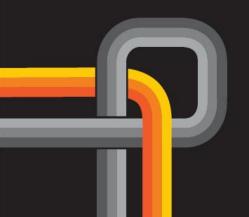
SolidPLANT Getting Started

Comprehensive Plant Design for SolidWorks



Route it your way!



Version SolidPlant 2016

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Introduction

The purpose of this document is to teach new users the fundamentals of SolidPlant3D and assist them with the main features.

SolidPlant 3D is an add-in on top of SOLIDWORKS standard edition which allows the designers to quickly create intelligent 3D process plant designs driven by specifications, and deliver easily de facto standard isometric drawings (which is Isogen[®], the Alias solution embedded in SolidPlant 3D).

SolidPlant 3D is a database driven system and provides an application called SpecCreator for the generation of the piping specifications, with a wide range of catalogues to build up the needed piping components for the project.

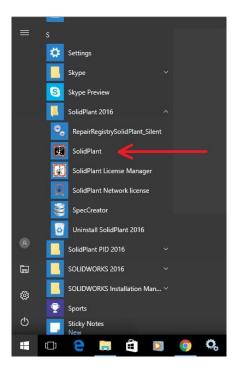
Launching

Solidplant 3D can be launched in three different ways.

• From the desktop shortcut.



From Windows Start, Programs, SolidPlant 2016, SolidPlant.

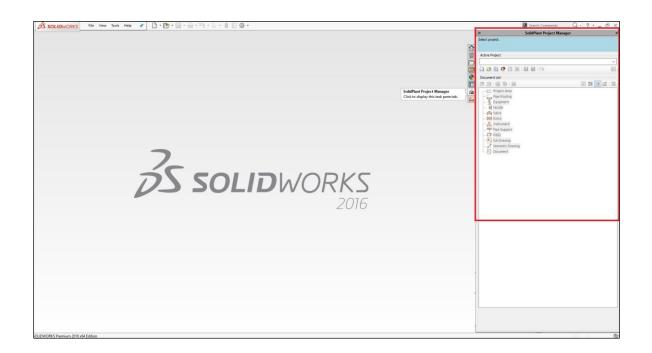


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	Macro +	SOLIDWORKS Toolbox Library		-	
	Add-Ins	SOLIDWORKS Toolbox Utilities			
	Save/Restore Settings	SOLIDWORKS Utilities			
	Customize	SOLIDWORKS Workgroup PDM 2016		-	
	Options	TolAnalyst			
		SOLIDWORKS Add-ins			
		Autotrace			
		SOLIDWORKS Composer		< 1s	
		SOLIDWORKS Electrical		1m 14s	
		SOLIDWORKS Forum 2016		< 1s	
		SOLIDWORKS Plastics		-	
		Other Add-ins			
1	$ \rightarrow $	SolidPlant		15	
		SOLIDWORKS 3DEXPERIENCE SmartLink		-	
		SOLIDWORKS PCB 2016		-	DDVC
		SOLIDWORKS XPS Driver 2016		-	JRKS
				~	
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• From SolidWorks, Tools, Add-ins, select the Active Add-ins check box SolidPlant.

SolidPlant Project Manager

On starting SolidPlant the **SolidPlant Project Manager** task pane will be automatically displayed by default on the right hand side of the screen. Through this interface it is possible to create new projects, open existing projects and create copies of projects.

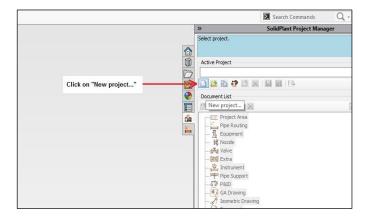


Creating a new project

In SolidPlant 3D, the projects are assemblies of SOLIDWORKS consisting of many components, which can be parts or other assemblies, called subassemblies. The pipe lines and equipment created with SolidPlant 3D are subassemblies and the structure models can be parts or subassemblies.

1. Click New project... command.

To create a new project with SolidPlant 3D, click **SolidPlant Project Manager** task pane > **New project...**



When prompted the **Create new project** dialog box, the user can define the fields **Project Name**, **Project Description** and **Directory**, where the project will be located.

On **Project Unit** area is displayed the three different types of system of units in which a project can be defined. Once the project has been created, the project unit can't be changed anymore.

Option	Description
Imperial	Imperial system of unit
Metric	International system of unit
Mix-Metric	A combination of imperial and metric units. Nominal diameter is on imperial unit and the rest of the values are on metric unit (length, weight, etc.)

2. Define the project properties.

Add the values on below for the property fields and option required.

- **Project Name**: Type **Project 01.**
- Project Description: Type Getting Started Project.
- Directory: C:\ProgramData\SolidPlant Projects
- Project Unit: Select option Mix-Metric.

Project Name :	
Project 01	
Project Description :	
Getting Started Project	
Directory :	
C:\ProgramData\SolidPlant Projects\	
Project Unit	
🔿 Imperial	
All unit are reported in inches.	
O Metric	
All unit are reported in millmeters.	
Mix-Metric	
Most unit are reported in millmeters, but nominal dia	meter is reported in inches.



- **3.** Click **Create** and the project assembly will be displayed.
- 4. Click Save 🔚 on the Standard toolbar.

Open a project

There are two ways to open a project.

• From the dropdown menu **Active Project**, it will list all the projects created previously. Select a project name and it will be opened.

	»	SolidPlant Project Manager
	Select project.	
	Active Project	
	1	÷
	SPPROJ2016	٨
	Project 01	
V	Document List	
	20 12 12 12 12 12 12 12 12 12 12 12 12 12	
6ª	Project Area	-
1	Pipe Routing	Drop down list
	Nozzle	ect the project
	Valve Sele	et the project

• Click SolidPlant Project Manager task pane > Open project... <a>[2].

	» SolidPlant Project Manager	×
	Select project.	
	Active Project	
Ď		~
		100 A
•	Doc Open project	
6å	Project Area	
E	Pipe Routing Equipment Nozzle	
	Valve	

When prompted the browse window, the project can be opened via two different types of files within the project folder. It doesn't matter which type of file the user chooses that the project will be opened.

By default the project folder is located in C:\ProgramData\SolidPlant Projects.

Via.mdf file.

SolidPlant 3D is a database driven system and uses Microsoft SQL Server 2014 as a relational database management system. The .mdf file of the project is used by Microsoft SQL Server for storing the database files and Schema.

Browse into the project folder that we have created previously, select Project 01.mdf file and click **Open** _____.

Irganize 👻 New folder					833 -	
	^					
E Pictures 🖈 ^	Name	Date modified	Туре	Size		
Documentacion	CableTray	10/6/2016 11:08 AM	File folder			
📙 Getting Started F	Component	10/6/2016 11:08 AM	File folder			
	- Document	10/6/2016 11:08 AM	File folder			
🏪 Local Disk (C:)	Drawing	10/6/2016 11:08 AM	File folder			
	Equipment	10/6/2016 11:08 AM	File folder			
ConeDrive	GA Drawing	10/6/2016 11:08 AM	File folder			
This PC		10/6/2016 11:08 AM	File folder			
Desktop	Isometric Drawing	10/6/2016 11:08 AM	File folder	Selec	t .mdf file of the proje	ct
Documents	P&ID	10/6/2016 11:08 AM	File folder	ocico	a line of the proje	
Downloads		10/6/2016 11:08 AM	File folder			
	Pipe Support	10/6/2016 11:08 AM	File folder			
J Music	Specification	10/6/2016 11:08 AM	File folder			
Pictures	Structural	10/0/2010 11/00 11/1	Filefolder			
📕 Videos	📴 Project 01.mdf	10/6/2016 11:08 AM	SQL Server Databa	69,632 KB		
Local Disk (C:)						
Network						
INELWOIK						
~						

Via . SDASM file.

The .SDASM file is the SOLIDWORKS assembly document, so that the project can be opened from the project assembly as well.

Select Project 01.SDASM file located in the Drawing folder within the project folder, and click **Open** .

	C → Local Disk (C:) → ProgramDa	ita > SolidPlant Projects > Pr	oject ur > Drawing	ٽ v	Search Drawing	
ganize 👻 New folder						
🔜 Desktop 🛛 🖈 📩	Name	Date modified	Туре		Size	
Documents 🖈	Project 01.SLDASM	10/6/2016 11:08 AI	M SOLIDWORKS Assemb	ly Document	66 KB	
📰 Pictures 🛛 🖈 📕						
- Documentacion						
Getting Started F						
July 2016	Calant CDACM File					
Local Disk (C:)	Select SDASM file	of the project				
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Automatically, once the project has been opened, it will be list on the dropdown menu Active Project on the task pane SolidPlant Project Manager.

User Interface

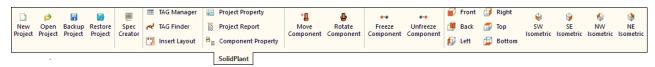
The SolidPlant 3D user interface is fully integrated within SOLIDWORKS interface. SolidPlant 3D has added two side **Task Panes** on the right hand side, and six **Application Tabs** on the top side in the CommandManager menu of SOLIDWORKS.

SS SOLIDWORKS File Edit View Insert Tools Window Help	• 🗋 • 🖄 • 🔚 • 🚔 • 🐃 • 🜄 • 🛢 📓 @ •	Project 01.SLDASM *	🖬 Search Commands 🛛 Q ? - 👝 🗗 🗙
Image: Specific Spectific Specific Specifi	Move Ratate Component Comp	SE NW ME Durn Rote Add strends SE NW ME Durn Rote Add strends SoleNets Log Rotation project June SoleNets Commercial Rotation	p Component EPDM Project that can be accessed from SolidPlant
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Application tabs

There are six tabs available in SolidPlant 3D on the CommandManager of SOLIDWORKS that is a context-sensitive toolbar. When you click a tab below the CommandManager, it dynamically updates based on the toolbar the user wants to access.

SolidPlant tab



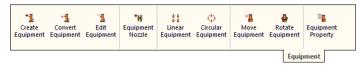
Provides a number of commands related to the management of the project, different views, access to SpecCreator application, project reports...

Structure tab

* Structure 3D Grid	\$ C	nsert Structure Member Draw Structure Member Channel Library	f Spiral Stair					∦* Pipe-Bridge	E Auto-Trim	Insert Connection	Structure Building
					Structure						

This toolbar includes commands to create structure 3D grids, insert structure profiles, automatic generation of structure models (stairs, ladders, platforms, handrails, etc.), cutting list report...

Equipment tab



This toolbar includes commands to create equipment from the SolidPlant libraries, convert equipment, edit equipment, management of the equipment nozzles, creation of linear and circular patterns...

Routing tab

- Manual Route			×	Delete Route	Rebuild Component *g Delete Component X# Assign Length		Rotate Component		Component		Valve	Isometric Attachment	◆ Duct/Cable-Tray	Import-PCF	Multi-Route Navigator
	Routing														

This toolbar contains all the tools necessary to create and edit the pipe lines, as well as commands to add pipe supports and valve operators, component properties...

Component tab

H 🏄 Pipe Bended Pipe	Stub-In Olet	U	180 Return	¥	Wye	 Concentric Reducer Eccentric Reducer Connector) Cap	 				expansion Joint		Components	Custom components	Convert to Fitting
									Com	ponent	t					

Provides a number of commands related to the piping components (elbow, flange, tee, instrument, valve, etc.), stub-in...

SP Drawing tab

🖧 Gen PCF	A POS Template	New drawing with annotations	GA drawing with annotations	Elevation cut	ff Structure Member Table	вом	Line Number	TAG	-	TOP COP BOP	Symbol	▶ब Pipe Cut Length	Insert weld check list table
													SP Drawing

The last toolbar includes the automatic generation of isometric drawings, commands to create different type of drawings, bill of material, TAG annotations...

Task Panes

There are two task panes available in SolidPlant 3D on the Task Pane of SOLIDWORKS on the right hand side of the interface. Selecting any of the SolidPlant tabs, the task pane dynamically updates according to the users' needs.

When opening a project in SolidPlant 3D, all the commands and features of the two SolidPlant task panes will be dynamically updated and made available.
 When there is no project opened, just a few commands on the SolidPlant Project Manager pane will be available.

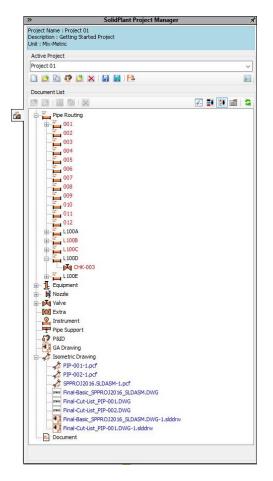


SolidPlant Project Manager task pane

When opening a project, the number of commands available is automatically updated in order to provide the tools to manage the project.

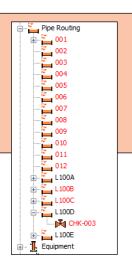
The node tree **Document List** is displaying all the tags of the components and the documentation generated in the project sorted by type.

Туре	Components
Pipe Routing	Pipe lines.
Equipment:	Pumps, Exchangers
Nozzle	Nozzle Equipment.
Valve	Valves and Control valves.
Extra	Strainers, Spacers
Instrument	Gauges, Thermowells
Pipe Support	Pipe Supports.
GA Drawing	.SLDDRW files.
Isometric Drawing	.PCF,.DWG,.SLDDRW files.
Document	.XLS, .docx, .PDF files



There are three different colors which are showing the status of each tag in the project.

Red: Just the data value is in the project database.Blue: There is a 3D model created ready to bring into the project.Black: The component is in the project assembly.



SolidPlant Pipe Routing task pane

This interface is divided up into four main sections providing the tools to create pipe lines as well as to edit the values of the active pipe line, etc.

X
~
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0

Active Route

The dropdown menu displays the active line in use.

New pipe model...: For creating a new line.

Delete pipe model..: For deleting the current line.

•••

Piping Reference

Specification: Dropdown menu displaying the piping specification in use.

Pipe Size: Dropdown menu displaying the pipe size in use from the available sizes contained into the specification.

The check boxes enable or disable different features according to the needs of the user.

Piping Sketch Tool

Snap: Snap to center, Along X Axis...Relation: Coincident to entity, Parallel to entity...Slope: Scale factor, Angle degrees of the slope.

Pipe Routing Component

The node tree displays the piping components available within the current piping specification (e.g. **Demo**) and pipe size (e.g. **8**").

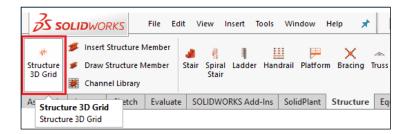
Creating structures

There are many different tools to create steel structures with SolidPlant 3D and all are displayed in the **Structure** tab. Depending on the command type that is used a variety of task panes, wizards, windows, etc. will become available to accomplish the job.

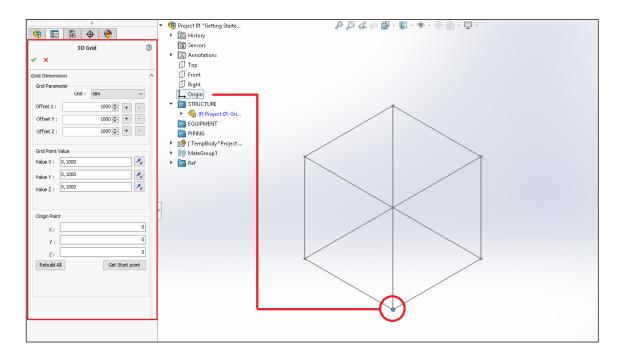
Structure 3D Grid command

To create a Structure 3D Grid:

1. Click Structure Tab > Structure 3D Grid.



The **3D Grid** task pane will be automatically displayed on the left hand side in the PropertyManager of SOLIDWORKS, and a 3D grid frame will be placed by default on the Origin point of the assembly project.



The origin point of the 3D grid frame can be controlled in two different ways.

Via Origin Point fields.

Type any random value along the axis fields **X**, **Y** and **Z** to test.

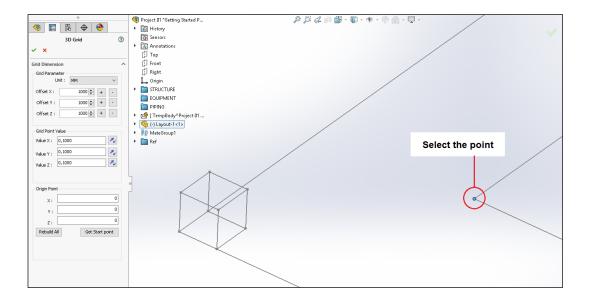
0	
	🔻 🍕 Project 01 "Getting Starte 🖉 🎉 🎸 🗊 🎬 - 🧊 - 🔶 🌦 - 🖵 -
🏟 📰 🖹 🕁 🥙	History
3D Grid (2)	🐼 Sensors
_	Annotations
✓ ×	Птор
Grid Dimension	
Grid Parameter	Right
Unit : MM 🗸	
Offset X : 1000 🜩 + -	
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Offset Y : 1000 🜩 + -	EQUIPMENT
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Grid Point Value	→ I MateGroup1
Value X : 300,1300	Ref
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Origin Point	
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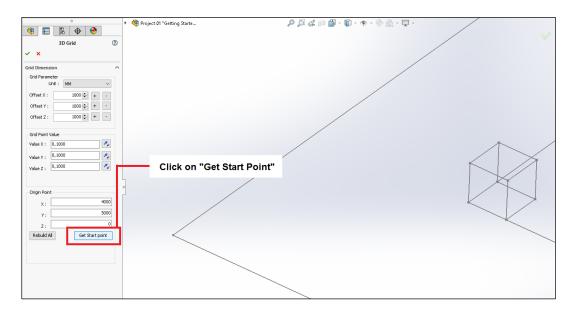
In order to update the 3D grid entity with the new values added into the parameter fields, after typing the value, the user must click on any other field within the **3D Grid** pane and automatically the grid entity will be updated.

• Via Get Start point command.

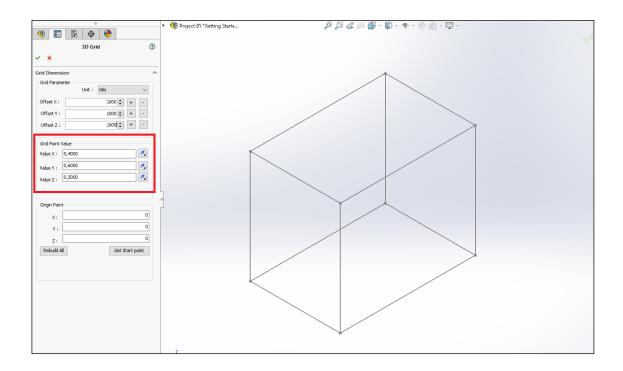
P

By selecting any sketch point within the project assembly, and then click **3D Grid** task pane > **Get Start point** Get Start point, the 3D grid will be placed automatically on the point selected.

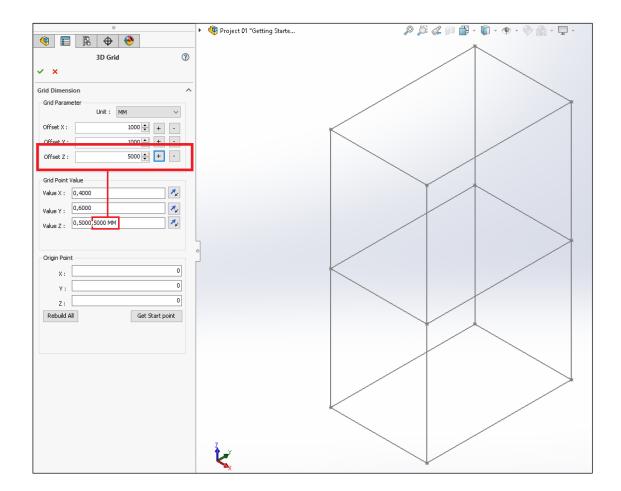




By typing values on the fields **Value X**, **Y** and **Z** grouped in the **Grid Point Value** area, the user controls the dimensions along the axis of the 3D Grid. With this command *I* the user changes to the opposite direction along the selected axis.



To add new levels in the 3D grid, the user can type any value on the fields **Offset X**, **Y** and **Z** on the Grid Parameter area, and then click command + to add a new level. The offset value will be added after a comma into the fields **Value X**, **Y** and **Z** on the **Grid Point Value** area, and the 3D grid will be updated automatically. Click command - to delete a level.





Also the user can change manually the values in the fields **Value X**, **Y** and **Z** on the **Grid Point Value** area.

- Add new levels by typing a comma after any value and type a new value of the level.
- Delete manually any level by suppressing its value.
- Edit any level by changing the value of an existing one.

Grid Point	Value	
Value X :	0,4000 <mark>,</mark> 3500	
Value Y :	0,6000, 4000	
Value Z :	0,5000,5000, 4200	

2. Add the values on the Grid Point Value fields.

Type the values on below to define the 3D grid entity.

- Value X: 0, 4000
- Value Y: 0, 6000
- Value Z: 0, 5000, 5000

Grid Point	Value	
Value X :	0,4000	X _2
Value Y :	0,6000	X _2
Value Z :	0,5000,5000	7 *

3. Click **3D Grid** task pane **> OK v** to confirm and close the pane.

To edit again the 3D grid created.

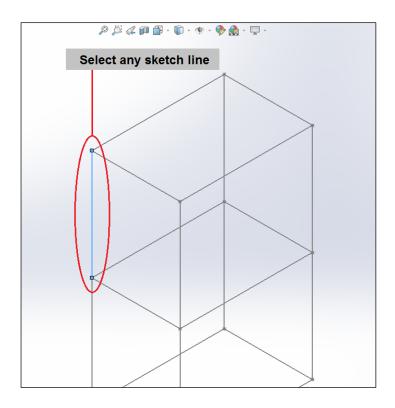
Click on any sketch line of the grid and then click **Structure** Tab > **Structure 3D Grid**. The **3D Grid** pane will be displayed again for further editing.

Insert Structure Member command

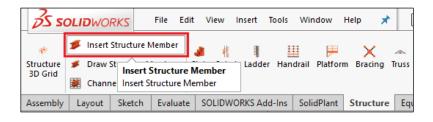
To insert structure profiles:

1. Select any sketch line.

Select any sketch line contained into the 3D grid previously created.



2. Click Structure Tab > Insert Structure Member.



The **Structural Member** task pane will be automatically displayed on the left hand side in the PropertyManager of SOLIDWORKS. SolidPlant 3D provides, apart from what is already in SOLIDWORKS, a new wide range of standards, types of profiles and sizes of structural members.

The user can define the profile of the structure from the dropdown menus **Standard**, **Type** and **Size** in the **Selections** area.

The SolidPlant weldment profiles are located in: C:\Program Files\SolidPlant\Structure\Weldment Profiles.

By selecting the sketch lines, the profiles will be added automatically. All structural members in a single group must use the same profile.

Define the values of the weldment profile.
 Select the values on next page from the dropdown menus.

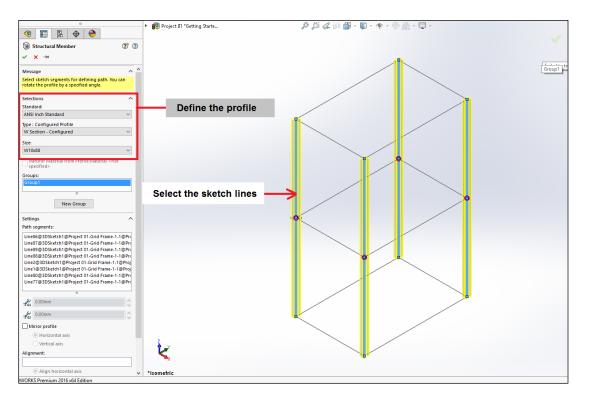


- Menu Standard: Select ANSI Inch Standard.
- Menu Type: Select W Section Configured.
- Menu Size: Select W10X88.

Selections	^
Standard:	
ANSI Inch Standard	\sim
Type : Configured Profile	
W Section - Configured	\sim
Size:	
W10x88	\sim

4. Select the vertical sketch lines.

Select all the vertical lines one by one in order to add the columns into the 3D grid.



- 5. Click Structure Member task pane > Ok 🔽 to confirm.
- 6. Add also the structure beams using the same procedure.
- 7. Click Edit Mode symbol to exit once the structure is finished.



Structure modelers

A number of structure modelers are available in the **Structure** tab to create automatically different type of structures easily like stairs, handrails, platforms, etc.

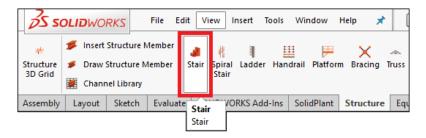
The way of working is similar on most of the modelers. As a sample, the user will learn to use the **Stair** modeler.

ructure Member		1	Ш	⊨ ×	*	÷		¢	雨	*	3
ructure Member	Stair Spiral Stair	Ladder	Handrail Pla	atform Bracing	Truss	Floor Plate	Footing	Walkway	Conveyor	Pipe-Bridge	Assign Member
l Library	Stan					Flate					TAG
Structure E	quipment	Routing	Component	SP Drawing							

Stair command

To create a Stair:

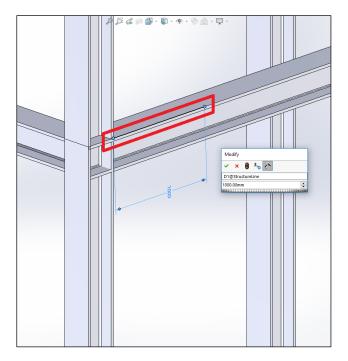
1. Click Structure Tab > Stair.



The user is on 3D sketch mode and the tool Line \checkmark is active. Drawing a single line will mark the start point of the stairs, and its dimension will set the width of the model.

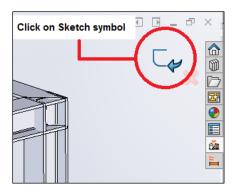
2. Sketch a line.

Sketch a line on the edge of the beam and set a dimension of 1000mm length.

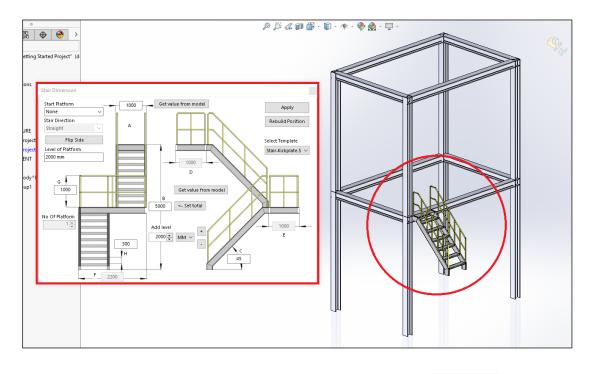




3. Exit from Sketch mode.



The window **Stair Dimension** will be displayed and the default stair structure will be created from the line previously drawn.

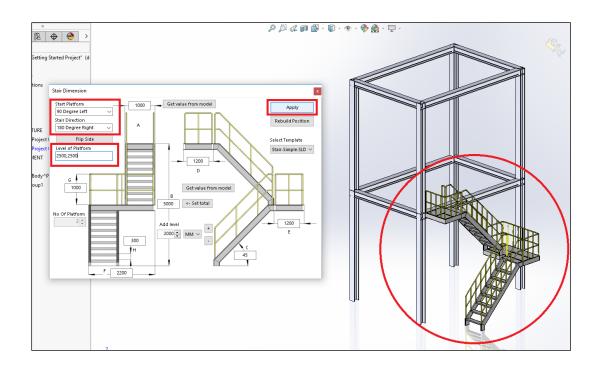


Setting the parameters on any of the available fields, and clicking **Apply**, the stair will be updated automatically.

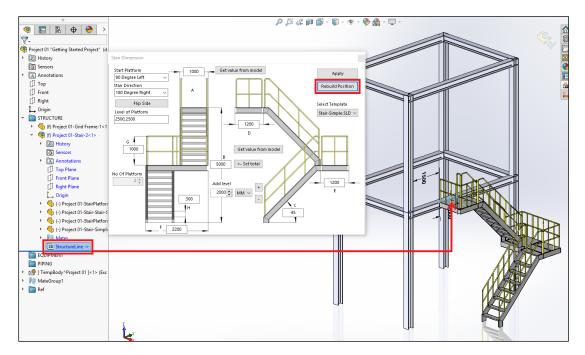
4. Define the parameters of the stair.

- Start Platform dropdown menu: Select 90 Degrees Left.
- Level of Platform: Type 2500, 2500. It can be added as many as levels needed after each comma.
- Start Direction dropdown menu: Select 180 Degrees Right.

5. Click Apply Apply to update the 3D model of the stair.



Moving to a new position the start sketch line, and then clicking on **Rebuild Position** button, the stair will move on to the new position. The StructureLine sketch is located into the stair assembly.

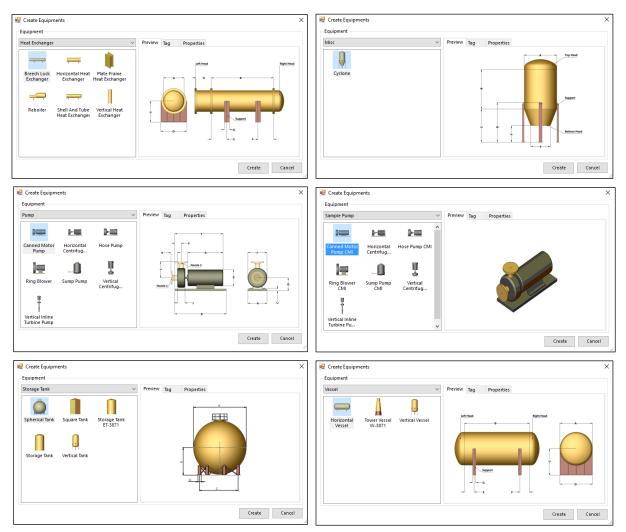


To edit again the stair created.

Select any part of the stair and then click **Structure** tab > **Stair**. The **Stair Dimension** window will be displayed again for further editing.

Create equipment

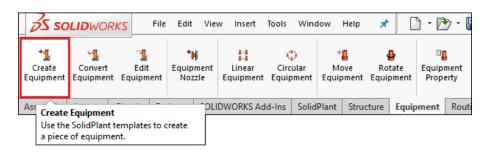
SolidPlant 3D provides a full library of templates with all types of equipment fully parameterized which allows the user to easily change the main dimensions. Apart from the library of SolidPlant 3D, the user can use his own designs and bring into the project any model, and intuitively assign the nozzles needed on the equipment.



Pump Equipment

To create a pump from SolidPlant 3D library:

1. Click Equipment Tab > Create Equipment.



- 2. When prompted the window Create Equipment:
 - From dropdown menu **Equipment**, select **Pump**.
 - Select the icon Horizontal Centrifugal Pump.
 - Click TAG Property 🗋.

ump			Preview Tag Properties
Canned Motor Pump	Horizontal Centrifug	Hose Pump	TAG: Click on this button
Ring Blower	Ü Sump Pump	Vertical Centrifug	
H I			Select this model
Vertical Inline Turbine Pump			

- 3. From the window TAG Property:
 - TAG field: Type PUMP-001.
 - Description field: Type Pump Getting Started (Optional).
 - Click Create.

TAG Property			_	x
Group :	Equipment	~	TAG :	PUMP-001 V
Description :	Pump Getting St	arted		
Service/Type :		~	Comp Turne	e a tag
Specification :		\sim	Main S.	; a lay 🗸 🗸
Line To Use :		~	Area :	~
From :		~	To :	~
Insul. Thick :	0		Color By :	~
Additioning Prop	perty			
Property		Value		
► LOCATION				
			Click	on Create
5				,
Edit Column				Create Cancel

4. Click Create Create to confirm.

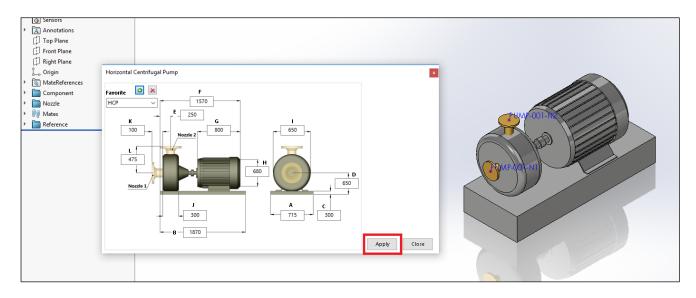
Pump			\sim	Previe	w Tag Properties		
-				TAG :	PUMP-001	~ 🗋	
	<u>1-100</u>				Property	Value	
Canned Motor Pump				•	DESCRIPTION	Pump Getting Started	
Pump	Centinug				TAGTYPE		
le l	臣 🕛 !				INSULATION_THICK	0	
		_			LOCATION		
Ring Blower	Sump Pump	Vertical Centrifug			AREA		
-		centinogiii			TYPE		
Vertical Inline Turbine Pump							
				<			>



The new assembly model of the pump is opened along with the **Horizontal Centrifugal Pump** dialog box to define the dimensions from **A** to **L** fields.

5. Set the dimensions of the pump.

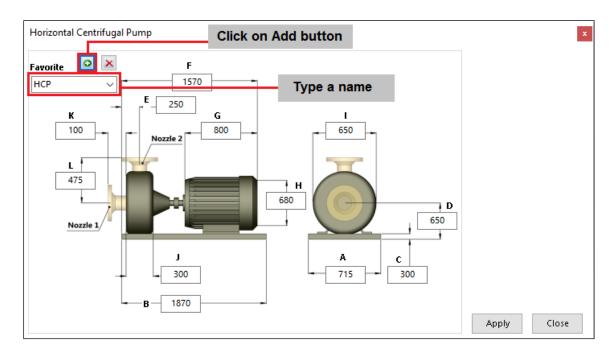
Type the values shown on below picture into the dimesions fields on the Horizontal Centrifugal Pump window.



6. Click Apply _____ to update the model with the dimensions just set.

7. Create a favorite setting.

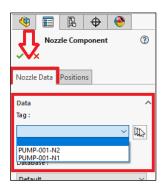
Type on **Favorite** field any name and then click **Add** in order to save the current parameters of the pump. From the dropdown menu **Favorite** will list the newly created setting. Click **Delete** to suppress the selected favorite.



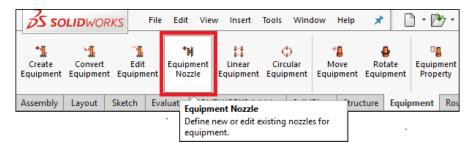
8. Click Close in the Horizontal Centrifugal Pump dialog box.

The **Nozzle Component** task pane is displayed on the left hand side in the PropertyManager of SOLIDWORKS in order to set the parameters of the equipment nozzle.

By default this pump comes with two nozzles, the user can set the parameters of the existing nozzles by selecting from the **Tag** dropdown menu the available tags on the **Nozzle Data** tab.



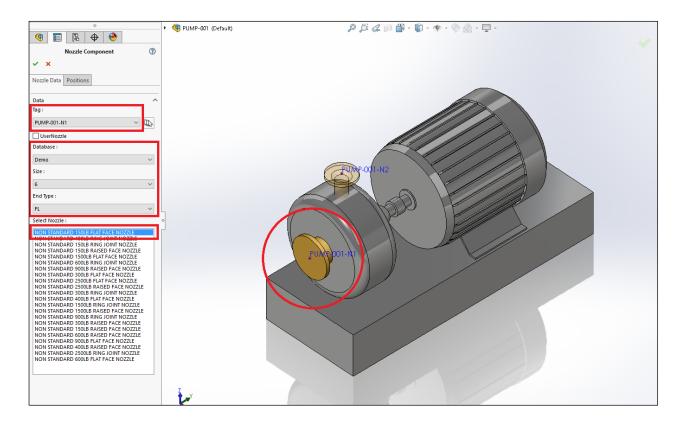
If the **Nozzle Component** task pane is not displayed in the PropertyManager, click **Structure** tab > **Equipment Nozzle** to display it.



To edit again the pump dimensions, click **Equipment** Tab **> Edit Equipment** to access the **Horizontal Centrifugal Pump** dialog box for further editing.

3S SOL		Edit Vie	w Insert	Tools	Wind	dow Hel	p 🖈) • 12	5-1	
Create Equipment E	Convert Equipment	Edit Equipment	*⊯ Equipment Nozzle	Linear Equipment	Circ) ular oment	Move Equipme	Ro	b tate pment	Equip Prop	ment
Assembly	Layout	ipment Parameters o ent.	of existing	Ins	Solid	Plant St	ructure	Equip	oment	Rout	

- Set the parameters of the nozzle pump PUMP-001-N1.
 Select the values on Nozzle Data tab in the Nozzle Component task pane.
 - Tag dropdown menu: Select PUMP-001-N1.
 - Database dropdown menu: Select Demo specification.
 - Size dropdown menu: Select 6 inches.
 - End Type dropdown menu: Select FL (Flange connection).
 - Selected Nozzle list: Select NON STANDARD 150LB FLAT FACE NOZZLE.



10. Set the parameters of the nozzle pump PUMP-001-N2. Select the values on **Nozzle Data** tab in the **Nozzle Component** task pane.

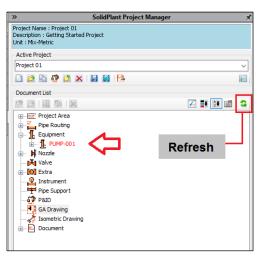
- Tag dropdown menu: Select PUMP-001-N2.
- Database dropdown menu: Select Demo specification.
- Size dropdown menu: Select 6 inches.
- End Type dropdown menu: Select FL (Flange connection).
- Selected Nozzle list: Select NON STANDARD 150LB FLAT FACE NOZZLE.

11. Click **Ok** \checkmark on the **Nozzle Component** task pane.

12. Save from the Standard toolbar and close the pump assembly.

Back to the project, on the **Document List** node tree is displayed the tag **PUMP-001**.

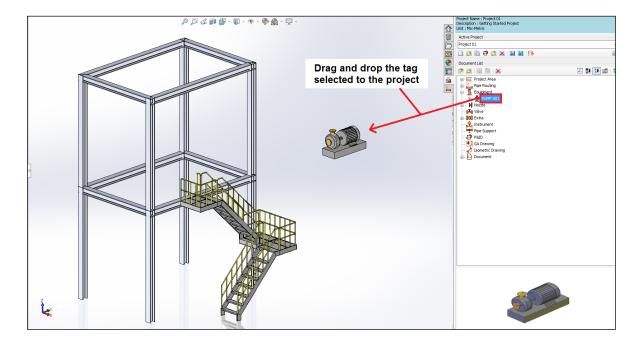
13. Click **Refresh** (2), if the tag is not displayed.



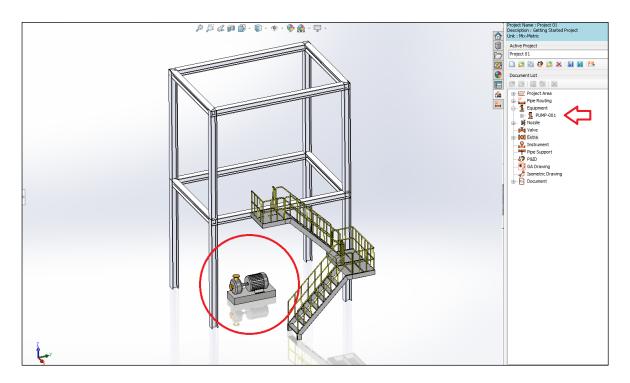


14. Drag & drop the pump into the project assembly.

Click on the tag **PUMP-001** on the **Document List** node tree with the scroll/middle mouse button and hold it pressed, then drag and drop the pump into the project assembly.



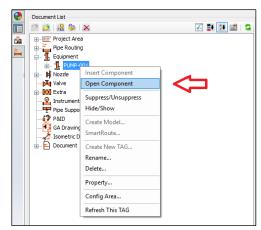
15. Place the pump in the middle of the structure using SOLIDWORKS mates. Click **Refresh** sutton to update the status of the tag **PUMP-001** on the node tree.



16. Save the project on the Standard toolbar.

To access the pump again.

Click right mouse button on the tag **PUMP-001** on the **Document List** node tree and select **Open Component** from the shortcut menu in order to open and edit the pump.



Vessel Equipment

To create a vessel from SolidPlant 3D library:

1. Click Equipment Tab > Create Equipment.

s	SOLID	WORK	S File	Edit	Vie	w Insert	Tools	Wind	dow	Help	*) • 🖻	þ • 🕻
Creat Equipm		1 nvert pment	Edit Equipment	•₩ Equipm Nozz	ient	Linear Equipment	Circ) ular oment		e ove oment		ate tate	Equip Prop	ment
Use	ate Equip the Solid iece of eq	Plant te	emplates to d	create	OLI	DWORKS Ad	d-Ins	Solid	Plant	Stru	cture	Equip	oment	Routi

- 2. When prompted the window Create Equipment:
 - From dropdown menu **Equipment**, select **Vessel**.
 - Select the icon Horizontal Vessel.
 - Click TAG Property 🛄.

💀 Create Equipments			×
Equipment			
Vessel 🗸 🗸 🗸	Preview Tag Properties		
	TAG :		~
	Property	Value	
Horizontal Tower Vessel Vertical Vessel W-3871			
	<		>
		Create	Cancel



- 3. From the window TAG Property:
 - **TAG** field: Type **V-001**.
 - Click Create.

TAG Property					x
Group :	Equipment	~	TAG :	V-001	~
Description :					
Service/Type :		~	Comp Type :		~
Specification :		\sim	Main Size :		\sim
Line To Use :		\sim	Area :		~
From :		\sim	To :		~
Insul. Thick :	0		Color By :		~
Additioning Prop	erty				
Property		Value			
► LOCATION					
<					>
Edit Column				Create	Cancel

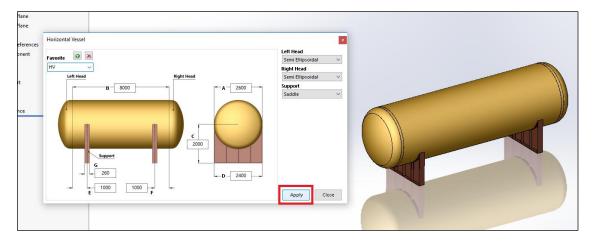
4. Click Create to confirm.

'essel		~	Preview	v Tag	Properties					
	4	0	TAG :	V-001			~			
البيب		()		Property		Value				
Horizontal Vessel	Tower Vessel W-3871	Vertical Vessel	•	DESCRIPTIO	ON					
VESSEI	11-3071			TAGTYPE						
				INSULATIO	N_THICK	0				
				LOCATION						
				AREA						
				TYPE						
						<				>

The new assembly model of the vessel is opened along with the **Horizontal Vessel** dialog box to define the dimensions from **A** to **F** fields.

5. Set the dimensions of the vessel.

Type the values shown on below picture into the dimesions fields on the **Horizontal Vessel** dialog box.





6. Click Close 🔼 .

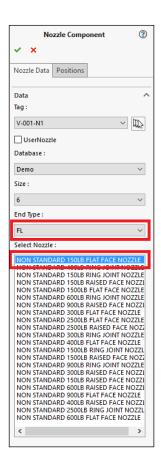
The **Nozzle Component** task pane is displayed on the left hand side. By default this equipment is without nozzle.

- 7. Click New Nozzle and the TAG Property window will be displayed.
- 8. From the window TAG Property:
 - Tag dropdown menu: Type V-001-N1.
 - Database dropdown menu: Select Demo specification.
 - **Size** dropdown menu: Select **6** inches.
 - Click **Create**.

· · · · · · · · · · · · · · · · · · ·	V-001 (Default) ->		ß) 🎉 📣 🖗 🗳 ·	· 🇊 • 🏟 •
Nozzle Component ⑦ ✓ ×	Click on "New l	Nozzle"			
Nozzle Data Positions					
Data	TAG Property				×
- UL	Group : Nozz	ie 🗸	TAG :	V-001-N1	~
UserNozzle	Description :				
Database :	Service/Type :	~	Comp Type :		\sim
Default 🗸	Specification : Dem) ~	Main Size :	6	~
Size :	Line To Use :	~	Area :		_
~	From : V-00	~	To :		~
End Type :	Insul. Thick : 0 Additioning Property		Color By :	<u> </u>	
Select Nozzle :	Property	Value			
	¢			_	2
	Edit Column	-		Create	ancel

The **Nozzle Component** task pane is updated with the new tag and it will list in the dropdown menu **Tag**.

- **9.** Set the parameters of the nozzle. Select the values on the Nozzle Component task pane that are not yet defined.
 - End Type dropdown menu: Select FL (Flange connection).
 - Selected Nozzle list: Select NON STANDARD 150LB FLAT FACE NOZZLE.



10. Click **Nozzle Component** pane > **Positions** tab.

Nozzle Component Nozzle Data Positions									
	?								
Nozzle Data Positions	✓ ×								
Parameter	^								
Radian									

The task pane **Positions** is displayed for adding and controlling the position of the nozzle in the vessel.

The nozzle model can be place in different types of surfaces and it is controlled from the dropdown menu **Parameter** on the **Positions** tab.

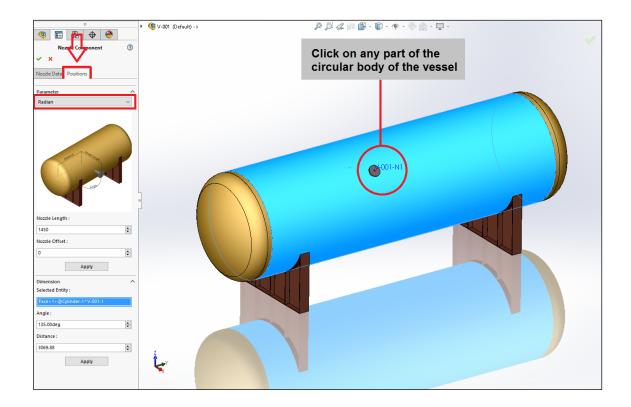
Parameter	Description
Radian	Circular surfaces
Flat	Flat surfaces
Coordinate	Point references and circular edges
Line	Along a line sketch entity
Head	Dome surface

By selecting any of the parameters from the **Parameter** menu, the **Positions** pane dynamically updates based on the parameter the user wants to access, displaying the fields and commands accordangly.

11. Select Parameter menu > Radian.

12. Place the nozzle in the vessel.

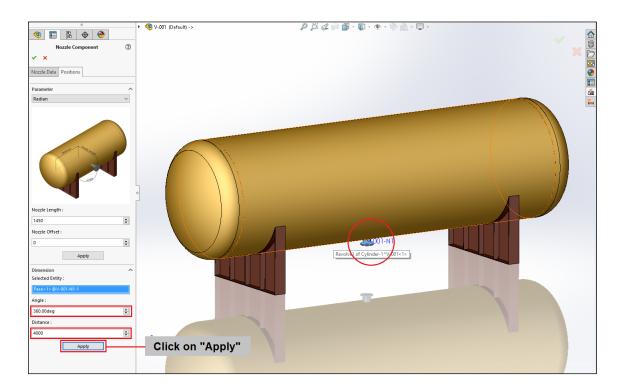
Click on any part of the circular body of the vessel, and the nozzle will be automatically inserted.



13. On the Dimension area on Position tab, set the parameters:

- Angle field: Type 360.
- **Distance** field: Type **4000**.

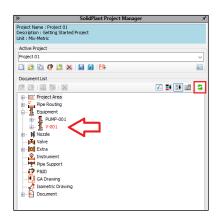
14. Click **Apply** to update the position of the nozzle.



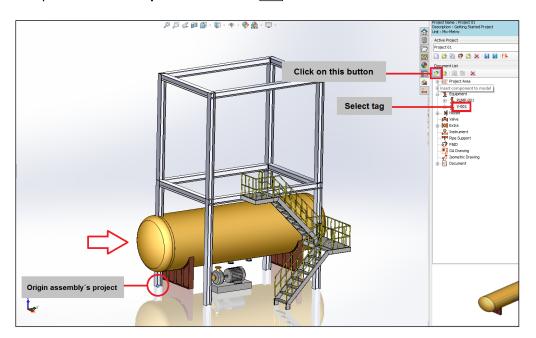
- **15.** Do the same for **V-001-N2** nozzle with the same values as **V-001-N1** on **Nozzle Data** tab. On the **Dimension** area on **Position** tab, set the parameters on below:
 - Angle field: Type **360**.
 - **Distance** field: Type **5000**.
- **16.** Click **Ok v** when finished.
- 17. Save on Standard toolbar.
- 18. Close the vessel assembly.

Back to the project, on the **Document List** node tree is displayed the tag **V-001**.

19. Click **Refresh** if the tag is not displayed.



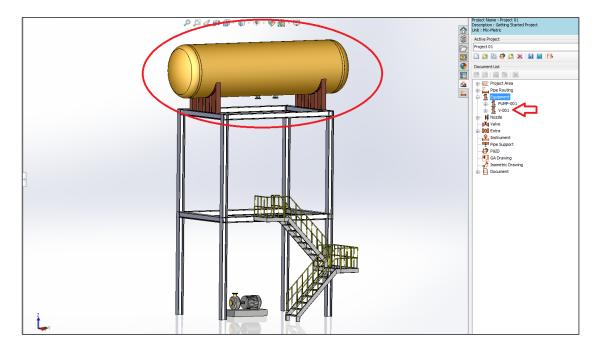
20. Click on the tag **V-001** on the **Document List** node tree and click **SolidPlant Project Manager** task pane > **Insert component to model** .



The vessel is placed automatically on the origin point of the project assembly.

21. Place the vessel at the top of the structure using SOLIDWORKS mates.

22. Click **Refresh** to update the status of the tag **V-001** on the node tree.



23. Click Save on Standard toolbar.

Routing

SolidPlant 3D provides the necessary tools to create intelligent pipe routes driven by a piping specification. Depending on the piping components in the piping specification assigned to the pipe line in use, the components will be generated automatically from the route that has been created.

The routing features are mainly located in the **Routing** and **Component** tabs, and in the side task pane **SolidPlant Pipe Routing**.

Creating a pipe line

In SolidPlant the pipe routes are special subassemblies of SOLIDWORKS that build a path of pipes, tubes or ducts with their piping components based on a 3D sketch of the center line of the route.

When you create a route with SolidPlant 3D, a route subassembly is created into the assembly project. The tag assigned to the route is the name of the subassembly. Unlike other types of subassemblies of SOLIDWORKS, you do not create a route assembly in its own window and then insert it as a component in the higher-level assembly.

SolidPlant uses functionality available in SOLIDWORKS and the user needs to be familiar with 3D sketching, assemblies, top-down design and mates, in order to become proficient.



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SolidPlant 3D is a database driven system and the piping specifications assigned to the route will define the components generated into the pipe subassembly. SolidPlant 3D doesn't use libraries or configurations in routing, so that the user doesn't need to care about preparing the 3D components before designing. All the information is contained into the piping specification list.

The piping specification list or pipe class is a collection of most compatible components considering dimensional and material properties for the intended service over a range on pressure and temperature specified. It is a list for use within its confines and which contains the definition of pipe and all related components.

SolidPlant 3D comes with **SpecCreator** that is the application to build up the piping specification list based on a huge range of catalogues of components. These catalogues contain most of the standards of the industry (ANSI-ASME, DIN, JIS, etc.) as well as a wide range of manufacturers (Swagelok, Bonney Forge, George Fisher, etc.). We won't cover in this document this application so that the user will use the piping specification **Demo** that automatically is assigned by default to any newly created project.

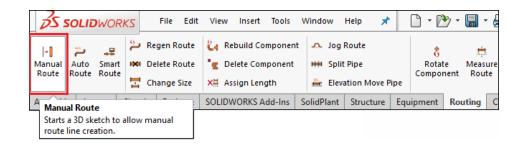
There are three methods to route a line with SolidPlant 3D. In this document we will learn how to use two of them, **Manual Route** and **Auto Route**.

Method	Description
Manual Route	The user manually sketch the path of the pipe line
Auto Route	Automatic generation of the pipe line path based on the selection of two reference points (e.g. From Nozzle, To Nozzle)
Smart Route	Automatic generation of the pipe line based on the information imported from a P&ID or .xls file into the database of the project

Manual Route method

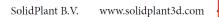
To create a pipe route:

1. Click Routing Tab > Manual Route.



The **Tag property** dialog box will display in order to define the parameters of the pipe route.

As the user has already realized, the **Tag property** dialog box is also used to define the properties of the equipment and their nozzles.



Field	Description
Group	Displays the type of the component
TAG	Unique piping identification
Description	Extra information to describe the route
Service/Type	Types of fluids. You can select from the dropdown menu or type a value
Specification	Dropdown menu listing the piping specifications available in the project
Main Size	Dropdown menu listing the sizes available from the selected piping specification
Line To Use	Disabled in pipe routing. It is used to assign a component to a pipe line
Area	Dropdown menu listing the areas available in which a pipe line can be grouped
From	Dropdown menu listing all the lines, equipment and equipment nozzle tags available where the pipe starts
То	Dropdown menu listing all the lines, equipment and equipment nozzle tags available where the pipe ends
Insul. Thick	Insulation thickness. You can type a number in millimeters
Color By	Dropdown menu listing the different ways to define the color of the pipe line
Additioning	A collection of fields available for adding further information to the pipe
Property	line which can be extended by the user

- 2. From the window TAG Property:
 - Tag field: Type PIP-001
 - Specification dropdown menu: Select Demo.
 - Description field: Type Getting Started Pipe 001 (Optional).
 - Main Size dropdown menu: Select 6 inches.
- 3. Click Create.

TAGP	roperty					x
IAO P	openy					*
Grou	р:	Pipe Routing	~	TAG :	PIP-001	~
Descr	iption :	Getting Start Pip	e 001			
Servio	:e/Type :		~	Comp Type :		~
Speci	fication :	Demo	~	Main Size :	6	~
Line T	To Use :		~	Area :		~
From	:		~	To:		~
Insul.	Thick :	0		Color By :		~
Add	itioning Prop	perty				
	Property		Value			^
•	MATERIAL					
	SCHEDULE					
	RATING					
	REVISION					
	PAINTING_	SPEC				
	PIPELINE_TE	EMP				
	TRACING_S	SPEC				
	SURFACE_A	REA				
	PIPELINE_T	YPE				×
<						>
Edit	t Column				Create	Cancel

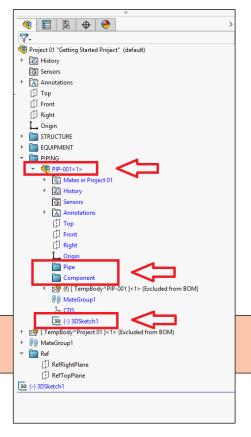


On creating the pipe line the user is automatically on **3D Sketch** mode and the tool **Line** is active. On the Feature Manager of SOLIDWORKS, the subassembly PIP-001 with the name of the tag is created and the sketch line of the route will be active, 3DSketch1.

The pipe subassembly contains two folders, Pipe and Component where it will store the piping components.

Folder	Description
Pipe	Contains the parts of the pipe sections of the route
Component	Contains the component parts of the route (Flanges, valves, elbows, etc.)

Once a line is sketched, the 3DSketch1 will be renamed to Main. If a line is not created, the **3D Sketch** won't be created.



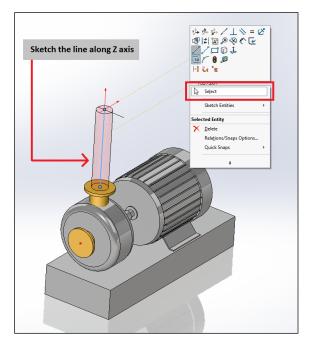
- 4. Hide the folder Structure for better performance and clearer environment.
- 5. Sketch a line.

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Sketch a line vertically along Z axis from the reference point of the nozzle pump **PUMP-001-N2**. The length is not important.

A preview of a pipe will display along the line to highlight the route.

Right-click in the graphic area and select
 Select from the shortcut menu to confirm the line.



SolidPlant 3D will display the **Select Component** window listing all the components available in the piping specification **Demo** for the current size of the pipe line, which is **6** inches. The user can select the components according to his needs for the referring pipe.

SolidPlant 3D is an intelligent system and it recognizes the nozzle point connection so that it will display the **Select Component** window for the needed components depending on the properties of the nozzle (Flange, gasket and bolts).

7. Select a pipe component.

When prompted, select the pipe component available from the **Demo** piping specification from **6** inches size. Select the component shown on the picture below and click **Create**.

ec. : Demo	~	🔵 Use Trii	m 🔿 Use	Bend 🔘	Use Miter		Custom size :		
DESCRIPTION		MAIN S	CLASS	GTVPF	SCHED	MATERI	CATALOG	PIPE O	PI /
PIPE, SCH 10		6	PIPE	PIP	10		ASME/ANSI B36.10	168.275	16
CHE CTANDARD DIDE CHILC CTA	NUCCO CTERI	<i>c</i>	DIDC	DID	100	<i>cc</i>	ACME CTANDARD	100.075	1/
PIPE, SCH 120		6	PIPE	PIP	120		ASME/ANSI B36.10	168.275	15
PIPE, SCH 160		6	PIPE	PIP	160		ASME/ANSI B36.10	168.275	15
PIPE, SCH 40		6	PIPE	PIP	40		ASME/ANSI B36.10	168.275	15
ASME STANDARD PIPE SMLS STAI	NLESS STEEL	6	PIPE	PIP	40S	SS	ASME STANDARD	168.275	15
PIPE, SCH 5		6	PIPE	PIP	5		ASME/ANSI B36.10	168.275	16
ASME STANDARD PIPE SMLS STAI	NLESS STEEL	6	PIPE	PIP	55	SS	ASME STANDARD	168.275	16
PIPE, SCH 80		6	PIPE	PIP	80		ASME/ANSI B36.10	168.275	14
	NI ECC CTEEL	6	DIDE	DID	805	cc		168 275	*
	ITEM DESC MAT LON SHO PIEC CATA MAII	femplate : C [_CODE : PII CRIPTION : I GRADE : G_DESCR : RT_DESC : I E_MARK : P .LOG : ASM N_SIZE : 6 _SIZE :	P6 PIPE, SCH 1 PIPE, SCH 1 PIPE IP6	0 0	ant\Pipe Rou	iting 2016\C	efault\PIPE\PIPE.SLDP	RT	

8. Select a flange component.

When prompted, select the flange component shown on the picture below and click Create.

Select Component : FLANGE , Main Size : 6, Run	Size : 6, Rat	ing : 150LB						
pec.: Demo 🗸 🔿	Use Trim	🔵 Use Ben	d 🔿 Use	Miter	🗌 Cust	tom size :		~
DESCRIPTION	MAIN_S	END_C	END_C	CLASS_NAME	GTYPE	STYPE	RATING	MA ^
LADISH 150LB RING JOINT WELD NECK FLANGE	6	FL	BW	FLANGE_WNK	FLG	WNK	150LB	SS
LADISH 150LB RAISED FACE WELD NECK FLANGE	6	FL	BW	FLANGE_WNK	FLG	WNK	150LB	SS
LADISH 150LB FLAT FACE WELD NECK FLANGE	6	FL	BW	FLANGE_WNK	FLG	WNK	150LB	SS
CRANE 150LB FLAT FACE WELD NECK FLANGE	6	FL	BW	FLANGE_WNK	FLG	WNK	150LB	SS
WELD NECK FLANGE, 150LB, RF	6	FL	BW	FLANGE_WNK	FLG	WNK	150LB	
CRANE 150LB RING JOINT WELD NECK FLANGE	6	FL	BW	FLANGE_WNK	FLG	WNK	150LB	SS
WELD NECK FLANGE, 150LB, RTJ	6	FL	BW	FLANGE_WNK	FLG	WNK	150LB	
	<u>.</u>		2111	FLANCE_MAIN	516	Man.	10010	- 22
WELD NECK FLANGE, 150LB, FF	6	FL	BW	FLANGE_WNK	FLG	WNK	150LB	
< Contract of the second se								>
ITEM_CC DESCRIF MAT_GR LONG_D SHORTJ PIECE_M	DE : FLGWI MION : WEL ADE : SESCR : WEL DESC : WEL IARK : FLGW 5 : ASME/AM ZE : 6	NK6 D NECK FLA D NECK FLA D NECK FLA /NK6	NGE, 150LB,		6\Default	\FLANGE\FL	ANGE_WNK.	SLDPRT
User Custom							Create	Cancel



9. Select a gasket component.

When prompted, select the component shown on the picture below and click **Create**.

Select Component : GASKET , Main Size : 6,	Rating : 150LB							
Spec. : Demo 🗸	🔵 Use Trim 🛛) Use Bend	🔿 Use Mi	ter	🗌 Cu	stom size :		~
DESCRIPTION		MAIN_S	END_C	CLASS	GTYPE	RATING	MATERI	CATAL ^
ASME STANDARD 150LB R-43 RING OCTAGON	NAL GASKET	6	FL	GASKET	GAS	150LB	SS	ASME
ASME STANDARD 150LB RAISED FACE NONM	ETALIC GASKET	6	FL	GASKET	GAS	150LB	NEO	ASME
ACHIE CTUNDARD ACOUD RAISED FACE CRIRAL		6	- CL	CACHER	616	10010		10110
GASKET, 150LB		6	FL	GASKET	GAS	150LB		ASME
GASKET, ISOLD		0	16	GADKET	GHD	IJULD		ADIVIL
ASME STANDARD 150LB FLAT FACE NONMETA	ALIC GASKET	6	FL	GASKET	GAS	150LB	NEO	ASME
ASME STANDARD 150LB FLAT FACE JACKETED	GASKET	6	FL	GASKET	GAS	150LB	SS	ASME
ASME STANDARD 150LB RAISED FACE JACKET	6	FL	GASKET	GAS	150LB	SS	ASME	
GASKET, 150LB, NEO		6	FL	GASKET	GAS	150LB	NEO	ASME
CASKET 1501B		6	FI	CASEET	CV6	150LB		A CRAF
11EN DES MAX LOP SHU PIEC CAT MA	Template : C:\Prog M_CODE : GAS6 SCRIPTION : GASKE T_GRADE : NG_DESCR : GASKE SRT_DESC : GASKE CE_MARK : GAS6 ALOG : ASME/ANS IN_SIZE : 6 N_SIZE :	T, 150LB T, 150LB T	olidPlant\Pip	e Routing á	2016\Defau	It\GASKET\G,	ASKET.SLDPR	Т
User Custom						E	Create	Cancel

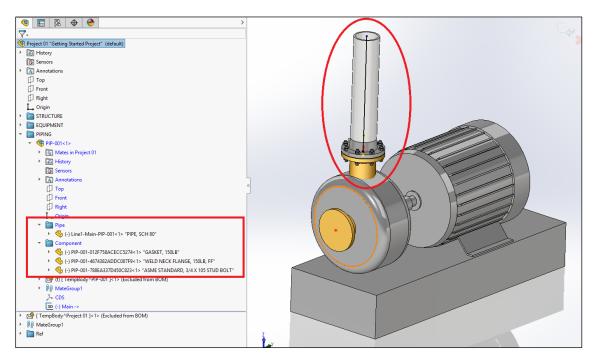
10. Select bolts component.

When prompted, select the component shown on the next picture and click on **Create** button.

ec. : Demo	~	🔵 Use Tri	im 🕜 Use I	Bend 🔘	Use Miter	[Custom s	size :	
ESCRIPTION		MAIN S	CLASS	GTYPE	STYPE	RATING	MATERI	CATALOG	FAC
SME STANDARD, 3/4 X 105 STU	D BOLT	6	BOLT	BOLT		150LB	SS	ASME STANDARD	FF
SME STANDARD, 3/4 X TTS STU	D BOLI	b	BOLI	BOLI		150LB	22	ASME STANDARD	КIJ
SME STANDARD, 3/4 X 105 STU	D BOLT	6	BOLT	BOLT		150LB	SS	ASME STANDARD	RF
STM A307, 3/4 X 115 STUD BOI	T	6	BOLT	BOLT		150LB		ASTM A307	RTJ
STM A307, 3/4 X 105 STUD BOI	T	6	BOLT	BOLT		150LB		ASTM A307	FF
STM A307, 3/4 X 105 STUD BOI	Ţ	6	BOLT	BOLT		150LB		ASTM A307	RF
SME STANDARD, 3/4 X 100 MA	CHINE BOLT	6	LUG_BOLT	BOLT	LUG	150LB	SS	ASME STANDARD	RTJ
SME STANDARD, 3/4 X 85 MAC	HINE BOLT	6	LUG_BOLT	BOLT	LUG	150LB	SS	ASME STANDARD	FF
SME STANDARD, 3/4 X 85 MAC	HINE BOLT	6	LUG_BOLT	BOLT	LUG	150LB	SS	ASME STANDARD	RF
CTM A307 3/4 Y 85 MACHINE	ROIT	6	THE BOIT	ROIT	1116	1501 B		ASTM A207	FF
	ITEN DES MA LON SHO PIE CAT MA	M_CODE : B SCRIPTION : T_GRADE : NG_DESCR : DRT_DESC : CE_MARK : F	OLT6 ASME STANE ASME STANE STUD BOLT)ARD, 3/4))ARD, 3/4)	(105 STUD B	OLT	Default\BOL	n.Bolt.Sldprt	

P Double-click on the selected component in the Select Component window will also confirm the component instead of clicking Create button.

The pipe line will be created with all the piping components added automatically. On the FeatureManager on the left hand side, the pipe section parts will be contained into the Pipe folder, and the rest of the piping component parts will be on the Component folder.



The user can mark and enable the check box **Disable Routing Mode** in the **SolidPlant Pipe Routing** task pane, in order to disable the automatic generation of the 3D piping components, so that the user can easily draw the path without any interferences.

»	SolidPlant Pipe Routing
Project Name : Pr Description : Gett Unit : Mix-Metric	roject 01 ing Started Project
Active Route	
PIP-001	~
Piping Preference	e
Specification :	Demo ~
Pipe Size :	6 v
Insulate	Thick :
🖂 Lise data fro	m setting 🔄 Hide ISO-Attachment
🗹 Disable Rout	ing Mode 🛛 💽 ase custom model
Regen fitting	g every rebuild 🔲 Use realistic model
Add bolt by (default 🗌 Use symbolic model
Piping Sketch To	bol TX YT. T. PA

11. Mark the check box Disable Routing Mode on the SolidPlant Pipe Routing pane.

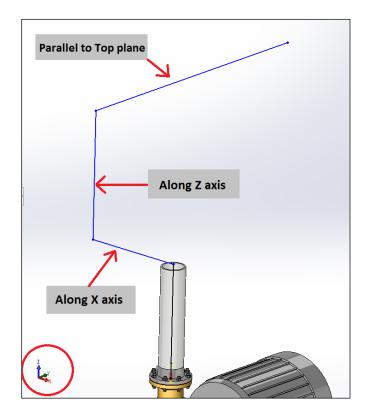


40

12. On the CommandManager, click **Sketch** tab > **Line** \checkmark .

13. Sketch a line.

Sketch a path from the end point of the previous pipe line and draw a path as shown on picture below. The length is not important.



14. Uncheck **SolidPlant Pipe Routing > Piping Reference > Disable Routing Mode**.

1	×	» SolidPlant Pipe Routing
		Project Name : Project 01 Description : Getting Started Project Unit : Mix-Metric
	1	Active Route
5	D	PIP-001
	27	
	•	Piping Preference
	E	Specification : Demo
	0ă	Pipe Size : 6
	-	Insulate Thick :
		Use data from cotting
		Disable Routing Mode Use custom model
	1	Regen fitting every rebuild 🗌 Use realistic model
		Add bolt by default
		Piping Sketch Tool
		Snap: ⊙] ‡ 1/→ 1/2 · 1/2 · /2

Now the automatic generation of the piping components is enabled again for generating the rest of the route.

15. Select elbow component.

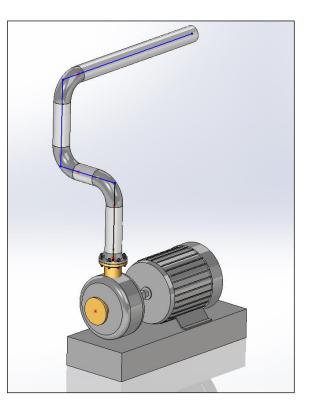
When prompted, select the component shown on the next picture and click **Create**.

ec.: Demo	Use Trim 🔘 Use Bend	I 🔵 Use Miter		Custon	n size :	
DESCRIPTION		MAIN_S	RUN_SI	END_C	END_C	CLASS_NAME
0 DEGREE BASE ELBOW, 15	i0 LB	6		FL		EB_90_BASE
ADISH 90 DEGREE THREE R	ADIUS BUTTWELD ELBOW 40S	6		BW		ELBOW_3R_90
ADISH 90 DEGREE THREE R	ADIUS BUTTWELD ELBOW 805	6		BW		ELBOW_3R_90
0 DEG. ELBOW, 125LB		6	6	SCF		ELBOW_LR_90
0 DEG. ELBOW, 150LB		6	6	SCF		ELBOW_LR_90
0.055.510.000,05010		6	4	665		51001110.00
0 DEG. LR ELBOW, SCH 10,	BW	6		BW		ELBOW_LR_90
ATLOK FORGE 90 DEGREE L	ONG RADIOS WELD ELBOW SCHEDULE 105	0		DW		ELBOW_LK_9C
VS 90 DEGREE LONG RADIU	6		BW		ELBOW_LR_90	
OUTCLAS REOS ON DECREE	SMOOTH FLOW/ RUITW/FLD FLROW/ 105	6		R\M		FIROW ID OF
8	File Template : C:\Program Files\S ITEM_CODE : 90LR6 DESCRIPTION : 90 DEG. IR ELBO' MAT_GRADE : LONG_DESCR : 90 DEG. IR ELBO' SHORT_DESC : ELBOW 90 DEG. I PIECE_MARK : 90LIR6 CATALOG : ASME/ANSI B16.9 MAIN_SIZE : 6	W, SCH 10 , BW W, SCH 10 , BW	outing 2016	\Default\ELF	BOWAELBOV	V_90.SLDPRT

SolidPlant 3D is sensitive of the path that is routed by the user.

It recognizes that on the point connection between two perpendicular lines a 90 degrees elbow is required.

Therefore it will display the **Select Component** window for the missing components and it will automatically generate the rest of the components that were used previously.

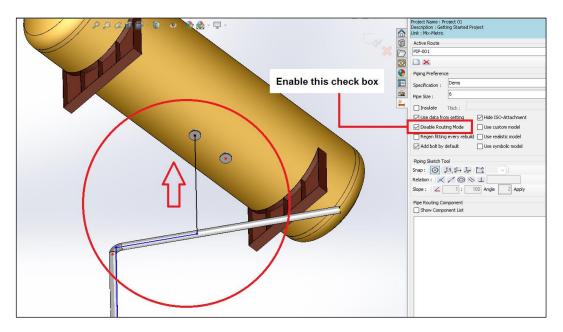




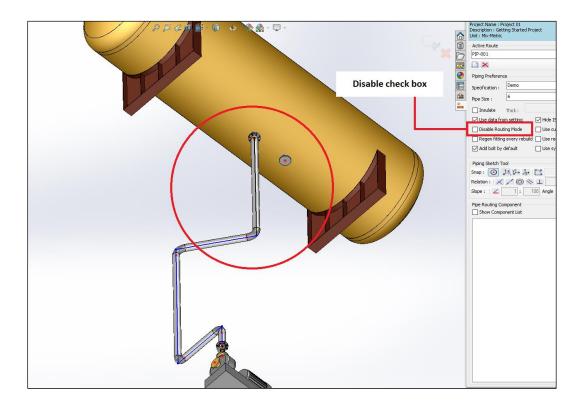
- **16.** Mark **SolidPlant Pipe Routing > Piping Reference > Disable Routing Mode** check box.
- **17.** On the CommandManager, click **Sketch** tab > **Line** \checkmark .

18. Sketch a line.

From the end point of the pipe line, sketch a line vertically along Z axis up to the vessel nozzle **V-001-N1**.



19. Uncheck **SolidPlant Pipe Routing > Piping Reference > Disable Routing Mode**.



SolidPlant 3D will generate all the piping components of the route with the same components used previously.

If any line of the path is toggled For construction in the PropertyManager, SolidPlant
 3D will remove the pipe section as it doesn't recognize Centerlines as part of the pipe path. The user will be able to use the Centerline tool for referencing purposes.

- 20. Exit from 3D Sketch mode.
- 21. Exit from Edit Assembly mode.
- 22. Save on Standard toolbar.

Auto Route method

To create a pipe route:

1. Click SolidPlant Pipe Routing Pane > New pipe model

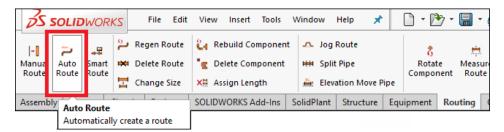
	» SolidPlant Pipe Routing ★
	Project Name : Project 01 Description : Getting Started Project Unit : Mix-Metric
0	Active Route
D	× I
23	
٠	Piping Preference
	New pipe model Specification :
10m	Pipe Size :
1	Insulate Thick:
	Use data from setting ISO-Attachment
	Disable Routing Mode Use custom model
	Regen fitting every rebuild Use realistic model
	Add bolt by default Use symbolic model
	Piping Sketch Tool
	Snap: ⊙ J≭, 1→ J→ 1 \
	Relation : 📈 🖊 🔘 📎 🔟
	Slope : Z 1 : 100 Angle 2 Apply
	Pipe Routing Component
	Show Component List

- 2. From the window TAG Property:
 - Tag field: Type PIP-002
 - **Specification** dropdown menu: Select **Demo**.
 - Description field: Getting Started Pipe 001 (Optional).
 - Size dropdown menu: Select 6 inches.
- 3. Click Create.

On creating the pipe line the user is automatically on **3D Sketch** mode **3** and the tool **Line** is active.

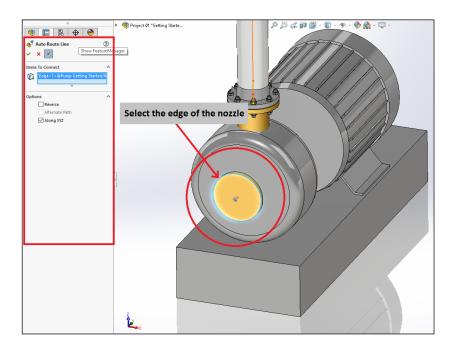


4. Click Routing Tab > Auto Route.

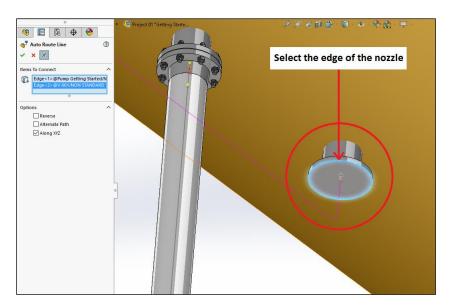


The task pane Auto Route Line is displayed on the left hand side on the PropertyManager.

5. Select the edge of the pump nozzle PUMP-001-N1.



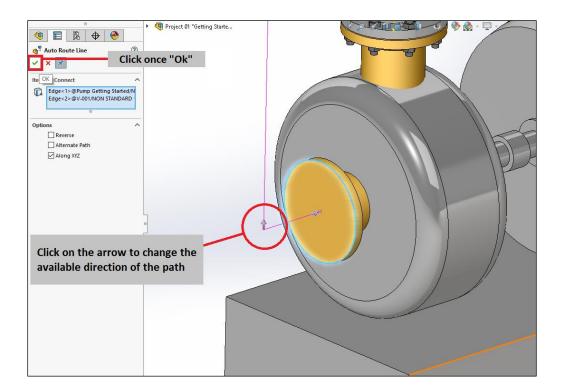
6. Select the edge of the vessel nozzle V-001-N2.



SolidPlant B.V.

A preview of the path between the two nozzles is displayed. Along the path are available arrows to alternate the direction of the current route.

7. Click just once **Auto Route Line** pane > **Ok** \checkmark when the path is finished.



8. Mark the check box **Disable Routing Mode** on the **SolidPlant Pip Routing** pane.

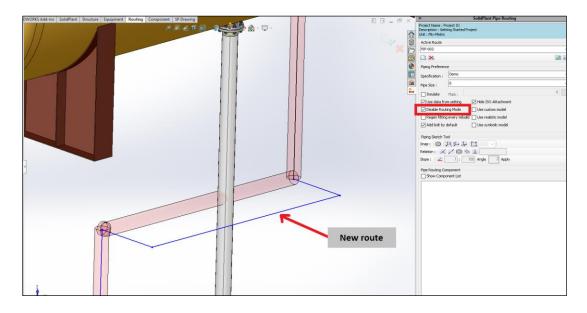
»	SolidPlant Pipe Routing	×
Project Name : Project Description : Getting S Unit : Mix-Metric		
Active Route		
PIP-002		~
Piping Preference		
Specification : Der	mo	~
Pipe Size : 6		\sim
Insulate Thio	ck : 0	
🖂 Use data from se	atting 🛛 Hide ISO-Attachment	
🗹 Disable Routing M	ModeUse custom mo el	
Regen fitting eve	ery rebuild 🔲 Use realistic model	
Add bolt by defa	ult Use symbolic model	
Piping Sketch Tool		
Snap : 💽 斗	1 I I I I I I I I I I I I I I I I I I I	
Relation : 📈 🦯	$O \wedge \bot$	

9. Click **Auto Route Line** pane > **Ok**

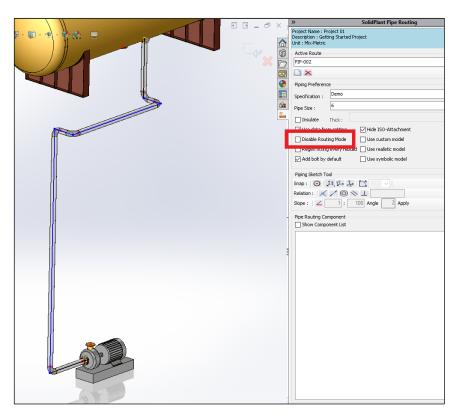
The user can change the route by adding or deleting new lines without generating the 3D piping components.

10. Sketch a line.

Sketch the path as shown on below picture to avoid the existing pipe **PIP-001**, by adding or deleting the necessary lines.



11. Uncheck **SolidPlant Pipe Routing > Piping Reference > Disable Routing Mode**.



12. When prompted **Select Component** windows the user can select the same components as **PIP-001** for Pipe, Flange, Gasket, Bolt and Elbow.

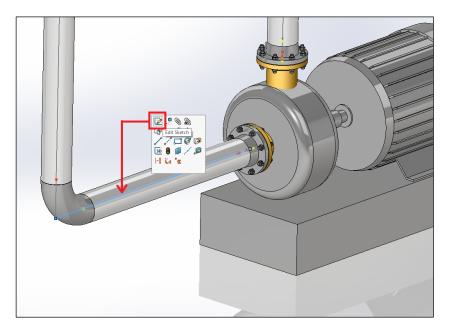
- 13. Exit from 3D Sketch mode.
- 14. Exit Edit Assembly mode.
- 15. Save the project.

Adding components

Valve component

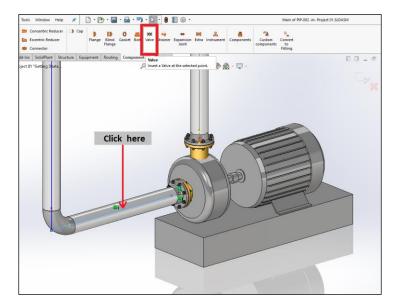
1. Make active the route.

Right-click on the sketch line of **PIP-002** shown on picture below and select **Edit Sketch** from the shortcut menu.



The route will be on **3D Sketch** mode ready to be edited.

- Just click on any point of the sketch line (recommended around the middle).
- **3.** Click **Component** Tab > **Valve**.

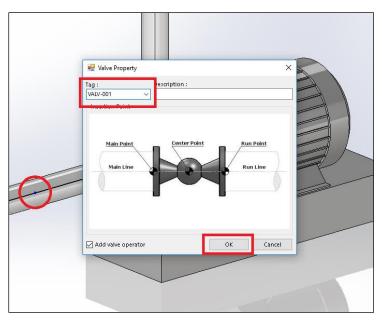


4. When prompted, select Selection Point option and click Apply.

🖳 Insert Point		×
Reference O Start Point	Selection Point	🔿 Valve Prope
Cancel		Apply

A point will be added where the user clicked on previously and it will be the middle reference point of the valve. The dialog box **Valve Property** will be displayed in order to define the tag of the valve.

- When prompted, type VALV-001 on the Tag field.
- 6. Click Ok.



7. Select valve component.

When prompted, select the component shown on the next picture and click Create.

Demo	✓ O Use Trim O Use Bend O Use Miter	🗌 Cus	tom size :		
DESCRIPTION		MAIN_S	BRAN	END_C	CLASS *
300LB FLAT FACE FULL OPEN	VING GATE VALVE O.S.& Y.	6		FL	VALVE
300LB RAISED FACE FULL OI	PENING GATE VALVE O.S.& Y.	6		FL	VALVE
CRANE 300LB RAISED FACE	BOLTED BONNET GATE VALVE	6		FL	VALVE
CIVANE SUCED FEATFACE DO		•		16	VALVE
GATE VALVE, 300LB, RF		6		FL	VALVE
		, in the second s			
	ENING GATE VALVE O.S.& Y.	6		FL	VALVE
GATE VALVE, 400LB, RF 600LB RAISED FACE FULL OPENING GATE VALVE O.S.& Y.				FI	VALVE
CHODT DATTEDNI GATE VALVE		6		FI	VALVE
c					>
	File Template: C:\Program Files\SolidPlant\Pipe Ror ITEM_CODE: DESCRIPTION : MAT_GRADE : LONG_DESCR : 600LB BUTTWELD GLOBE VALVE LOI SHORT_DESC : GLOBE VALVE PIECE_MARK : VALVGLB6 CATALOG : ANSI B16.10-1986 MAIN_SIZE : 6		(VALVE(VAL)	/E_GLOBE.S	LUPRI



8. Select flange component.

When prompted, select the component shown on the next picture and click Create.

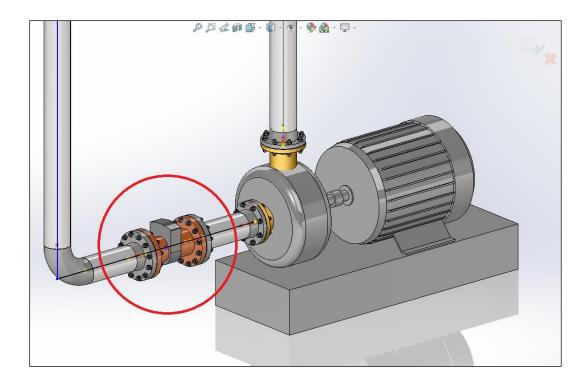
ec.: Demo	<u> </u>	Jse Trim) Use Bend	🕖 Use N	Aiter	Custor	m size :		~
DESCRIPTION		MAIN_S	END_C	END_C	CLASS_NAME	GTYPE	STYPE	RATING	M
RANE 2500LB RAISED FACE W	ELD NECK FLANGE	6	FL	BW	FLANGE_WNK	FLG	WNK	2500LB	55
VELD NECK FLANGE, 2500LB, F	RF	6	FL	BW	FLANGE_WNK	FLG	WNK	2500LB	
ADISH 2500ED KAISED FACE V	VELD NECK FLANGE	0	FL.	DW	FLANGE_WINK	FLG	WINK	ZOUULD	32
VELD NECK FLANGE, 300LB, RI	F	6	FL	BW	FLANGE_WNK	FLG	WNK	300LB	
		4	F 1	2111	FLANCE WAR	510		20010	- 66
ADISH 300LB RAISED FACE WI	6	FL	BW	FLANGE_WNK	FLG	WNK	300LB	\$5	
VELD NECK FLANGE, 400LB, RI	6	FL	BW	FLANGE_WNK	FLG	WNK	400LB		
ADISH 400LB RAISED FACE WI	6	FL	BW	FLANGE_WNK	FLG	WNK	400LB	55	
RANE 400LB RAISED FACE WE	6	FL	BW	FLANGE_WNK	FLG	WNK	400LB	S 5	
ADISH GOOLB DAISED FACE WI	FLD NECK FLANCE	6	FI	R\M/	FLANCE WAR	FLC	WNE	6001 R	•••
									>
0	ITEM_CO DESCRIPI MAT_GR4 LONG_DI SHORT_D PIECE_M/	DE : FLGWN FION : WELD DE : ESCR : WELD ESCC : WELD ARK : FLGWP : ASME/ANS ZE : 6	NECK FLAN NECK FLAN NECK FLAN NECK FLAN	IGE, 300LB, I IGE, 300LB,		,	1	•	

9. Select bolt component.

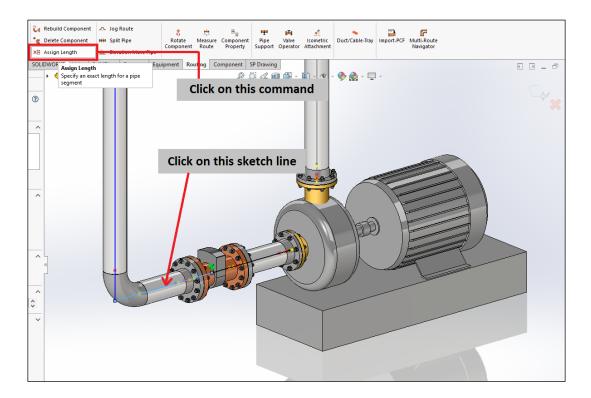
When prompted, select the component shown on the next picture and click Create.

ec.: Demo	~	🔵 Use Tr	im 🔵 Use I	Bend 🔘	Use Miter		Custom s	ize :	
ECONTION .		MAINER	CLASS	CTVDE	CTVDE	DATING	MATERI	CATHLOC	FAC
SME STANDARD, 3/4 X 125 STUD BO	IT	6	BOLT	BOLT		300LB	SS	ASME STANDARD	FF
CME CTANDARD - 3/4 Y 140 CTUD RO		~	DOIT	DOIT		2001.0		ACME CTANDADD	DTI
SME STANDARD, 3/4 X 125 STUD BO	ιτ	6	BOLT	BOLT		300LB	SS	ASME STANDARD	RF
STM A307, 3/4 X 125 STUD BOLT	6	BOLT	BOLT		300LB		ASTM A307	FF	
STM A307, 3/4 X 140 STUD BOLT		6	BOLT	BOLT		300LB		ASTM A307	RTJ
STM A307, 3/4 X 125 STUD BOLT	6	BOLT	BOLT		300LB		ASTM A307	RF	
SME STANDARD, 3/4 X 110 MACHINI	6	LUG_BOLT	BOLT	LUG	300LB	SS	ASME STANDARD	FF	
SME STANDARD, 3/4 X 130 MACHINI	E BOLT	6	LUG_BOLT	BOLT	LUG	300LB	SS	ASME STANDARD	RTJ
SME STANDARD, 3/4 X 110 MACHINI	E BOLT	6	LUG_BOLT	BOLT	LUG	300LB	SS	ASME STANDARD	RF
STM AR07 R/A V 110 MACHINE BOIT		6	LUG BOIT	ROIT	106	3001 B		ASTM A307	DF
	ITEI DE: MA LOI SHI PIE CAT	M_CODE : B SCRIPTION : T_GRADE : NG_DESCR : DRT_DESC : CE_MARK : I	OLT6 ASME STANE ASME STANE STUD BOLT)ARD, 3/4))ARD, 3/4)	(125 STUD E	зоіт	Default\BOL	r\Bolt.Sldprt	

The valve is added automatically on the sketch line and its reference point is the middle point which has split the line into two sketch lines.



- **10.** Select the sketch line highlighted on the picture below.
- **11.** Click Routing Tab > Assign Length.



LILE

- 12. When prompted, type 500 mm on field Change pipe length to (mm).
- 13. Click Apply.

Assign Pipe Length	×
Change pipe lengt	th to (mm) : 500
Apply	Cancel

The pipe section will change to 500mm length automatically from the selected sketch line. The user doesn't need to calculate the dimension value to get the needed length for the section.

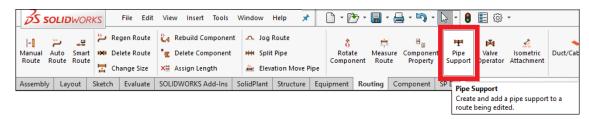
	Edge<1>@PIP-002/PIPE, SCH 10 Edge<2>@PIP-002/WEID NECK FLANGE, 300LB, RF
	Detta X: 9.19mm Detta X: 9.00,00m Detta Z: 7.44mm Total Length: 1057.3mm
	File: PP-002-3@Project 01/Line8-Main.PIP-002-1@PIP- 002 To: PIP-002-3@Project 01/PIP-002 5327287342CEF94-2@PIP-002 File: Project 01:SLDASM Config: default
12. 7.44mm	
	Pipe section length: 500mm

- 14. Exit from 3D Sketch mode.
- 15. Exit Edit Assembly mode.
- 16. Save on Standard toolbar.

Pipe Support component

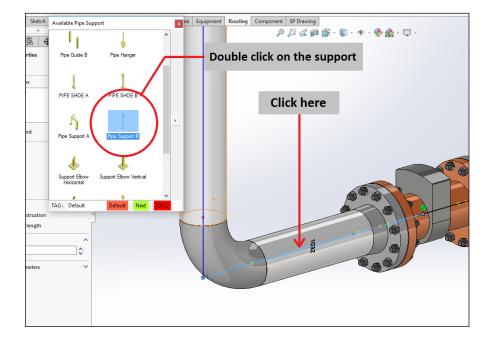
1. Right-click on any sketch line of **PIP-002** and from the shortcut menu click **Edit Sketch** with to make the line active.

2. Click Routing Tab > Pipe Support.

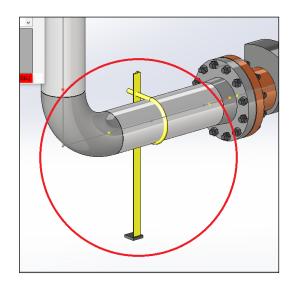


The window **Available Pipe Support** is displayed with the pipe supports available in the library of SolidPlant 3D.

- **3.** Just click on any point of the sketch line (recommended around the middle).
- 4. Double-click on the icon **Pipe Support B** on the **Available Pipe Support** window.

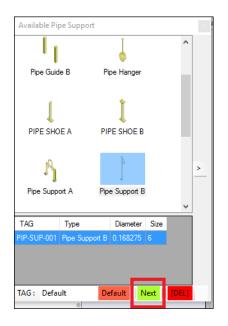


The pipe support is placed on the selected point on the pipe line. Notice that the support has been adjusted automatically to the outer diameter of the pipe.





5. Click Next on the Available Pipe Support window.



The **Editing Pipe Support** dialog box is displayed to set a number of parameters of the pipe support selected.

- 6. Set the dimension length.
 - Length field: Type 650.
 - Click Set Set .

Editting Pipe Support 1 Click on "Set" Pipe Support Modification Bot Modification Support Modification Bot Modification Adjustment Destremce Point Length Est Set Slope Set Angle 0.00 C Flip Slope Set Base Direction Set Set Slope Set Set Slope Set Set Slope Set Angle 0.00 C Flip Slope Set Set Slope Set Set Slope Set Set Slope Set	
--	--

- 7. Close Editing Pipe Support dialog box and then Available Pipe Support window.
- 8. Save on Standard toolbar.

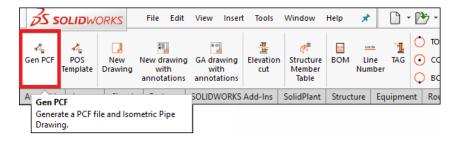


SolidPlant 3D has embedded Isogen[®] from Alias for the total automation of piping isometric drawing production and is the de facto standard CAD system for drawing piping isometrics. The user will eliminate the need to manually edit isometric drawings which significantly reduces fabrication and on-site construction errors.

Also SolidPlant 3D provides the tools needed to create professional GA Drawings, Plane Sections, etc.

Gen PCF command

1. Click Drawing Tab > Gen PCF.



- 2. From the window ISOGen will be listed all the pipe lines created on the project.
 - Select the check box PIP-002 from the list.
 - From the Drawing Style dropdown menu: Select Final-Cut-List.
 - Click PCF & Drawing PCF & Drawing .

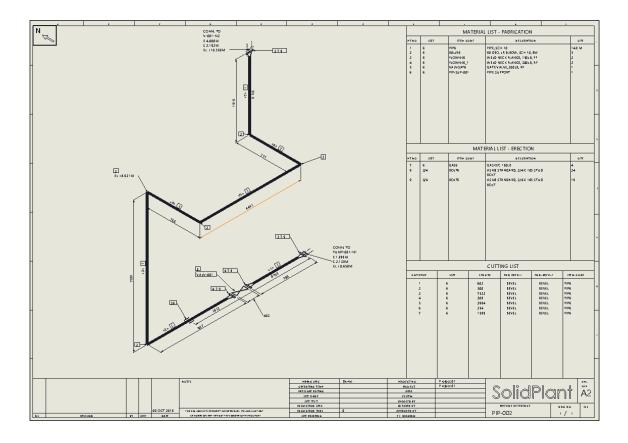
ISOGen		
File Configu	ration Tool Edit	
Generate Item :	Ref origin : X;0 : Y;0 : Z;0	Show all connected items Selected items
Item		
<		>
All done, pleas	e make selection Final-Cut-List v	PCF & Drawing PCF only Drawing only Import Close

SolidPlant 3D will generate three different types of documents from the pipe line selected.

Туре	Description
PCF	The Piping Component File is the primary input for personal ISOGEN.
	PCFs are text files containing the component and routing information.
DWG/DXF	CAD Drawing files.
SLDDRW	SOLIDWORKS Drawing Document.

The PCF can be directly imported into pipe stress analysis software like AutoPIPE, CAESER II or ROHR 2. By default, these documents are located in: C:\ProgramData\SolidPlant Projects\Project 01\Isometric Drawing.

SolidPlant 3D automatically will display the Isometric Drawing in **SLDDRW** document.



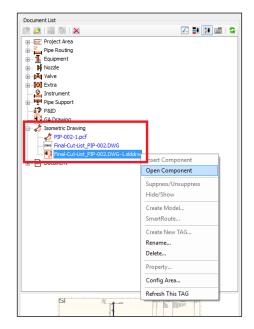
3. Close the SOLIDWORKS drawing of the piping isometric.

These documents will be listed into the **Document List** node tree on the **SolidPlant Project Manager** pane.

4. Click **Refresh** if not displayed.



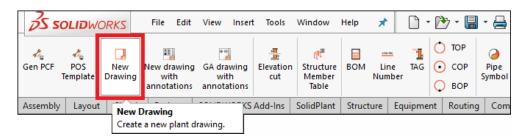
The user can access the files directly by right-click on the document in the **Document List** node tree and select **Open Component** from the shortcut menu.



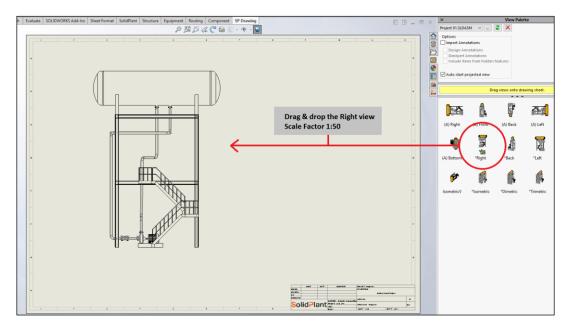
5. Save on Standard toolbar.

New Drawing command

1. Click Drawing Tab > New Drawing.

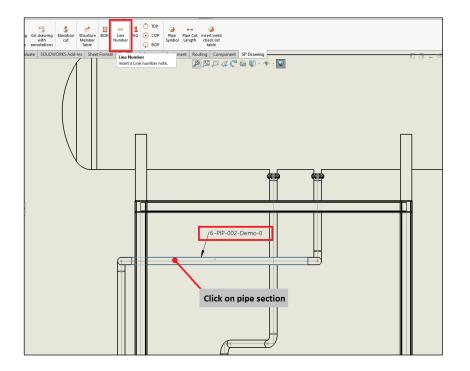


2. Drag and drop the right view of the plant from View Palette task pane.



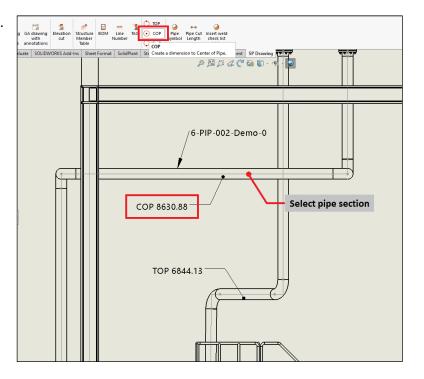


- **3.** Click on any pipe section.
- 4. Click Drawing Tab > Line Number command.



An annotation will be displayed on the pipe section.

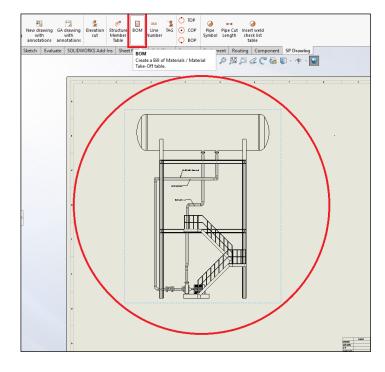
- 5. Click on any pipe section.
- Click on Drawing Tab > COP COP . The dimension annotation will be displayed between the center of the pipe and the ground level.
 - TOP: Top Of Pipe.
 - COP: Center Of Pipe.
 - BOP: Bottom Of Pipe.





BOM command

- Select the project view from the drawing.
- 2. Click Drawing Tab > BOM.



The bill of materials is displayed.

3. Click **To Excel** button to export to an .xls file.

Favorites		~ 0 X		Sorted E	ay :	~	Add Column
No.	Name	Description	Spec	Size	Quantities	_	
Run #	✓ ITEM_CODE	✓ DESCRIPTION ✓	SPEC	✓ MAIN_SIZE	✓ Quantity ✓	/	
PUMP-001			_				
1	NOZ6	NON STANDARD	Demo	6	2		
V-001	NOZ6	NON STANDARD	Dama	6	2		
PIP-001	NO26	NON SIANDARD	Demo	0	2		
3	90LLR6	90 DEG. LR ELB	Demo	6	4		
4	BOLT6		Demo	6	16		
5	FLGWNK6	WELD NECK FLA		6	2		
6	GAS6		Demo	6	2		
7	PIP6	PIPE, SCH 10	Demo	6	10852 mm		
Joint	BW-6				10		
Washer	Size = 6				32		
Nut	Size = 6				32		
PIP-002							
8	PIP-SUP-001	PIPE SUPPORT	Undefined	6	1		
9	90LLR6	90 DEG. LR ELB	Demo	6	5		
10	BOLT6	ASME STANDAR	Demo	6	24		
- 11	BOLT6	ASME STANDAR	Demo	6	16		
12	FLGWNK6	WELD NECK FLA		6	2		
13	FLGWNK6	WELD NECK FLA		6	2		Click on "To E
14	GAS6		Demo	6	4		
15	VALVGAT6	GATE VALVE, 300		6	1		
16	PIP6	PIPE, SCH 10	Demo	6	14771 mm		
Joint	BW-6				14		
Washer	Size = 6				80		
Nut	Size = 6				80		

The bill of materials is exported to an .xls file and it is automatically opened.

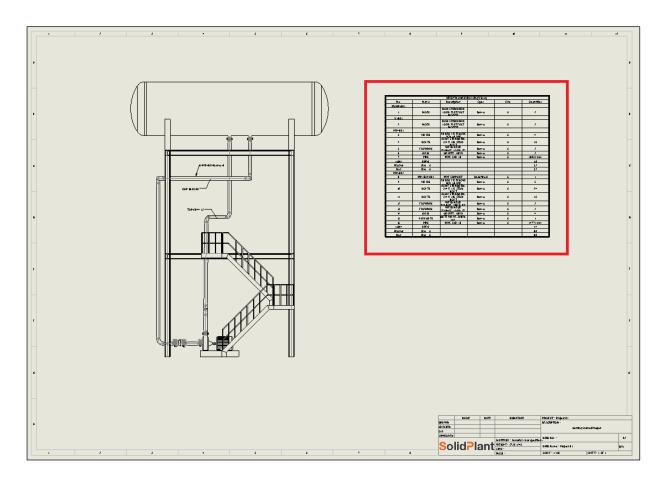
	А	В	С	D	E	F	G	Н
1								
3				AssemblyInfo : C:\ProgramData\SolidPlant Projects\Project 01\Drawing\Project 01.SLDASM				
4	No.	Name	Description	Spec	Size	Quantities		
5	PUMP-001							
6	1	NOZ6	NON STANDARD 150LB FLAT FACE NOZZLE	Demo	6	2		
7	V-001							
8	2	NOZ6	NON STANDARD 150LB FLAT FACE NOZZLE	Demo	6	2		
9	PIP-001							
10	3	90LLR6	90 DEG. LR ELBOW, SCH 10, BW	Demo	6	4		
11	4	BOLT6	ASME STANDARD, 3/4 X 105 STUD BOLT	Demo	6	16		
12	5	FLGWNK6	WELD NECK FLANGE, 150LB, FF	Demo	6	2		
13	6	GAS6	GASKET, 150LB	Demo	6	2		
14	7	PIP6	PIPE, SCH 10	Demo	6	10852 mm		
15	Joint	BW-6				10		
16	Washer	Size = 6				32		
17	Nut	Size = 6				32		
18	PIP-002							
19	8	PIP-SUP-001	PIPE SUPPORT	Undefined	6	1		
20	9	90LLR6	90 DEG. LR ELBOW, SCH 10, BW	Demo	6	5		
21	10	BOLT6	ASME STANDARD, 3/4 X 125 STUD BOLT	Demo	6	24		
22	11	BOLT6	ASME STANDARD, 3/4 X 105 STUD BOLT	Demo	6	16		
23	12	FLGWNK6	WELD NECK FLANGE, 300LB, RF	Demo	6	2		
24	13	FLGWNK6	WELD NECK FLANGE, 150LB, FF	Demo	6	2		
25	14	GAS6	GASKET, 150LB	Demo	6	4		
26	15	VALVGAT6	GATE VALVE, 300LB, RF	Demo	6	1		
27	16	PIP6	PIPE, SCH 10	Demo	6	14771 mm		
28	Joint	BW-6				14		
29	Washer	Size = 6				80		
30	Nut	Size = 6				80		
31								
32								
33								
34								

- 4. Close the .xls file.
- 5. Click **Close** to place the bill of materials into the drawing.

Favorites	`	× • •		Sorted b	y:	~	Add Column
No.	Name	Description	Spec	Size	Quantities		
Run #	✓ ITEM_CODE	✓ DESCRIPTION ✓	SPEC	✓ MAIN_SIZE	✓ Quantity	•	
PUMP-001							
- 1	NOZ6	NON STANDARD	Demo	6	2		
V-001							
2	NOZ6	NON STANDARD	Demo	6	2		
PIP-001							
3	90LLR6	90 DEG. LR ELB	Demo	6	4		
4	BOLT6	ASME STANDAR	Demo	6	16		
5	FLGWNK6	WELD NECK FLA	Demo	6	2		
6	GAS6	GASKET, 150LB	Demo	6	2		
7	PIP6	PIPE, SCH 10	Demo	6	10852 mm		
Joint	BW-6				10		
Washer	Size = 6				32		
Nut	Size = 6				32		
PIP-002							
8	PIP-SUP-001	PIPE SUPPORT	Undefined	6	1		
9	90LLR6	90 DEG. LR ELB	Demo	6	5		
10	BOLT6	ASME STANDAR	Demo	6	24		
- 11	BOLT6	ASME STANDAR	Demo	6	16		
12	FLGWNK6	WELD NECK FLA	Demo	6	2		Click on "Clo
13	FLGWNK6	WELD NECK FLA	Demo	6	2		chek off clo
14	GAS6	GASKET, 150LB	Demo	6	4		
15	VALVGAT6	GATE VALVE, 300	Demo	6	1		
16	PIP6	PIPE, SCH 10	Demo	6	14771 mm		
Joint	BW-6				14		
Washer	Size = 6				80		
Nut	Size = 6				80		



6. Move the bill of materials table wherever you want.



- 7. Close SOLIDWORKS Drawing.
- 8. Save on Standard toolbar.



Finding Answers and Instructions

- Video tutorials on the web : To view multimedia tutorials about using SolidPlant, go to www.solidplant3d.com.
- User guides on the web : To download illustrated PDF files with complete instructions for each applications, go to www.solidplant3d.com.
- SolidPlant support : For techical support, go to www.solidplant3d.com/contact.php.
- Product information : For general product information and updates, go to www.solidplant3d.com.

System requirements :

Microsoft Windows 7 (*64-bit recommended)
Intel or AMD with SSE2 support
8 GB minimum
5 GB minimum free hard disk space
1 GB Microsoft OpenGL or Direct3D capable
(*SolidWorks Certified cards and drivers recommended)
SolidWorks 2015 (64 bit recommended)
Microsoft Office 2010 (64 bit recommended)





J

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