**. The colors for the images can be changed by using these settings. For more information on these settings, see *Examine detail settings* (on page 96).

Baffle gap adjustment for double segment baffle

1. In the drawing select the baffle.



A light blue grip point  displays on the flat side of the baffle on the inside of the gap.

2. Select the light blue grip point , and drag the point in the direction you want to expand or lessen the distance of the gap.

The distance between the baffles changes and the tubes adjusts accordingly.

3. Click any point to finish.

Baffle width and double width adjustment

1. In the drawing select the baffle.
A dark blue grip point  displays on the flat side of the baffle on the inside of the gap.
2. Select the dark blue grip point , and drag the point in the direction you want to expand or lessen the distance of the width of the baffle.
The width changes and the tubes adjust accordingly.
3. Click anywhere to finish.

Change Tube to Rod

Enables you to change the tube to a rod.



For more information, see *Change Tube To Rod* (on page 108).

Change Rod to Tube

Enables you to change the rod to a tube.



For more information, see *Change Rod to Tube* (on page 108).

Create Sketch for extra details

1. In the drawing select the baffle.
Pink grip points  display at the point from which detail is pulled from the baffle.
2. Click a pink grip point , and drag the point into the drawing to view the detail.
The detail displays.

TIP If you want to edit an existing baffle detail, double-click the detail to open the **Tubesheet Configuration** dialog box.



Change position of baffle

1. In the drawing select the baffle.
Brown double arrow grip points  display.
2. Click a brown double arrow grip point , and drag the point in the direction you want the baffle to change.
The baffle changes direction.
3. Click anywhere to finish.

Examine and Manipulate Tube Bundle details

Create Sketch for extra details

In the drawing, you can examine the extra details of the tube bundle, such as the tie rods details or spacers.


1. In the drawing select the tube bundle.
Pink grip points  display at the point at which detail is pulled from the tube bundle.
2. Click a pink grip point , and drag the point into the drawing to view the detail.
The detail displays.

NOTE A rod must be placed in the drawing on the tubesheet before detail for tie rods or spacers can be extracted.


TIP If you want to edit an existing tube bundle detail, double-click the detail to open the **Tubesheet Configuration** dialog box.

Update Detail

 Fabricator console: **Details > Update Detail** 

 Command line: **UPD**

Updates the changes made to the model.

1. Make changes to the model, and then click a paper space tab.
2. Click **Update Detail** .
3. Select a detail, and then press ENTER.
4. Press ENTER to regenerate the support detail, or click **Yes**.

OR

Click or type **No** to cancel.

The detail is updated.

Generate Detail BOM

 Fabricator console: **Details > Generate Detail BOM** 

 Command line: **GENDTBOM**

Generates a bill of material for the detail drawing.

All - Generates a details bill of material (BOM) for all the details in the paper space.

Selection - Generates a details bill of material for the selected object in the paper space.

append to Existing BOM - Adds a details bill of material to an existing BOM in the paper space.

Current Layout - Generates a detail BOM for the currently selected paper space.


Drawing - Generates a detail BOM for the entire drawing. This includes all paper spaces.

NOTE For the **Drawing** option to appear differently from that of the **Current Layout** option you must have more than one paper space.

What do you want to do?


- *Generate All BOMs* (on page 103)
 - *Generate Selection BOMs* (on page 104)
 - *Append Existing BOMs* (on page 104)
 - *Move a BOM with a red grip point* (on page 104)
 - *Edit an existing BOM* (on page 104)
-

Generate All BOMs


1. Click a paper space tab.
2. Generate a detail. For more information, see *Generate Detail* (on page 95).
3. Click **Generate Detail BOM** .
4. Select **All**, or type **ALL** on the command line, and then press ENTER.

5. Select **Current Layout**, or type **C** on the command line, and then press ENTER.
The software generates the BOM details for the currently selected paper space.
OR
Select **Drawing**, or type **D** on the command line, and then press ENTER.
The software generates the BOM details for all available paper spaces.
6. Click in paper space to place the bill of materials.
The bill of materials is placed.



Generate Selection BOMs

1. Click a paper space tab.
2. Generate a detail. For more information, see *Generate Detail* (on page 95).
3. Click **Generate Detail BOM** .
4. Select **Selection**, or type **S** on the command line, and then press ENTER.
5. Select the details to be included in the BOM, and then press ENTER to finish.
6. Click in paper space to place the bill of materials.
The bill of materials is placed.



Append Existing BOMs

1. Click a paper space tab.
2. Generate a detail. For more information, see *Generate Detail* (on page 95).
3. Click **Generate Detail BOM** .
4. Select **append to Existing BOM**, or type **E** on the command line, and then press ENTER.
5. Select the BOM to append to.
6. Select the details to add to the BOM, and then press ENTER to finish.
The bill of materials is updated with the new information.

Move a BOM with a red grip point


1. Click the BOM you want to move.
A red grip point  displays.
2. Click the red grip point , and drag it into the drawing to view.
The bill of materials moves.

Edit an existing BOM

1. Click the BOM you want to change.
Pink grip points  display where detail can be pulled from the bill of materials.
2. Click a pink grip point , and drag it into the drawing to view.
The section of the bill of materials displays.

Update Detail BOM

 Fabricator console: **Details > Update Detail BOM** 

 Command line: **DTBOMUPD**


Updates a bill of material for the detail drawing.

Generate Detail Tag

 Fabricator console: **Details > Generate Detail Tag** 

 Command line: **GDTAG**

Generates tags for the items that you select.

1. Click a paper space tab.
2. Generate a detail. For more information, see *Generate Detail* (on page 95).
3. Click **Generate Detail Tag** .
4. Select the detail.
5. Select the point to draw the tag.
6. Select the point for the tag.

The tag displays in the drawing.

Tag setting

Allows you to edit generated tag settings.


- To change the tag settings, select the tag in the drawing. The **Tag setting** detail displays in the **Support Detail** tab.

Tag setting - Lists the tag settings of the detail.


- **Layer** - Allows you to select the layer from the list. Default is **0**.
- **Balloon Type** - Allows you to select the balloon type from the list. The default is **Circle**.
- **Balloon Color** - Allows you to select the balloon color from the list. The default is **ByLayer**.
- **Balloon Linetype** - Allows you to select the balloon linetype from the list. The default is **ByLayer**.
- **Balloon Size** - Allows you to type the balloon size. The default is **1.5000**.
- **Text Style** - Allows you to select the text style from the list. The default is **Standard**.
- **Text Color** - Allows you to select the text color from the list. The default is **ByLayer**.
- **Text Height** - Allows you to type the text height. The default is **0.5000**.

Generate Automatic Detail Tag

 Fabricator console: **Details > Generate Automatic Detail Tag** 

 Command line: **GADTAG**


Automatically creates tags.

1. Click a paper space tab.
2. Generate a detail. For more information, see *Generate Detail* (on page 95).
3. Click **Generate Automatic Detail Tag** .
4. Select the object to draw the tag.


The tag displays in the drawing.

Add Linear Dimensions

 Fabricator console: **Details > Add Linear Dimensions** 

 Command line: **LINEARDIM**

Draws linear dimensions for detail.

1. Click a paper space tab.
2. Click **Add Linear Dimensions** .
3. Select the object to append the dimension.
4. Specify the first extension.
5. Specify the second extension.

The dimension lines appear enabling you to adjust them.

6. Manipulate the dimension, and then click to finish.


The linear dimension displays in the drawing.

Add Aligned Dimension

 Fabricator console: **Details > Add Aligned Dimension** 

 Command line: **ALIGNEDDIM**

Draws aligned dimensions for detail.

1. Click a paper space tab.
2. Click **Add Aligned Dimension** .
3. Select the object to append the dimension.
4. Specify the first extension.
5. Specify the second extension.


The dimension lines appear enabling you to adjust them.

6. Manipulate the dimension, and then click to finish.


The linear dimension displays in the drawing.

Add Radius Dimension



 Fabricator console: **Details > Add Radius Dimension** 

 Command line: **RADIUSDIM**

Draws radial dimensions for detail.


1. Click a paper space tab.
2. Click **Add Radius Dimension** .
3. Select the arc or circle.
The dimension line appears enabling you to adjust it.
4. Manipulate the dimension, click in the drawing to place, and then press ENTER to finish.
The radius dimension displays in the drawing.

Add Jogged Dimension

 Fabricator console: **Details > Add Jogged Dimension** 

 Command line: **JOGGEDDIM**

Draws jogged dimensions for detail.

1. Click a paper space tab.
2. Click **Add Jogged Dimension** .
3. Select the arc or circle.
4. Specify the center location for the override.
5. Specify the dimension line location.
The dimension line appears enabling you to adjust it.
6. Manipulate the dimension, and then click to finish.
The jogged dimension displays in the drawing.


Add Angle Dimension

 Fabricator console: **Details > Add Angle Dimension** 


 Command line: **ANGLEDIM**

Draws an angle dimension for detail.

Draw an angle dimension with Arc or Circle

1. Click a paper space tab.
2. Click **Add Angle Dimension** .
3. Select an arc or a circle for dimension.
4. Specify the dimension line location.
5. Manipulate the dimension, click where you want the dimension placed, and then press ENTER to finish.
The dimension is displays in the drawing.

Draw an angle dimension with Line

1. Click a paper space tab.
2. Click **Add Angle Dimension** .
3. Select a line for angle dimension.
4. Select a second line.
The dimension lines appear enabling you to adjust them.
5. Manipulate the dimension, click where you want the dimension placed, and then press ENTER to finish.

The dimension displays in the drawing.



Change Tube To Rod

 Fabricator console: **Details > Change Tube To Rod** 

 Command line: **CTTR**

Changes the tube in a tubesheet to a rod. See *Change Rod To Tube* (on page 108) to do the opposite.

Change a tube in a tubesheet

1. Generate a tubesheet. For more information, see *Tubesheet Configuration* (on page 83).
2. Click **Change Tube To Rod** .
3. Select a tube in the tubesheet. Tubes are represented by this symbol: 

The tube changes to a rod and displays on both the baffle and the tubesheet.

Change Rod To Tube



 Fabricator console: **Details > Change Rod To Tube** 

 Command line: **CRTT**

Changes the rod in a tubesheet to a tube. See *Change Tube To Rod* (on page 108) to do the opposite.

Change the rod in a tubesheet



NOTE There must be a rod in a tubesheet to change it to a tube.

1. Generate a tubesheet. For more information, see *Tubesheet Configuration* (on page 83).
2. Click **Change Rod To Tube** .
3. Select a rod in the tubesheet. Rods are represented by this symbol: 



The rod changes to a tube.


Change the rod in a baffle

NOTE There must be a rod in a baffle to change it to a tube.


1. Click **Change Rod To Tube** .
2. Select a rod in the baffle. Rods are represented by this symbol: 
The rod changes to a tube.

Add Rod

 Fabricator console: **Details > Add Rod** 

 Command line: **ADDROD**

Adds a rod to the tubesheet.

1. Generate a tubesheet. For more information, see *Tubesheet Configuration* (on page 83).
2. Click **Add Rod** .
3. Select a tubesheet.


The rod represented by this symbol  appears in the drawing.

4. Select an insertion point.
The rod is added to the tubesheet.

NOTE You can move an added rod with a yellow grip point.




Remove Rod

 Fabricator console: **Details > Remove Rod** 



 Command line: **RMROD**


Removes the rod from a tubesheet.

NOTE There must be a rod in a tubesheet to remove it.


1. Generate a tubesheet. For more information, see *Tubesheet Configuration* (on page 83).
2. Click **Remove Rod** .
3. Select a rod to remove from the tubesheet. Rods are represented by this symbol  on a tubesheet, and this symbol  on a baffle.
The rod is removed from the tubesheet.

Make Tubesheet Groove

 Fabricator console: **Details > Make Tubesheet Groove** 



 Command line: **MTSGR**

Adds a groove to the partition.

1. Generate a tubesheet. For more information, see *Tubesheet Configuration* (on page 83).
2. Click **Make Tubesheet Groove** . Alternatively, type **MTSGR** on the command line, and press ENTER.
3. Select the tubesheet to add the groove.
4. Pick a point for the first groove path. This must be in a vertical or horizontal direction; diagonal does not work.
5. Pick the last point.


The partition changes color, and the elevation view shows the groove that was added.

Remove Tubesheet Groove

 Fabricator console: **Details > Remove Tubesheet Groove** 

 Command line: **RTSGR**

Adds a groove to the partition.

1. Generate a tubesheet. For more information, see *Tubesheet Configuration* (on page 83).
2. Click **Remove Tubesheet Groove** . Alternatively, type **RTSGR** on the command line, and press ENTER.
3. Select a groove to remove from the tubesheet.

The groove is removed from the tubesheet.

Remove Tube Hole

 Fabricator console: **Details > Remove Tube Hole** 

 Command line: **RMTSHOLE**

Removes a tube hole on a tubesheet.


NOTE If a baffle is placed in the drawing, then the tube hole is removed from the corresponding baffle. You cannot select the baffle tube holes to remove the tube hole. You may only select the tubesheet.

Single - Enables you to remove a single tube hole in the tubesheet.


Row - Enables you to remove a row of tubes in the tubesheet. The row is only removed within the partition area of the tubesheet.

All tube holes of the partition - Enables you to remove all the tube holes in a selected partition on the tubesheet.


Remove a single tube hole

1. Generate a tubesheet. For more information, see *Tubesheet Configuration* (on page 83).
2. Click **Remove Tube Hole** . Alternatively, type **RMTSHOLE** on the command line, and press ENTER.
3. Select **Single**, or type **S** on the command line.
4. Select a tube hole to remove from the tubesheet.
The tube hole removes from the tubesheet.
5. Repeat as needed, and then press ESC or ENTER to finish.

Remove a row of tube holes

1. Generate a tubesheet. For more information, see *Tubesheet Configuration* (on page 83).
2. Click **Remove Tube Hole** . Alternatively, type **RMTSHOLE** on the command line, and press ENTER.
3. Select **Row**, or type **R** on the command line.
4. Select a tube hole on the row you want to remove from the tubesheet.
The tube hole row removes from the tubesheet.
5. Repeat as needed, and then press ESC or ENTER to finish.

Remove all the tube holes in a partition

1. Generate a tubesheet. For more information, see *Tubesheet Configuration* (on page 83).
2. Click **Remove Tube Hole** . Alternatively, type **RMTSHOLE** on the command line, and press ENTER.
3. Select **All tube holes of the partition**, or type **A** on the command line.
4. Select a tube hole of a partition to remove all the tube holes in that partition from the tubesheet.
All the tube holes in that partition are removed from the tubesheet.
5. Repeat as needed, and then press ESC or ENTER to finish.

Restore Tube Hole

 Fabricator console: **Details > Restore Tube Hole** 

 Command line: **RSTSHOLE**

Restores a tube hole on a tubesheet.

NOTE If a baffle is placed in the drawing, then the tube hole is restored for the corresponding baffle. You cannot select the baffle tube holes to restore the tube hole. You may only select the tubesheet.


Single - Enables you to restore a single tube hole in the tubesheet.

Row - Enables you to restore a row of tubes in the tubesheet. The row is only removed within the partition area of the tubesheet.

All tube holes of the partition - Enables you to restore all the tube holes in a selected partition on the tubesheet.

Restore a single tube hole

A tubesheet should be generated and tube holes already removed before executing this command.

1. Click **Restore Tube Hole** . Alternatively, type **RSTSHOLE** on the command line, and press ENTER.
2. Select a tubesheet with removed holes.


The missing tube holes highlight.

NOTE The missing tube holes are represented as dotted lines. Setting the **Hidden Color** in the **2D Outline** to the same color as the **Main Color** can result in a confusing display. For optimal viewing, set the **Hidden Color** and the **Main Color** to different colors.

3. Select **Single**, or type **S** on the command line.
4. Select a highlighted tube hole to restore to the tubesheet.
The tube hole restores to the tubesheet.
5. Repeat as needed, and then press ENTER to finish.

Restore a row of tube holes

A tubesheet should be generated and tube holes already removed before executing this command.


1. Click **Restore Tube Hole** . Alternatively, type **RSTSHOLE** on the command line, and press ENTER.
2. Select a tubesheet with removed holes.
The missing tube holes highlight.

NOTE The missing tube holes are represented as dotted lines. Setting the **Hidden Color** in the **2D Outline** to the same color as the **Main Color** can result in a confusing display. For optimal viewing, set the **Hidden Color** and the **Main Color** to different colors.

3. Select **Row**, or type **R** on the command line.
4. Select a highlighted tube hole on the row you want to restore to the tubesheet.
The tube hole row restores to the tubesheet.
5. Repeat as needed, and then press ENTER to finish.

Restore all the tube holes in a partition

A tubesheet should be generated and tube holes already removed before executing this command.

1. Click **Restore Tube Hole** . Alternatively, type **RSTSHOLE** on the command line, and press ENTER.
2. Select a tubesheet with removed holes.

The missing tube holes highlight.

NOTE The missing tube holes are represented as dotted lines. Setting the **Hidden Color** in the **2D Outline** to the same color as the **Main Color** can result in a confusing display. For optimal viewing, set the **Hidden Color** and the **Main Color** to different colors.

3. Select **All tube holes of the partition**, or type **A** on the command line.
4. Select a highlighted tube hole of a partition to restore all the tube holes in that partition to the tubesheet.
All the tube holes in that partition are restored to the tubesheet.
5. Repeat as needed, and then press ENTER to finish.

Reset Settings

 Fabricator console: **Details > Reset Settings** 

 Command line: **RESDTSET**

Resets settings of the detail and the detail BOM.

Outline - Resets the outline for the detail or the detail BOM.

Balloon Tag - Resets the balloon settings of all detail tags.


Tag content - Resets the text of the detail tags.

Mtext - Resets the text settings for the detail.

Section - Resets the section settings for the detail.

Data - Resets all data on the detail BOM.

All - Resets all the aspects of the tag or the detail BOM.


1. Click a paper space tab.
2. Click **Reset Settings** .
3. Select the support detail with tags or the tag to reset.
4. Select an option from the list: **Outline**, **Tag**, **Mtext**, **Section**, **BOM**, or **All**.

The tag resets.

NOTE This command resets your selection to the highest level. The settings can be changed at the part level (double clicking the part itself and making changes), the detail level (selecting the part and making changes to the **Tag setting**), and they can be changed in the *Common Settings* (on page 81). Wherever these changes are made, when you use the **RESDTSET** command, the change updates to the higher level. So from part level to detail level or from detail level to *Common Settings* (on page 81).

SECTION 12









Equipment Console












 PVFabricator menu: **Equipment Palette**

 Command line: **CADWORXEQUIPMENT** (also **EM**)

Displays the **Equipment** console.

The icons at the top of the console represent components that you can add to a piece of equipment. When you start a new piece of equipment, the **Enter value** dialog box displays so that you can specify a name for that piece of equipment.

	Cylinder - Creates a cylinder. For more information, see <i>Cylinder</i> (on page 120).
	Elliptical Head - Creates an elliptical head. For more information, see <i>Elliptical Head</i> (on page 121).
	Torispherical Head - Creates a torispherical head. For more information, see <i>Torispherical Head</i> (on page 121).
	Spherical Head - Creates a spherical head. For more information, see <i>Spherical Head</i> (on page 122).
	Cone - Creates a cone. For more information, see <i>Cone</i> (on page 122).
	Welded Flat Head - Creates a welded flat head. For more information, see <i>Welded Flat Head</i> (on page 123).
	Body Flange - Creates a body flange. For more information, see <i>Body Flange</i> (on page 123).
	Skirt - Creates a skirt. For more information, see <i>Skirt</i> (on page 124).
	Box - Creates a box. For more information, see <i>Box</i> (on page 125).
	Stiffening Ring - Adds a stiffening ring to a parent component. For more information, see <i>Stiffening Ring</i> (on page 126).
	Nozzle - Adds a nozzle to a parent component. For more information, see <i>Nozzle</i> (on page 127).
	Platform - Adds a platform to a parent component. For more information, see <i>Platform</i> (on page 131).
	Saddle - Adds a saddle to a parent component. For more information, see <i>Saddle</i> (on page 132).

	Lugs - Adds lugs to a parent component. For more information, see <i>Lugs</i> (on page 133).
	Legs - Adds legs to a parent component. For more information, see <i>Legs</i> (on page 134).
	User Shape - Adds a user-created shape to a parent component. For more information, see <i>User Shape</i> (on page 135)
	Impeller - Creates an impeller. For more information, see <i>Impeller</i> (on page 136). This component is not available for PVFabricator.
	Shaft - Creates a shaft. For more information, see <i>Shaft</i> (on page 136). This component is not available for PVFabricator.
	Coupling - Creates a coupling. For more information, see <i>Coupling</i> (on page 136). This component is not available for PVFabricator.
	Motor - Creates a motor. For more information, see <i>Motor</i> (on page 136). This component is not available for PVFabricator.
	Box - Creates a box. For more information, see <i>Box</i> (on page 125). This component is not available for PVFabricator.
	Pump Base - Creates a pump base. For more information, see <i>Pump Base</i> (on page 137). This component is not available for PVFabricator.
	Pump Nozzle - Creates a pump nozzle. For more information, see <i>Pump Nozzle</i> (on page 138). This component is not available for PVFabricator.
	Setup - Displays the PVFabricator Setup dialog box. For more information, see <i>PVFabricator Setup</i> (on page 25).
	Help - Displays the PVFabricator User's Guide help file.

User Shapes

Data file - Specifies the file name of the user shape drawing and user-created data file to connect to a vessel parent component or a vessel nozzle child component. The list contains all the equipment objects present in the current drawing.

Size - Specifies the size associated with the user shape.

Insert - Displays the **Enter value** dialog box so that you can specify a name for the piece of equipment and then places the user shape in the model.

PVFabricator Name - Specifies the name of the active equipment component.

Placement Tab (Equipment Console) (on page 119)

Components Tab (Equipment Console) (on page 120)

Enter value Dialog Box


Specifies the name for a new piece of equipment.

Enter equipment name - Specifies the equipment name.

What do you want to do?

- *Add a nozzle to an existing nozzle* (on page 117)
 - *Add a user shape to the model* (on page 117)
 - *Use a nozzle as a base component for other components* (on page 117)
 - *Import a model from PVElite* (on page 118)
 - *Export the model to PVElite* (on page 118)
-

Add a nozzle to an existing nozzle


1. Select the base nozzle on the **Components** tab of the **PVFabricator** dialog box.
2. Click **Nozzle**  on the **PVFabricator** dialog box to add the new nozzle.

NOTE Using a nozzle as a parent component is only valid for vessel components.

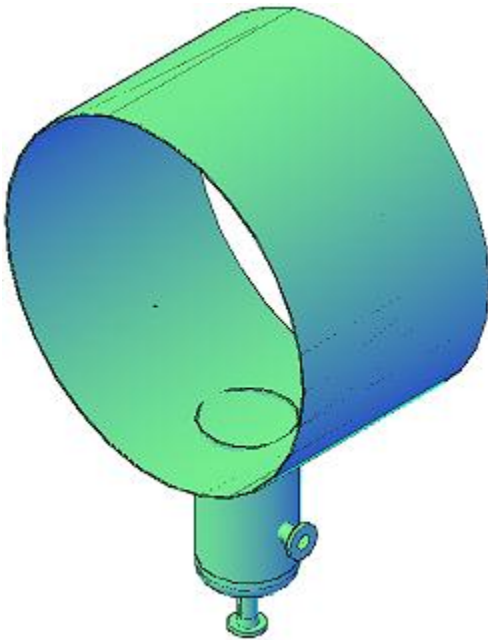
Add a user shape to the model

1. Select the parent vessel or nozzle from the **Components** tab on the **PVFabricator** dialog box.
2. Select the data file from the **Data file** list.
3. Select the size from the **Size** list.
4. Click **Insert**.

Use a nozzle as a base component for other components

1. Select the nozzle on the **Components** tab on the **<AIT_DELETE_END> PVFabricator** dialog box.
2. Select the **Nozzle details** box.
3. Click  to display the **Nozzle details** dialog box.
4. Select **N/A** in the **Flange type** box.
5. Select **N/A** in the **Flange rating** box.
6. Select the base nozzle on the **Components** tab on the **<AIT_DELETE_END> PVFabricator** dialog box.

- Click **Elliptical Head**  on the **<AIT_DELETE_END> PVFabricator** dialog box to add the elliptical head.



Import a model from PVElite

- Click the **Placement** tab on the **Equipment** console.
- Click **Import**.
- Select the PVElite file to import.

*The **Select PVElite File** dialog box displays.*

The model imports.

NOTE You must immediately export the model after you import it to maintain the relationships between the components in the PVElite and PV Fabricator models. This is only required the first time that you import a model from PVElite.

Export the model to PVElite

- Click the **Placement** tab on the **Equipment** console.
- Click **Export**.
- Specify the file name and location for the file.

*The **Save As** dialog box displays.*

Placement Tab (Equipment Console)

Specifies parameters for placing equipment.

New - Displays the **Enter value** dialog box. This option creates a new piece of equipment and adds it to the **Equipment Name** list.

The starting location of the equipment defaults to 0, 0, 0 and is marked by the X attribute. The X attribute is a block (xatb.DWG) and is located in the *[Product Folder]\PVFabricator\Support* folder. The X attribute is required and must not be removed.

Rename - Displays the **Enter value** dialog box. This option changes the name of the equipment displayed in the **Equipment Name** box.

Delete - Removes the equipment displayed in the **Equipment Name** box. This option erases any components associated with the equipment and removes it from the **Equipment Name** list. A confirmation message displays requiring you to confirm the deletion.

Location

Specify On-screen - Click the location in the model view to specify the starting point for the equipment.

X - Specifies the X-location for the equipment starting point. This value defaults to zero.

Y - Specifies the Y-location for the equipment starting point. This value defaults to zero.

Z - Specifies the Z-location for the equipment starting point. This value defaults to zero.

Rotation

Specify On-screen - Click the location in the model view to indicate the rotation of the equipment.

Angle - Specifies the degree of rotation about the world Z-axis for the equipment. This value defaults to zero.

Orientation - Specifies the orientation of the equipment. By default, all equipment is vertically oriented.

NOTE Platforms and legs can be drawn only when the equipment is vertical. Saddles can be drawn only when the equipment is horizontal. If you draw these components, you cannot change the orientation.

Options

Hollow - Creates hollow equipment components. This option can significantly slow down the performance of the model based on the number of components in each equipment.

Reference Line

Enable - Specifies a reference line that can then be used to specify certain components such as nozzles, saddles, platforms, and so on. The option is saved in the configuration file. However, each reference value is stored with each equipment.

Value - Specifies the location for the reference line. For vertical equipment, the value lies along the Z-axis. For horizontal equipment, the value lies along the X-axis.

PVElite

Import - Imports the current equipment from PVElite. The PVElite information is stored in the drawing.

Export - Exports the current equipment to PVElite. PVElite does not support apex nozzles on a cone component. Instead, the apex nozzle on a cone must be modeled using a cylinder and a body flange.

Components Tab (Equipment Console)

Displays equipment components as they are added to the equipment. The AutoCAD **UNITS** setting controls the display of most numeric data in the grid.

A right-click menu is available for **Cut**, **Copy**, **Paste**, and **Delete**.

Move options

Grid - Indicates that movement is based on the grid.

Model - Indicates that movement is based on the model.

Move Up - Moves the component up in the list or model.

Move Down - Moves the component down in the list or model.

Cylinder

 Toolbar: 

Specifies the parameters of the cylinder.

Description - Specifies an alphanumeric description for the component. This description is visible even when the component is collapsed in the grid.

Length - Specifies the distance from the starting point to the ending point of the component.

Diameter Basis - Specifies either the inside or outside diameter. Select the diameter type from the list.

Diameter - Specifies the diameter of the component based on the **Diameter Basis** value.

Thickness - Specifies the component thickness.

Material - Displays the **Select Item** dialog box, which lets you select the material from the list. For more information, see *Select Item Dialog Box* (on page 139). The **Select Item** dialog box uses the file specified by the **MaterialFile** configuration file variable. For more information, see *MaterialFile* (on page 32).

Insulation - Displays the **Insulation** dialog box. For more information, see *Insulation Dialog Box* (on page 140).

Elliptical Head



Specifies the parameters of the elliptical head.

Description - Specifies an alphanumeric description for the component. This description is visible even when the component is collapsed in the grid.

Length - Specifies the distance from the starting point to the ending point of the component.

Diameter Basis - Specifies either the inside or outside diameter. Select the diameter type from the list.

Diameter - Specifies the diameter of the component based on the **Diameter Basis** value.

Thickness - Specifies the component thickness.

Flip Orientation - Reverses the orientation of the component.

Material - Displays the **Select Item** dialog box, which lets you select the material from the list. For more information, see *Select Item Dialog Box* (on page 139). The **Select Item** dialog box uses the file specified by the **MaterialFile** configuration file variable. For more information, see *MaterialFile* (on page 32).

Head Factor - Specifies the head factor of the elliptical head.

Insulation - Displays the **Insulation** dialog box. For more information, see *Insulation Dialog Box* (on page 140).

Torispherical Head



Specifies the parameters of the torispherical head.

Description - Specifies an alphanumeric description for the component. This description is visible even when the component is collapsed in the grid.

Length - Specifies the distance from the starting point to the ending point of the component.

Diameter Basis - Specifies either the inside or outside diameter. Select the diameter type from the list.

Diameter - Specifies the diameter of the component based on the **Diameter Basis** value.

Thickness - Specifies the component thickness.

Flip Orientation - Reverses the orientation of the component.

Material - Displays the **Select Item** dialog box, which lets you select the material from the list. For more information, see *Select Item Dialog Box* (on page 139). The **Select Item** dialog box uses the file specified by the **MaterialFile** configuration file variable. For more information, see *MaterialFile* (on page 32).

Crown Radius - Specifies the crown radius for the torispherical head.

Knuckle Radius - Specifies the knuckle radius for the torispherical head.

Miscellaneous - Displays the **Insulation** dialog box. For more information, see *Insulation Dialog Box* (on page 140).

Spherical Head



Specifies the parameters of the spherical head.

Description - Specifies an alphanumeric description for the component. This description is visible even when the component is collapsed in the grid.

Length - Specifies the distance from the starting point to the ending point of the component.

Diameter Basis - Specifies either the inside or outside diameter. Select the diameter type from the list.

Diameter - Specifies the diameter of the component based on the **Diameter Basis** value.

Thickness - Specifies the component thickness.

Flip Orientation - Reverses the orientation of the component.

Material - Displays the **Select Item** dialog box, which lets you select the material from the list. For more information, see *Select Item Dialog Box* (on page 139). The **Select Item** dialog box uses the file specified by the **MaterialFile** configuration file variable. For more information, see *MaterialFile* (on page 32).

Insulation - Displays the **Insulation** dialog box. For more information, see *Insulation Dialog Box* (on page 140).

Cone



Specifies the parameters of the cone.

Description - Specifies an alphanumeric description for the component. This description is visible even when the component is collapsed in the grid.

Length - Specifies the distance from the starting point to the ending point of the component.

Diameter Basis - Specifies either the inside or outside diameter. Select the diameter type from the list.

Bottom Diameter (Vertical Cone) or From Diameter (Horizontal Cone) - Specifies the diameter of the component based on the **Diameter Basis** value.

Thickness - Specifies the component thickness.

Material - Displays the **Select Item** dialog box, which lets you select the material from the list. For more information, see *Select Item Dialog Box* (on page 139). The **Select Item** dialog box uses the file specified by the **MaterialFile** configuration file variable. For more information, see *MaterialFile* (on page 32).

Top Diameter (Vertical Cone) or To Diameter (Horizontal Cone) - Specifies the cone end diameter.

Concentric - Specifies whether the cone is concentric.

Shell Section - Specifies whether the cone is a shell section.

Cone Length - Specifies the length of the conical portion of the component. This box is only available only if the cone is not a shell section.

Rotation Angle - Specifies the rotation angle of the cone. This box is only available if the cone is eccentric.

Insulation - Displays the **Insulation** dialog box. For more information, see *Insulation Dialog Box* (on page 140).

Welded Flat Head



Specifies the parameters of the welded flat head.

Description - Specifies an alphanumeric description for the component. This description is visible even when the component is collapsed in the grid.

Length - Specifies the distance from the starting point to the ending point of the component.

Diameter Basis - Specifies either the inside or outside diameter. Select the diameter type from the list.

Diameter - Specifies the diameter of the component based on the **Diameter Basis** value.

Thickness - Specifies the component thickness.

Material - Displays the **Select Item** dialog box, which lets you select the material from the list. For more information, see *Select Item Dialog Box* (on page 139). The **Select Item** dialog box uses the file specified by the **MaterialFile** configuration file variable. For more information, see *MaterialFile* (on page 32).

Insulation - Displays the **Insulation** dialog box. For more information, see *Insulation Dialog Box* (on page 140).

Body Flange



Specifies the parameters of the body flange.

ANSI flange lookup - Displays the **ANSI Flange** dialog box.

Description - Specifies an alphanumeric description for the component. This description is visible even when the component is collapsed in the grid.

Length - Specifies the distance from the starting point to the ending point of the component.

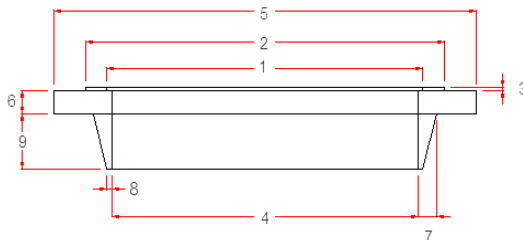
Diameter Basis - Specifies either the inside or outside diameter. Select the diameter type from the list.

Diameter - Specifies the diameter of the component based on the **Diameter Basis** value.

Thickness - Specifies the component thickness.

Flip Orientation - Reverses the orientation of the component.

Material - Displays the **Select Item** dialog box, which lets you select the material from the list. For more information, see *Select Item Dialog Box* (on page 139). The **Select Item** dialog box uses the file specified by the **MaterialFile** configuration file variable. For more information, see *MaterialFile* (on page 32).



Flange Face ID - Specifies the component inner diameter. See dimension 1.

Flange Face OD - Specifies the component outer diameter. See dimension 2.

Flange Face Thickness - Specifies the component thickness. See dimension 3.

Gasket ID - Specifies the component inner diameter.

Gasket OD - Specifies the component outer diameter.

Gasket Thickness - Specifies the component thickness.

Flange ID - Specifies the component interior flange diameter. If you set this value to zero for a blind flange, the software disables all hub-related boxes. The hub-related boxes are enabled if the **Flange ID (D)** box has a value greater than zero. See dimension 4

Flange OD - Specifies the component outer diameter. See dimension 5

Flange Thickness - Specifies the component thickness. See dimension 6

Large Hub Thickness - Specifies the component thickness. See dimension 7.

Small Hub Thickness - Specifies the component thickness. See dimension 8

Hub Length - Specifies the distance from the starting point to the ending point of the component. See dimension 9.

Insulation - Displays the **Insulation** dialog box. For more information, see *Insulation Dialog Box* (on page 140).

ANSI Flange Dialog Box

Specifies parameters for ANSI flanges. This dialog box lets you select an ANSI flange and then populates the body flange dimension boxes automatically. The flange data is read from the *[Product Folder]/Equipment/System/BodyFlangeImperial.DAT* file.

Flange Series - Specifies the series associated with the flange. Select a series from the list.

Flange type - Specifies the type associated with the flange. Select a type from the list.

Flange rating - Specifies the rating associated with the flange. Select a rating from the list.

Nominal Diameter - Specifies the nominal diameter associated with the flange. Select a diameter from the list.

Bore Diameter - Specifies the bore diameter associated with the flange.

Flange Blind - Specifies whether the flange is a blind flange.

Skirt

 Toolbar: 

Specifies the parameters of the skirt.

Description - Specifies an alphanumeric description for the component. This description is visible even when the component is collapsed in the grid.

Length - Specifies the distance from the starting point to the ending point of the component.

Diameter Basis - Specifies either the inside or outside diameter. Select the diameter type from the list.

Diameter - Specifies the diameter of the component based on the **Diameter Basis** value.

Thickness - Specifies the component thickness.

Flip Orientation - Reverses the orientation of the component.

Material - Displays the **Select Item** dialog box, which lets you select the material from the list. For more information, see *Select Item Dialog Box* (on page 139). The **Select Item** dialog box uses the file specified by the **MaterialFile** configuration file variable. For more information, see *MaterialFile* (on page 32).

Basering Type - Specifies the type of basering to attach to the skirt. Available options are:

- **None**
- **Simple - No Gussets**
- **With Gussets**
- **With Top Plates**
- **Continuous Top Plate**

Base Diameter - Specifies the skirt base diameter.

Basering ID - Specifies the component inner diameter.

Basering OD - Specifies the component outer diameter.

Bottom Plate Thickness - Specifies the component thickness.

Top Plate Radial Width - Specifies the component width.

Top Plate Thickness - Specifies the component thickness.

Top Plate Width - Specifies the component width.

Gusset Height - Specifies the component height.

Gusset Width - Specifies the skirt gusset width.

Distance Between Gussets - Specifies the distance between gussets on the skirt.

Gusset Thickness - Specifies the component thickness.

Number Of Bolts - Specifies the number of bolts on the skirt.

Insulation - Displays the **Insulation** dialog box. For more information, see *Insulation Dialog Box* (on page 140).

Box

 Toolbar: 

Specifies the parameters of the box.

Description - Specifies an alphanumeric description for the component. This description is visible even when the component is collapsed in the grid.

Dimension basis - Specifies either the inside or outside dimension. Select the dimension type from the list.

Length - Specifies the distance from the starting point to the ending point of the component.

Width - Specifies the component width.

Height - Specifies the component height.

Thickness - Specifies the component thickness.

Start offset X - Specifies an offset distance along the box X-axis. This value is applied at the start point of the box.

Start offset Y - Specifies an offset distance along the box Y-axis. This value is applied at the start point of the box.

End offset X - Specifies an offset distance along the box Y-axis of the box. This value is applied at the end point of the box.

End offset Y - Specifies an offset distance along the box Y-axis. This value is applied at the end point of the box.

Rotation angle - Specifies a box rotation angle.

Insulation - Displays the **Insulation** dialog box. For more information, see *Insulation Dialog Box* (on page 140).

Stiffening Ring



Specifies the parameters for a stiffening ring. This component cannot be created by itself. It must be attached to one of the following components:

- *Cylinder* (on page 120)
- *Cone* (on page 122)

Description - Specifies an alphanumeric description for the component. This description is visible even when the component is collapsed in the grid.

Distance - Specifies the distance from the bottom of the component to which this component is attached.

Ring Type - Specifies the ring type. The available types are:

- **Bar Ring**
- **Section Ring**

Ring Location - Specifies whether the ring is placed on the outer or inner diameter side of the parent component. This option is only available when the **Ring Type** box is set to **Section Ring**.

Initial Flip - Flips the ring in the X-Y plane. The orientation reference of the ring is always based on the initial orientation default (0 degree angle-of-rotation). This option is only available when the **Ring Type** box is set to **Section Ring**.

Member rotation angle - Rotates the ring to preset rotation angles (0, 90, 180, or 270 degrees). This option is only available when the **Ring Type** box is set to **Section Ring**.

Inside Diameter - Specifies the component inner diameter. This option is only available if the **Ring Type** box is set to **Bar Ring**.

Outside Diameter - Specifies the component outer diameter. This option is only available if the **Ring Type** box is set to **Bar Ring**.

Thickness - Specifies the component thickness. This option is only available if the **Ring Type** box is set to **Bar Ring**.

Member - Displays the **Select Steel Member** dialog box so that you can specify the kind of section to use to build the ring. For more information, see *Select Steel Member Dialog Box* (on page 127).

Material - Displays the **Select Item** dialog box, which lets you select the material from the list. For more information, see *Select Item Dialog Box* (on page 139). The **Select Item** dialog box uses the file specified by the **MaterialFile** configuration file variable. For more information, see *MaterialFile* (on page 32).

Select Steel Member Dialog Box

Specifies parameters for steel members.

Select Type - Specifies the steel member type. The value in this box determines the options in the **Select Data file** box.

Select Data file - Specifies the steel member data file. The value in this box determines the options in the **Select Member** list.

Select Member - Displays the available members. The values in this list depend on the values in the **Select Type** and **Select Data file** boxes.

Pipe selection

The pipe selection boxes are only available if the **Select Type** box is set to **Pipe**.

Nominal/Actual options - Specifies whether you are using ANSI nominal/actual parameters or CADWorx Steel data file parameters.

NOTE The ANSI pipe data files, ScheduleMetric.DAT and ScheduleImperial.DAT, are located in the *[Product Folder]/PVFabricator/System* folder.

Pipe basis - Specifies whether the pipe is based on nominal or actual values.

Diameter - Specifies the pipe diameter. This option is only available if the **Pipe basis** is set to **Nominal**.

Schedule - Specifies the pipe schedule. This option is only available if the **Pipe basis** is set to **Nominal**.

Wall thickness - Specifies the pipe wall thickness.

Outside diameter - Specifies the pipe outer diameter.

Nozzle

 **Toolbar:** 

Specifies parameters for the nozzle.

This component cannot be created by itself. It must be attached to one of the following components:


- *Cylinder* (on page 120)
- *Elliptical Head* (on page 121)
- *Torispherical Head* (on page 121)
- *Spherical Head* (on page 122)
- *Cone* (on page 122)
- *Welded Flat Head* (on page 123)
- **Nozzle** - See *Add a nozzle to an existing nozzle* (on page 117) for more information.


Nozzles are recognized by the ISOGEN module of CADWorx Plant Professional when the piping connected to the nozzle is sent to ISOGEN.


Description - Specifies an alphanumeric description for the component. This description is visible even when the component is collapsed in the grid.

Nozzle type - Specifies the nozzle type. The available types are:

- **Standard** - Specifies an ANSI nozzle. Displays the **Nozzle details** dialog box.
- **FVC** - Specifies a Forge Vessel Connection (FVC) type nozzle. Displays the **FVC Nozzle Details** dialog box.

Distance - Specifies the distance from the bottom of the component to which this component is attached. Click  to display the **Nozzle details** dialog box.

Standard Nozzle Details - Displays the standard nozzle details. This box is only available if the **Nozzle type** box is set to **Standard**. Click  to display the **Nozzle Details** dialog box.

FVC Nozzle Details - Displays the FVC nozzle details. This box is only available if the **Nozzle type** box is set to **FVC**. Click  to display the **FVC Nozzle Details** dialog box.

Layout Angle - Specifies the nozzle layout angle.

Outside Projection - Specifies the nozzle outside projection. The nozzle outside projection is measure from the OD surface of the parent to the flange face or the outer most point along the nozzle center line.

Inside Projection - Specifies the nozzle inside projection.

Nozzle Placement Dialog Box

Specifies advanced options for nozzle placement. A figure on the dialog box displays the values. This figure changes based on the type of component to which the nozzle is attached.

(A) Distance - Specifies the distance from the bottom of the component to which this component is attached. When the **Nozzle direction** box (enabled only for a nozzle on a cone) is set to **Apex**, the **Distance** box becomes the radial distance of the nozzle from the centerline of the cone.

(B) Lateral angle - Specifies the nozzle lateral angle.

Rotation angle (Y) - Specifies the rotation angle about the Y-axis.

Rotation angle (X) - Specifies the rotation angle about the X-axis.

Rotation angle (Z) - Specifies the rotation angle about the Z-axis.

Centerline Distance - Specifies the component centerline distance.

(C) Offset distance - Specifies the offset distance from the center of the parent component.

(D) Center to face distance - Specifies the component face distance to the center of the parent component.

(E) Layout angle - Specifies the angle between the component centerline and the X-axis.

Nozzle direction - Specifies the component direction.

Nozzle Details Dialog Box

Specifies standard nozzle parameters.


The lists for **Diameter**, **Flange rating**, **Flange type**, and **Schedule** are populated based on the entries of the NozzleDropDown.TBL file located in the *[Product Folder]\Equipment\System* folder.

Nozzle basis - Specifies whether you are using nominal or actual parameters or CADWorx Steel data file parameters. When you select **Nominal**, Equipment provides a look-up system for flange dimensions and schedule information.

Diameter - Specifies the diameter of the component.

Schedule - Specifies the schedule rating for the nozzle.

Wall thickness - Specifies the component thickness.

Nozzle extension - Indicates whether a nozzle extension (elbow) is included. Click  to display the **Nozzle extension** dialog box.

Nozzle material - Displays the **Select Item** dialog box, which lets you select the material from the list. For more information, see *Select Item Dialog Box* (on page 139). The **Select Item** dialog box uses the file specified by the **MaterialFile** configuration file variable. For more information, see *MaterialFile* (on page 32).

Flange type - Specifies the type of nozzle flange. If you set this value to **N/A**, then you can manually specify the **Flange thickness** and **Flange OD** values. For **Flange type** values other than **N/A**, the software uses the dimensions from the FlangeImperial.DAT or FlangeMetric.DAT file. These files are located in the *[Product Folder]\Electrical\System* folder.


Flange rating - Specifies the nozzle flange rating.


Flange thickness - Specifies the component thickness.

Flange OD - Specifies the component outer diameter.

Flange material - Displays the **Select Item** dialog box, which lets you select the material from the list. For more information, see *Select Item Dialog Box* (on page 139). The **Select Item** dialog box uses the file specified by the **MaterialFile** configuration file variable. For more information, see *MaterialFile* (on page 32).

Thread/Engagement length - Specifies the nozzle end point location. This end point provides a snap location for any component attached to the nozzle.

Nozzle reinforcing pad - Indicates whether a reinforcing pad exists. Click  to display the **Nozzle Pad** dialog box.

Nozzle Insulation - Indicates whether insulation exists. Click  to display the **Insulation** dialog box.

FVC Nozzle Details Dialog Box

Specifies parameters for FVC nozzles. The modifiable FVC nozzle data, including the nozzle type, rating, and size drop down parameters, are listed in the *[Product Folder]\Equipment\System\FVCNozzleImperial.dat* file. Refer to the data file itself for instructions on modifying the list options selection and activation control.

FVC Table Lookup

Populates the nozzle dimension boxes from the FVC nozzle database based on the following parameters:

Type - Specifies the FVC nozzle type.

Flange rating - Specifies the flange rating.

Size - Specifies the nozzle size.

Option Diameter - Indicates whether **Stop/Relief Diameter** is set to **Nut Stop** diameter or **Nut Relief** diameter.

Option ASME codes - Specifies the Theta angle. Select **Div. 1** to indicate 30 degrees or **Div. 2** to indicate 45 degrees.

Actual - Indicates that the FVC nozzle parameters can be modified manually.

Flange OD - Specifies the component outer diameter.

Flange thickness - Specifies the component thickness. This value includes the flange face thickness.

Bore Diameter - Specifies the nozzle bore diameter.

Barrel OD - Specifies the component outer diameter.

Stop/Relief Diameter - Specifies the diameter to either the **Nut Stop** or **Nut Relief** diameter depending on the **Option Diameter** setting.


Relief Length - Specifies the distance from the starting point to the ending point of the component.

Neck Thickness - Specifies the component thickness.

Overall Length - Specifies the distance from the starting point to the ending point of the component.

Nozzle material - Displays the **Select Item** dialog box, which lets you select the material from the list. For more information, see *Select Item Dialog Box* (on page 139). The **Select Item** dialog box uses the file specified by the **MaterialFile** configuration file variable. For more information, see *MaterialFile* (on page 32).

Thread/Engagement length - Specifies the nozzle end point location. This end point provides a snap location for any component attached to the nozzle.

Nozzle Insulation - Indicates whether insulation exists. Click  to display the **Insulation** dialog box.

Nozzle extension Dialog Box

Controls the parameters for nozzle extensions.

Radius multiplier - Specifies the radius of the elbow. You can use any valid radius value such as **1.5** for a long radius or **1.0** for a short radius.

(A) Elbow angle - Specifies the angle of the elbow. You can use any valid angle between 5 and 90 degrees.

(B) Horizontal projection - Specifies the horizontal projection value.

(C) Straight run angle direction - Specifies the direction angle for the straight run. This angle is measured relative to the layout angle.

(D) Outside projection - Specifies the projection distance for the outside portion of the extension.

Nozzle Pad Dialog Box

Specifies parameters for the nozzle pad.

Pad width - Specifies the distance from the outer edge of nozzle pad to the nozzle outside diameter.

Pad thickness - Specifies the component thickness.

Pad material - Displays the **Select Item** dialog box, which lets you select the material from the list. For more information, see *Select Item Dialog Box* (on page 139). The **Select Item** dialog box uses the file specified by the **MaterialFile** configuration file variable. For more information, see *MaterialFile* (on page 32).

Platform

 Toolbar: 

Specifies parameters for the platform.

This component cannot be created by itself. It must be attached to one of the following components:

- *Cylinder* (on page 120)
- *Elliptical Head* (on page 121)
- *Torispherical Head* (on page 121)
- *Spherical Head* (on page 122)
- *Cone* (on page 122)

Description - Specifies an alphanumeric description for the component. This description is visible even when the component is collapsed in the grid.

Distance - Specifies the distance from the bottom of the component to which this component is attached. This component uses the *PlatformHandRailFile* configuration file variable to create the hand rails that are part of the platform.

Start Angle - Specifies the start angle of the platform.

End Angle - Specifies the end angle of the platform.

Width - Specifies the component width.

Height - Specifies the component height.

Clearance - Specifies the clearance from the component to which this platform attaches.

Ladder - Specifies whether a ladder is drawn with the platform. If a ladder is required, this component uses the *PlatformLadderFile* (on page 32) configuration file variable to create the ladder.

Ladder Start Elevation - Specifies the starting elevation (world) of the ladder.

Ladder End Elevation - Specifies the ending elevation (world) of the ladder.

Ladder Cage - Specifies whether the ladder is caged. Specific parameters of the cage are controlled by the *PlatformLadderFile* (on page 32) configuration file variable.

Ladder Layout Angle - Specifies the layout angle for the ladder.

Ladder Vessel Clearance - Specifies the clearance between the ladder and the vessel.

Ladder Platform Clearance - Specifies the clearance between the ladder and the platform.

Ladder Alignment - Specifies the ladder alignment. The available options are:

- **Platform**
- **Vessel**

Material - Displays the **Select Item** dialog box, which lets you select the material from the list. For more information, see *Select Item Dialog Box* (on page 139). The **Select Item** dialog box uses the file specified by the **MaterialFile** configuration file variable. For more information, see *MaterialFile* (on page 32).

Saddle

 Toolbar: 

Specifies the parameters for the saddle.

This component cannot be created by itself. It must be attached to the following component:

- *Cylinder* (on page 120)

Description - Specifies an alphanumeric description for the component. This description is visible even when the component is collapsed in the grid.

Distance - Specifies the distance from the bottom of the component to which this component is attached.

Width - Specifies the component width.

Height - Specifies the component height.

Contact Angle - Specifies the saddle contact angle.

Plate Thickness - Specifies the component thickness.

Wear Pad Width - Specifies the component width.

Wear Pad Thickness - Specifies the component thickness.

Wear Pad Contact Angle - Specifies the saddle wear pad contact angle.

Base Plate Length - Specifies the distance from the starting point to the ending point of the component.

Base Plate Width - Specifies the component width.

Base Plate Thickness - Specifies the component thickness.

Number of Ribs - Specifies the number of ribs.

Rib Thickness - Specifies the component thickness.

Web Thickness - Specifies the component thickness.

Web Location - Specifies the web location.

Bolts - Displays the **Bolt Details** dialog box. For more information, see *Bolt Details Dialog Box* (on page 140).

Material - Displays the **Select Item** dialog box, which lets you select the material from the list. For more information, see *Select Item Dialog Box* (on page 139). The **Select Item** dialog box uses the file specified by the **MaterialFile** configuration file variable. For more information, see *MaterialFile* (on page 32).

Layout angle - Specifies the angle of the saddle from the equipment.

Fixed saddle - Specifies the saddle is fixed or not.

Lugs

 Toolbar: 

Specifies the parameters for the lug.

This component cannot be created by itself. It must be attached to one of the following components:

- *Cylinder* (on page 120)
- *Elliptical Head* (on page 121)
- *Torispherical Head* (on page 121)
- *Spherical Head* (on page 122)
- *Cone* (on page 122)
- *Welded Flat Head* (on page 123)

Description - Specifies an alphanumeric description for the component. This description is visible even when the component is collapsed in the grid.

Distance - Specifies the distance from the bottom of the component to which this component is attached.

Lug Type - Specifies the lug type. The available options are:

- Simple
- With Top Plate
- With Top Plate Ring

Number of Lugs - Specifies the number of lugs.

Distance Between Gussets - Specifies the distance between the lug gussets.

Gusset Height - Specifies the component height.

Gusset Width - Specifies the component width.

Gusset Thickness - Specifies the component thickness.

Bottom Plate Radial Width - Specifies the component width.

Bottom Plate Length - Specifies the distance from the starting point to the ending point of the component.

Bottom Plate Thickness - Specifies the component thickness.

Top Plate Radial Width - Specifies the component width.

Top Plate Length - Specifies the distance from the starting point to the ending point of the component.

Top Plate Thickness - Specifies the component thickness.

Material - Displays the **Select Item** dialog box, which lets you select the material from the list. For more information, see *Select Item Dialog Box* (on page 139). The **Select Item** dialog box uses the file specified by the **MaterialFile** configuration file variable. For more information, see *MaterialFile* (on page 32).

Legs



Specifies the parameters for the leg.

This component cannot be created by itself and must be attached to one of the following components:

- *Cylinder* (on page 120)
- *Elliptical Head* (on page 121)
- *Torispherical Head* (on page 121)
- *Spherical Head* (on page 122)
- *Cone* (on page 122)

Description - Specifies an alphanumeric description for the component. This description is visible even when the component is collapsed in the grid.

Distance - Specifies the distance from the bottom of the component to which this component is attached.

Number of Legs - Specifies the number of legs.

Leg Length - Specifies the distance from the starting point to the ending point of the component.

Member Offset - Specifies the leg offset from the member.

Rotation Angle - Specifies the overall rotation angle applied to all the legs relative to the component to which the legs are attached.

Member Size - Displays the **Select Steel Member** dialog box so that you can select the section to use to build the legs. See *Select Steel Member Dialog Box* (on page 127) for more information.

Member Rotation Angle - Specifies the rotation angle applied to each member of the leg.

Baseplates - Indicates whether baseplates should be created.

Baseplate Offset - Specifies the baseplate offset.

Baseplate Length - Specifies the distance from the starting point to the ending point of the component.

Baseplate Width - Specifies the component width.

Baseplate Thickness - Specifies the component thickness.

Baseplate Rotation Angle - Specifies the baseplate rotation angle.

Material - Displays the **Select Item** dialog box, which lets you select the material from the list. For more information, see *Select Item Dialog Box* (on page 139). The **Select Item** dialog box uses the file specified by the **MaterialFile** configuration file variable. For more information, see *MaterialFile* (on page 32).

User Shape

Specifies the parameters for a user-created shape.

This component cannot be created by itself. It must be attached to a vessel component or a vessel nozzle component.

The user shape data file (.DAT) and its corresponding drawings must be stored in the folder specified in the configuration file.

The naming convention for user shape drawings is: *[datafilename] + [size].dwg*

where:

[datafilename] = Name of the user shape data file excluding the extension (.DAT).

[size] = Size corresponding to the data specified in the user shape data file.

For more information, refer to the davit user shape example included in the *[Product Folder]/PVFabricator/UserShape* folder.

All available data files located in the UserShape folder can be selected from the **Data file** list. After you have selected the data file, the available drawings can be selected from the **Size** list. Click **Insert** to attach the user shape drawing to the selected parent or nozzle component.

The location of the inserted user shape can be adjusted using the **User Shapes** dialog box.

Description - Specifies an alphanumeric description for the component. This description is visible even when the component is collapsed in the grid.

Distance - Specifies the distance from the bottom of the component to which this component is attached.

Layout Angle - Specifies the angle between the component centerline and the X-axis.

Additional user data entries listed in the user shape data file - The additional user data displayed in the grid can be modified manually through the grid or through the data file itself. You can modify the data file using any text editor. Value changes in the grid are saved in the drawing only, not the data file.

User Shapes Dialog Box

Specifies parameters for a user-defined shape.

(A) Distance - Specifies the distance from the bottom of the component to which this component is attached. When the **User Shapes direction** box (enabled only for a user shape on a cone) is set to **Apex**, the **Distance** box becomes the radial distance of the user shape from the centerline of the cone.

Rotation angle (Y) - Specifies the rotation angle about the Y-axis.

Rotation angle (X) - Specifies the rotation angle about the X-axis.

Rotation angle (Z) - Specifies the rotation angle about the Z-axis.

Centerline distance - Specifies the component centerline distance.

(C) Offset distance - Specifies the offset distance from the center of the parent component.

(D) Center to face distance - Specifies the component face distance to the center of the parent component.

(E) - Layout angle - Specifies the angle between the component centerline and the X-axis.

User Shapes direction - Specifies the component direction.

Impeller



Specifies the parameters for the impeller.

Description - Specifies an alphanumeric description for the component. This description is visible even when the component is collapsed in the grid.

Width - Specifies the component width.

Diameter - Specifies the diameter of the component.

Extension Diameter - Specifies the diameter of the extension.

Extension Length - Specifies the length of the extension.

Flip Orientation - Reverses the orientation of the component.

Shaft



Specifies the parameters of the shaft.

Description - Specifies an alphanumeric description for the component. This description is visible even when the component is collapsed in the grid.

Length - Specifies the distance from the starting point to the ending point of the component.

Diameter - Specifies the diameter of the component.

Coupling



Specifies the parameters for the coupling.

Description - Specifies an alphanumeric description for the component. This description is visible even when the component is collapsed in the grid.

Length - Specifies the distance from the starting point to the ending point of the component.

Diameter - Specifies the diameter of the component.

Motor



Specifies the parameters of the motor.

Description - Specifies an alphanumeric description for the component. This description is visible even when the component is collapsed in the grid.

Cylinder Length - Specifies the distance from the starting point to the ending point of the component.

Cylinder Radius - Specifies the cylinder radius.

Shaft Length - Specifies the distance from the starting point to the ending point of the component.

Shaft Diameter - Specifies the diameter of the component.

Flip Orientation - Reverses the orientation of the component.

Box



Specifies the parameters of the box.

Description - Specifies an alphanumeric description for the component. This description is visible even when the component is collapsed in the grid.

Dimension basis - Specifies either the inside or outside dimension. Select the dimension type from the list.

Length - Specifies the distance from the starting point to the ending point of the component.

Width - Specifies the component width.

Height - Specifies the component height.

Thickness - Specifies the component thickness.

Start offset X - Specifies an offset distance along the box X-axis. This value is applied at the start point of the box.

Start offset Y - Specifies an offset distance along the box Y-axis. This value is applied at the start point of the box.

End offset X - Specifies an offset distance along the box Y-axis of the box. This value is applied at the end point of the box.

End offset Y - Specifies an offset distance along the box Y-axis. This value is applied at the end point of the box.

Rotation angle - Specifies a box rotation angle.

Insulation - Displays the **Insulation** dialog box. For more information, see *Insulation Dialog Box* (on page 140).

Pump Base



Specifies the parameters for the pump base.

This component cannot be created by itself. It must be attached to the following component:

- *Impeller* (on page 136)

Description - Specifies an alphanumeric description for the component. This description is visible even when the component is collapsed in the grid.

Length - Specifies the distance from the starting point to the ending point of the component.

Width - Specifies the component width.

Height - Specifies the component height.

Start X - Specifies the X-coordinate of the starting point of the pump base. The value is relative to the starting point of the component to which it attaches.

Start Y - Specifies the Y-coordinate of the starting point of the pump base. The value is relative to the starting point of the component to which it attaches.

Start Z - Specifies the Z-coordinate of the starting point of the pump base. The value is relative to the starting point of the component to which it attaches.

Plane - Specifies the plane to which the pump base is parallel. There are three planes available:

- **XY**
- **YZ**
- **ZX**

Rotation Angle - Specifies the pump base rotation angle.

Pump Nozzle



Toolbar:

Specifies the parameters for the pump nozzle.

This component cannot be created by itself. It must be attached to the following component:

- *Impeller* (on page 136)

Description - Specifies an alphanumeric description for the component. This description is visible even when the component is collapsed in the grid.

Distance - Specifies the distance from the bottom of the component to which this component is attached. Click to display the **Nozzle placement** dialog box.

Nozzle Details - Displays the standard nozzle details. Click to display the **Nozzle details** dialog box.

Layout Angle - Specifies the pump nozzle layout angle.

Length - Specifies the distance from the starting point to the ending point of the component.

Suction Side - Specifies the nozzle suction side.

Direction - Specifies the nozzle direction.

Nozzle placement Dialog Box

Specifies advanced options for nozzle placement. A figure on the dialog box displays the values. This figure changes based on the type of component to which the nozzle is attached.

(A) Distance - Specifies the distance from the bottom of the component to which this component is attached. When the **Nozzle direction** box (enabled only for a nozzle on a cone) is set to **Apex**, the **Distance** box becomes the radial distance of the nozzle from the centerline of the cone.

(B) Lateral angle - Specifies the nozzle lateral angle.

Rotation angle (Y) - Specifies the rotation angle about the Y-axis.

Rotation angle (X) - Specifies the rotation angle about the X-axis.

Rotation angle (Z) - Specifies the rotation angle about the Z-axis.

Centerline Distance - Specifies the component centerline distance.

(C) Offset distance - Specifies the offset distance from the center of the parent component.

(D) Center to face distance - Specifies the component face distance to the center of the parent component.

(E) Layout angle - Specifies the angle between the component centerline and the X-axis.

Nozzle direction - Specifies the component direction.

Nozzle Details Dialog Box

Specifies standard nozzle parameters.


The lists for **Diameter**, **Flange rating**, **Flange type**, and **Schedule** are populated based on the entries of the NozzleDropDown.TBL file located in the *[Product Folder]\Equipment\System* folder.

Nozzle basis - Specifies whether you are using nominal or actual parameters or CADWorx Steel data file parameters. When you select **Nominal**, Equipment provides a look-up system for flange dimensions and schedule information.

Diameter - Specifies the diameter of the component.

Schedule - Specifies the schedule rating for the nozzle.

Wall thickness - Specifies the component thickness.

Nozzle extension - Indicates whether a nozzle extension (elbow) is included. Click  to display the **Nozzle extension** dialog box.

Nozzle material - Displays the **Select Item** dialog box, which lets you select the material from the list. For more information, see *Select Item Dialog Box* (on page 139). The **Select Item** dialog box uses the file specified by the **MaterialFile** configuration file variable. For more information, see *MaterialFile* (on page 32).

Flange type - Specifies the type of nozzle flange. If you set this value to **N/A**, then you can manually specify the **Flange thickness** and **Flange OD** values. For **Flange type** values other than **N/A**, the software uses the dimensions from the FlangeImperial.DAT or FlangeMetric.DAT file. These files are located in the *[Product Folder]\Electrical\System* folder.


Flange rating - Specifies the nozzle flange rating.


Flange thickness - Specifies the component thickness.

Flange OD - Specifies the component outer diameter.

Flange material - Displays the **Select Item** dialog box, which lets you select the material from the list. For more information, see *Select Item Dialog Box* (on page 139). The **Select Item** dialog box uses the file specified by the **MaterialFile** configuration file variable. For more information, see *MaterialFile* (on page 32).

Thread/Engagement length - Specifies the nozzle end point location. This end point provides a snap location for any component attached to the nozzle.

Nozzle reinforcing pad - Indicates whether a reinforcing pad exists. Click  to display the **Nozzle Pad** dialog box.

Nozzle Insulation - Indicates whether insulation exists. Click  to display the **Insulation** dialog box.

Select Item Dialog Box

Lists all the materials available in the material file or all the motor sizes available in the motor file. The material file can be specified using the *MaterialFile* (on page 32) configuration file variable. The motor list is located in the *[Product Folder]\PVFabricator\System\Emotor.TBL* file (Emotorm.TBL file for metric).

Filter - Specifies a filter to quickly find the item you need. The filter is case sensitive and accepts the following wild-card characters:

Character	Definition
# (pound)	Matches any single numeric digit.
@ (at)	Matches any single alphabetic character.

. (period)	Matches any single non-alphanumeric character.
* (asterisk)	Matches any character sequence, including an empty one. This character 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).

Insulation Dialog Box

Specifies parameters for insulation.

The modifiable insulation material database is stored in *[Product Folder]\Equipment\System\InsulationMaterial.TXT* file.

Apply insulation - Indicates whether insulation is applied to the component.

Description - Specifies an alphanumeric description for the component. This description is visible even when the component is collapsed in the grid.

Material - Displays the **Select Item** dialog box, which lets you select the material from the list. For more information, see *Select Item Dialog Box* (on page 139). The **Select Item** dialog box uses the file specified by the **MaterialFile** configuration file variable. For more information, see *MaterialFile* (on page 32).

Density - Specifies the density of the insulation.

Thickness - Specifies the component thickness.

Bolt Details Dialog Box

Diameter - Enter the nominal bolt diameter.

Thread Series - Select the thread series identifier:

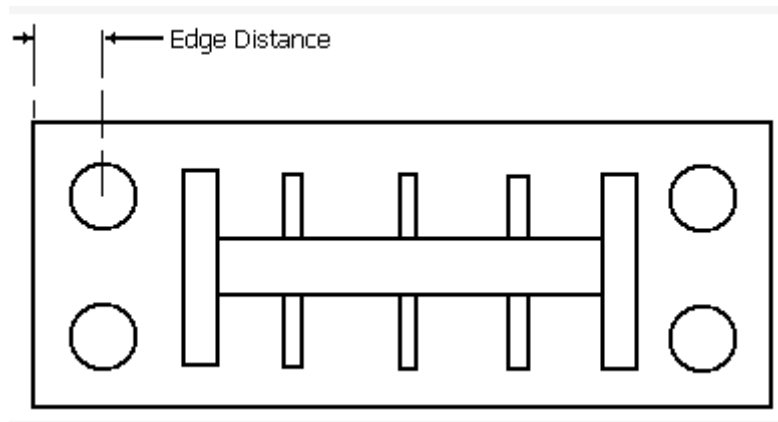
- **TEMA** - 8-thread series, adapted from the TEMA Standard
- **UNC** - Unified National Course threads, adapted from Mark's Handbook
- **User Defined Root Area** - Enables you to enter a custom root area in **Root Area**. You can obtain this information from a standard engineering handbook.
- **TEMA Metric**

- **BS 3643**
- **SABS 1700**

Root Area - For nonstandard or metric bolts, enter the root cross-sectional area of the bolt. You must use the **User Defined Root Area** option to enter the bolt root area.

Material - Displays the **Select Item** dialog box, which lets you select the material from the list. For more information, see *Select Item Dialog Box* (on page 139). The **Select Item** dialog box uses the file specified by the **MaterialFile** configuration file variable. For more information, see *MaterialFile* (on page 32).

Number of bolts - Enter the total number of bolts to be used on the baseplate. Bolts are assumed to be at the edge of the baseplate along the short side.



Edge distance - Enter the distance from the edge of the baseplate to the centerline of the bolts.

SECTION 13

FAQ

Provides answers to frequently asked questions.

Creating a Base Profile

To create a base profile, you can copy the AutoCAD icon or PVFabricator icon, and specify a profile. This profile is based on the default AutoCAD profile.

1. Make a copy of the PVFabricator shortcut on the desktop.
2. Right-click the shortcut and select **Properties**.

*The **Properties** dialog box appears.*

3. On the **Shortcut** tab, replace the value after /P in **Target** with the new profile name. For example:

"C:\Program Files\Autodesk\AutoCAD - English\acad.exe" /P UserBase

NOTE You can use any name that is not used on the **Profile** tab of the **Options** dialog box (**Tools > Options**) in AutoCAD. Use quotes around the profile name if it contains spaces.

4. Click **OK**.
5. Double-click the shortcut to start the software.

AutoCAD creates a new profile based on the software defaults.

Xref Bind Restriction

In order to run the AutoCAD **Xref Bind** option command on a drawing that contains CADWorx data, CADWorx must be running.

AutoCAD Redo Limitations

The AutoCAD **REDO** command is not available.

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