



BTG SOLIDWORKS Add-In

Version: 1.5

Date: 19 Sep 2024

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This is the guide for BTG SolidWorks Add-In which is developed by Bend-tech group. This Add-In is designed to aid the engineers in Bend-tech group and any related party to automate redundancy tasks when designing products using SolidWorks. This Add-In utilizes SolidWorks API and is written in C# using Visual Studio IDE. All interfaces in the Add-In were designed with WPF (Window Presentation Foundation). Newer functions were implementing MVVM (Model-View-ViewModel) architecture for better maintainability and testability. Version control as well as publishing are through GitHub. This Add-In required Serial No. from Bend-Tech Group for activation.



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1. What's New

1.1. 6.1.6.3 Bug Fixing Project Export rename body

Fixing bug when renaming body doesn't work if there is same name exist in the assembly.

1.2. Added interface to Bom Drawing macro.

Added interface to allow user to choose which model to be excluded from drawing generation. Moved setting to this interface.

1.3. Added Check Insert Part Tab to Design Checker macro.

Added capability to check whole assembly or a single part file whether the Insert Part feature import custom properties or cut list properties.

1.4. Remove finishing option from Access & Inspection Door Configurator macro.

Remove Hot dip galvanized + painted Y14 yellow option from Finishing in Access & Inspection Door Configurator macro.

2. Setup

2.1. Prerequisite

2.1.1. This add-in is developed and tested with the following software installed on the PC. There might be some error if this prerequisite does not meet.

2.1.1.1. SolidWorks 2022 sp5.0 or newer.

2.1.1.2. Microsoft 365

2.1.1.3. .NET 8.0 desktop runtime

2.1.2. If you don't have .NET 8.0 desktop runtime installed on your computer, the installer will show you the link for installation.

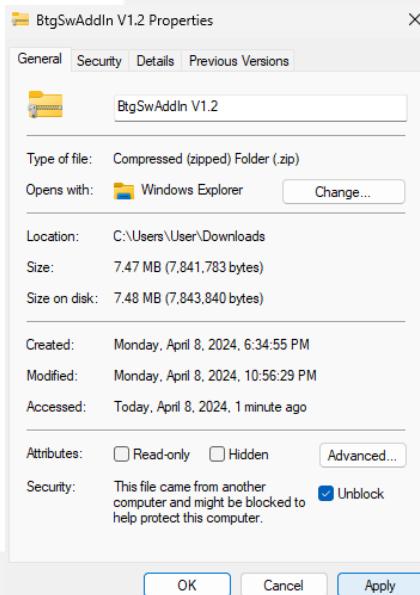
2.2. Installation

2.2.1. Visit website: [BendTech App \(btgappweb.firebaseio.com\)](https://btgappweb.firebaseio.com/)



2.2.2. Choose "Download Latest" and wait for installer to finish download.

2.2.3. Unblock the zipped file by Right click > Properties > General tab > Check on Unblock > Apply.

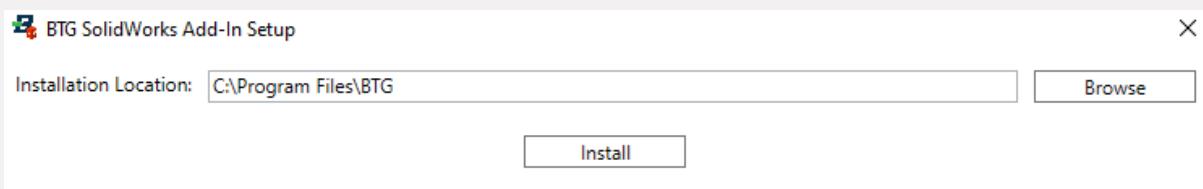


2.2.4. Extract the setup file.

2.2.5. Make sure SolidWorks is **NOT** running.

2.2.6. Run “BtgSwAddInSetup2.exe” as **ADMIN**.

2.2.7. Follow prompted message to install the Add-In.



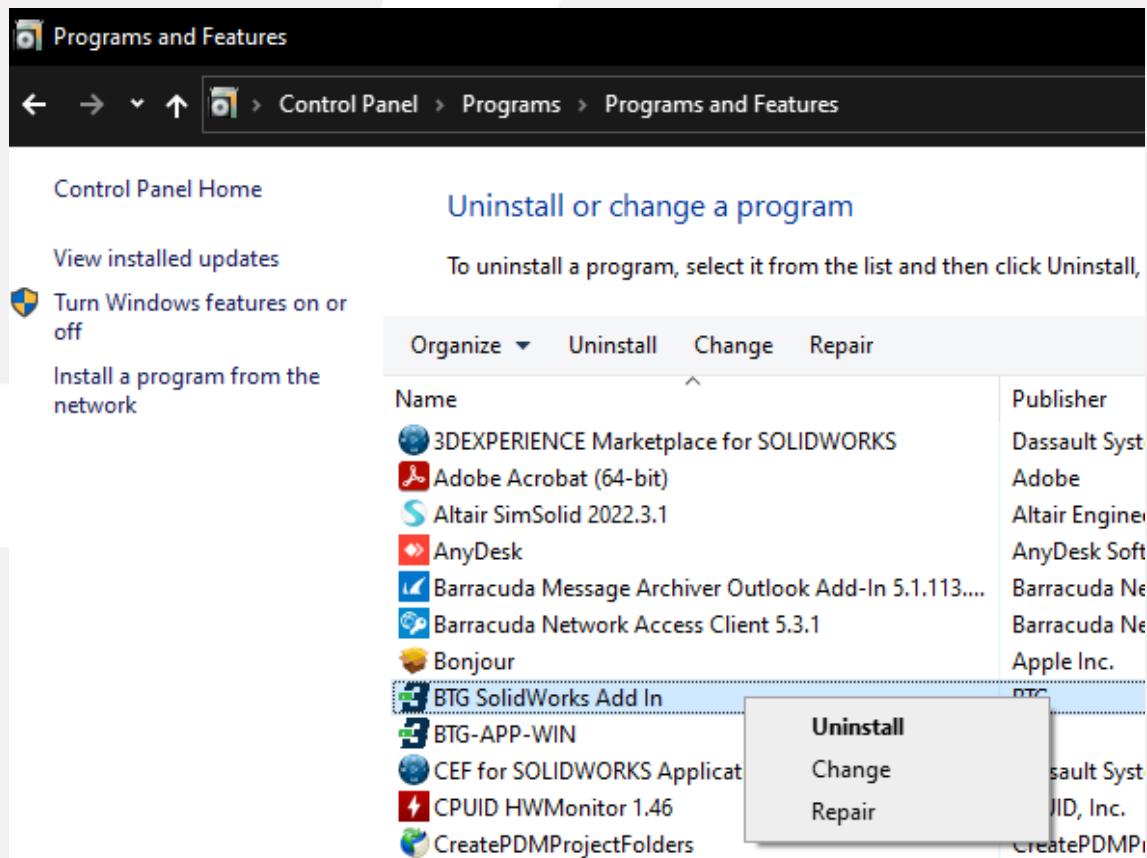
2.2.8. Click close when installation completed.

Info: 20/03/2024 11:26:43 : Start installing...
Info: 20/03/2024 11:26:43 : Done Installation!

2.2.9. If you have older version, it should update the add-in automatically.

2.3. Uninstallation

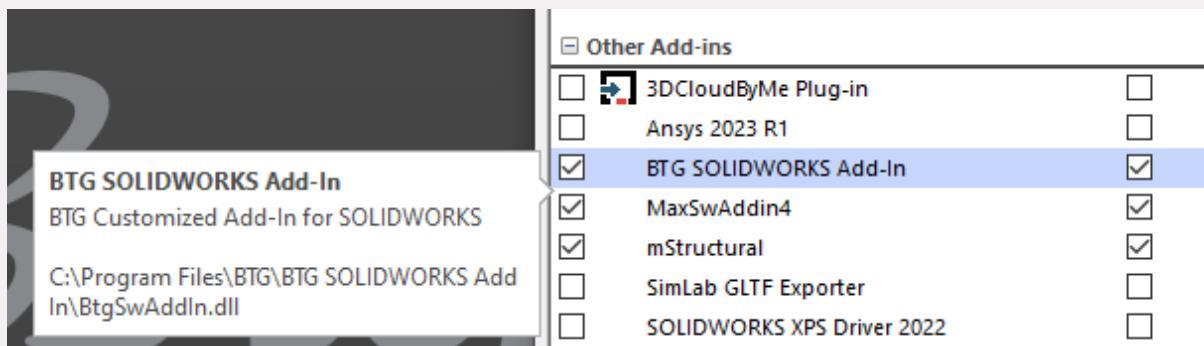
2.3.1. Go to Control Panel > Programs > Programs and Features to uninstall this Add-In.



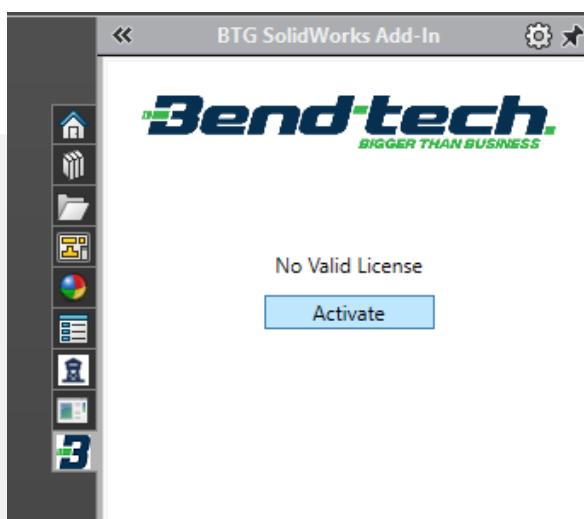
2.4. License Activation

2.4.1. Start SolidWorks.

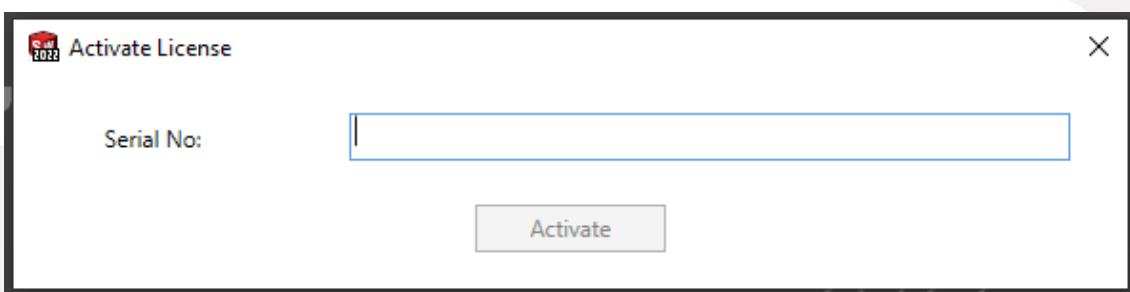
2.4.2. Make sure BTG SolidWorks Add-In is activated at Tools > Add-Ins...



2.4.3. Click "Activate" button at BTG SOLIDWORKS Add-In task pane.



2.4.4. Key in serial number and click "Activate" button.

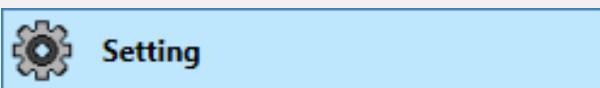


2.4.5. In case built-in activation failed, download standalone activator to activate through this link:

<https://github.com/MaxwellBTG/BtgSwAddInPublish/blob/main/Activator%20V1.0.2.zip>

3. Tools

3.1. Setting



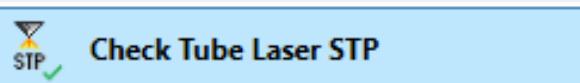
3.1.1. Some macro settings are stored at here. Refer to individual macro for each setting.

Setting	Value	Browse	Open
Start Model Path	D:\VaultView\BendTech\Library\02-DESIGN LIBRARY\BT LIBRARY\START MODE	<input type="button" value="Browse"/>	
Edge Note	ETCHING;ETCHING (TYP);LINE ETCHING;LINE ETCHING (TYP);SLOT;2x SLOTS		
Delete Tube Laser Body (Part)	B:\Macros\mStructural\Settings\dtlbstring.txt	<input type="button" value="Browse"/>	<input type="button" value="Open"/>
Delete Tube Laser Body (Assembly)	B:\Macros\mStructural\Settings\adtlbstring.txt	<input type="button" value="Browse"/>	<input type="button" value="Open"/>
CreateAll View Gap (mm)	10		
Auto Generate DXF Sheets Configuration File Location	B:\Macros\mStructural\Settings\AutoDxf	<input type="button" value="Browse"/>	<input type="button" value="Open"/>
Sort Cutlist Setting	B:\Macros\mStructural\Settings\sortsetting.txt	<input type="button" value="Browse"/>	<input type="button" value="Open"/>
Prep DXF Setting	B:\Macros\mStructural\Settings\PrepDxf	<input type="button" value="Browse"/>	<input type="button" value="Open"/>
Fast Cap Model	D:\VaultView\BendTech\Library\02-DESIGN LIBRARY\BT LIBRARY\CAPS\MD 33	<input type="button" value="Browse"/>	
Checkbox Default	<input type="checkbox"/> Export to production folder <input checked="" type="checkbox"/> Check Weldment Cut List Length		
<input type="button" value="Default"/>		<input type="button" value="Confirm"/>	

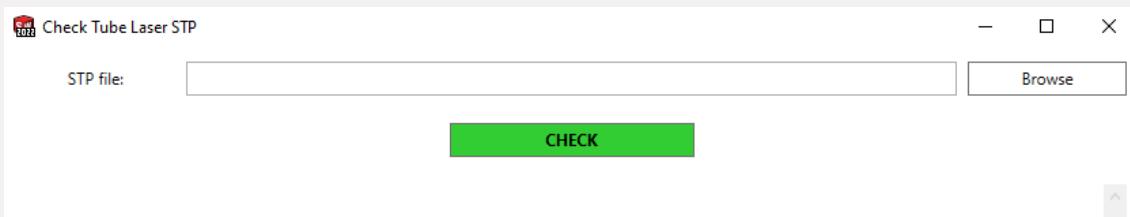
3.1.2. Default button will set all settings to default, most of the settings are pointing to mapped drive directory (B:\). If default setting does not set the directory properly, use browse button to point it to the correct directory.

3.1.3. Click on “Confirm” button to save changes.

3.2. Check Tube Laser Step

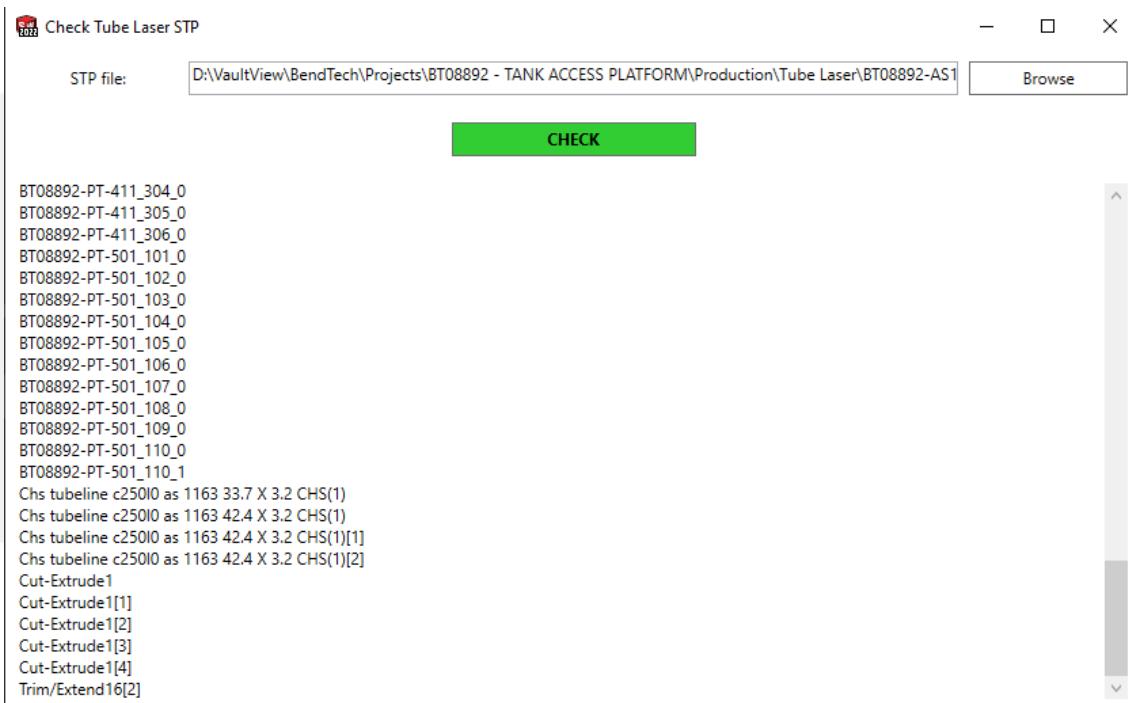


3.2.1. A macro to check for the body name of STEP file.



3.2.2. Browse for the STEP or STP file using the Browse button or type in the path of that file. Then click CHECK button.

3.2.3. The place below will display the list of names inside the STEP file.

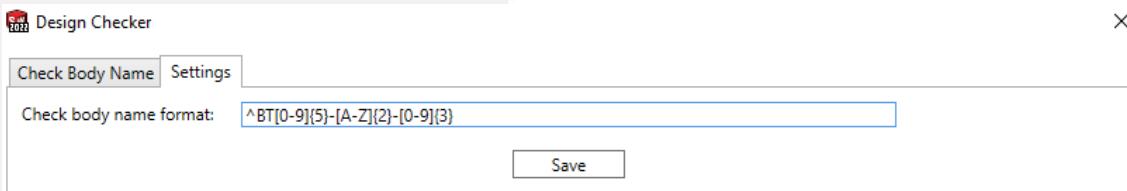


3.3. Design Checker



3.3.1. A tool to check your design.

3.3.2. Settings tab



3.3.2.1. Store all the settings for this macro.

3.3.2.2. Must click on the Save button to save any modified setting.

3.3.3. Log

Log:

09/08/24 15:59:57 : Error : Open a document before running check macro.
09/08/24 15:59:38 : Info : Setting saved!

3.3.3.1. Any message from the macro will be logged into this area.

3.3.4. Check Body Name Tab

The screenshot shows the 'Design Checker' application window. At the top, there's a toolbar with icons for file operations and a 'Design Checker' button. Below the toolbar, a navigation bar has 'Check Body Name' selected and a 'Settings' tab. The main area is titled 'CHECK' in a green bar. A table lists various part numbers and their corresponding body names, with a 'Complied' column indicating whether the name matches the set format. Most entries are in green, while some like 'Cut-Extrude1[1]' through 'Cut-Extrude1[4]' are in red. Below the table is a 'Log' section containing a single entry: '09/08/24 16:21:48 : Info : Completed check body name macro.'

Part Name	Body Name	Complied
BT08892-PT-201.SLDprt	BT08892-PT-201_110_0	True
BT08892-PT-207.SLDprt	BT08892-PT-207_201_0	True
BT08892-PT-207.SLDprt	BT08892-PT-207_202_0	True
BT08892-PT-207.SLDprt	BT08892-PT-207_202_1	True
BT08892-PT-207.SLDprt	BT08892-PT-207_203_0	True
BT08892-PT-208.SLDprt	BT08892-PT-208_303_0	True
BT08892-PT-208.SLDprt	BT08892-PT-208_303_1	True
BT08892-PT-208.SLDprt	BT08892-PT-208_304_0	True
BT08892-PT-208.SLDprt	BT08892-PT-208_301_0	True
BT08892-PT-208.SLDprt	BT08892-PT-208_302_0	True
BT08892-PT-208.SLDprt	BT08892-PT-208_305_0	True
BT08892-PT-221.SLDprt	Cut-Extrude1[1]	False
BT08892-PT-221.SLDprt	Cut-Extrude1[2]	False
BT08892-PT-221.SLDprt	Cut-Extrude1[3]	False
BT08892-PT-221.SLDprt	Cut-Extrude1[4]	False
BT08892-PT-501.SLDprt	BT08892-PT-501_101_0	True
BT08892-PT-501.SLDprt	BT08892-PT-501_102_0	True
BT08892-PT-501.SLDprt	BT08892-PT-501_106_0	True
BT08892-PT-501.SLDprt	BT08892-PT-501_110_0	True

Log:

```
09/08/24 16:21:48 : Info : Completed check body name macro.
```

3.3.4.1. Body name should be renamed according to the format:

PartNumber_BalloonNumber_Instance

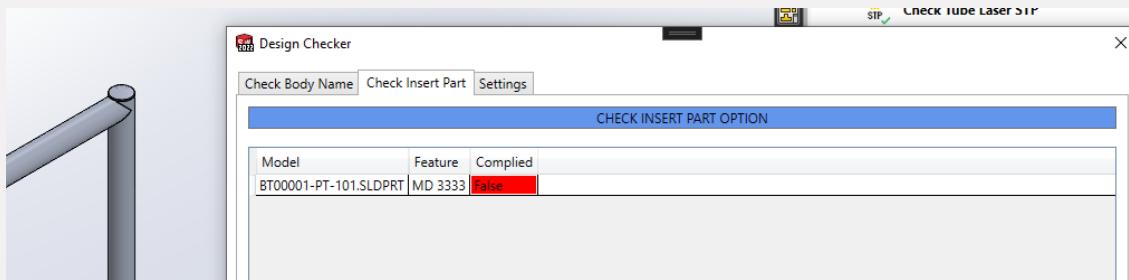
3.3.4.2. For example, for a file with part number “BT08892-PT-501” with balloon number “101” in drawing and is the first instance, the body name should be “BT08892-PT-501_101_0”. Second instance should be “BT08892-PT-501_101_1” and so on.

3.3.4.3. Open any part or assembly file then click on the check button. It will extract all body name and display it in the table. If the body name does not comply with the format set in the setting, it will show false.

3.3.4.4. The check body name format is using Regular Expression method which can be modified in the setting tab.

The screenshot shows the 'Settings' tab of the 'Check Body Name' interface. It features a text input field labeled 'Check body name format:' containing the regular expression '^BT[0-9]{5}-[A-Z]{2}-[0-9]{3}'. This regular expression specifies a five-digit part number followed by a two-letter instance identifier and a three-digit instance number.

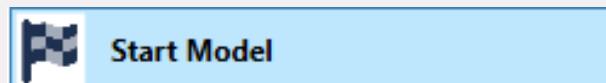
3.3.5. Check Insert Part Tab



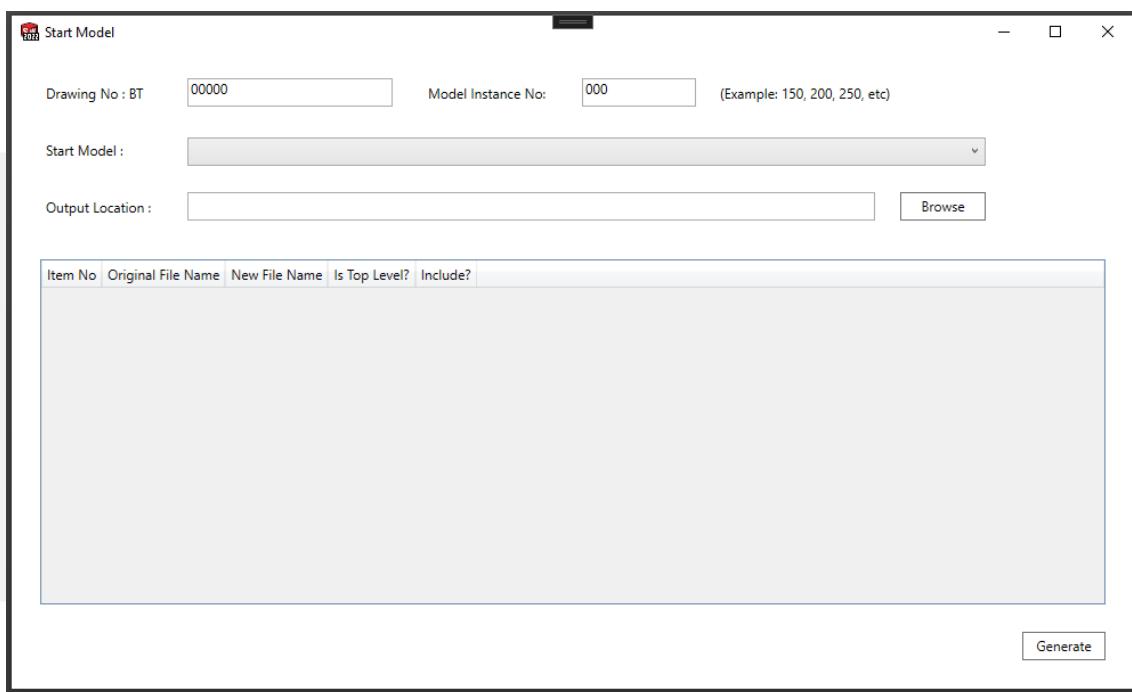
- 3.3.5.1. Most of the library parts were setup using Cut List Properties thus when inserting part into another part using Insert Part feature should only import Cut List Properties unless otherwise specified.
- 3.3.5.2. Open any part or assembly file then click on the check button. It will extract all Insert Part Feature and display it in the table. If the feature does not comply with the setting, it will show false.

4. Automation (Model)

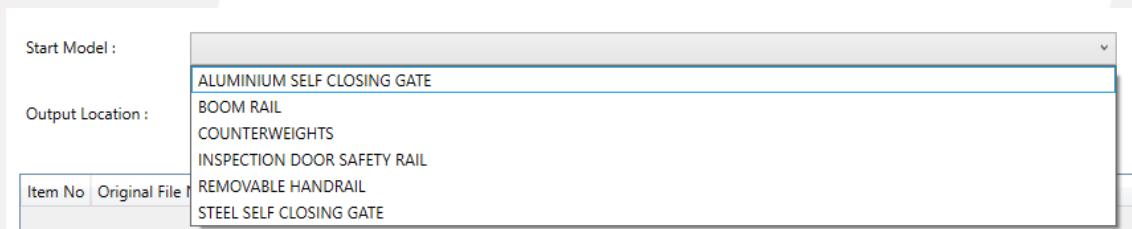
4.1. Start Model



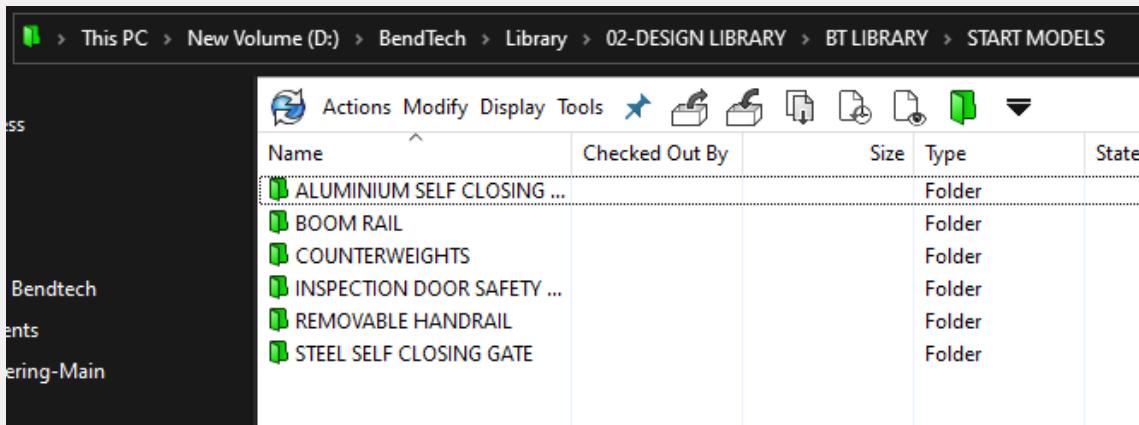
- 4.1.1. A Macro to save and rename files in a folder according to predefined template. It only supports one level of folder structure, any subfolder inside that folder will be ignored. It only supports part and assembly file.
- 4.1.2. Set “Start Model Path” at Setting.
- 4.1.3. Make sure no document is opened when running this macro.
- 4.1.4. Click on the macro to show Start Model view.



- 4.1.5. From the Start Model drop down, you can select the model you wish to create.



4.1.6. The dropdown list is the same as the folder structure of the folder defined in setting.



4.1.7. A table with all the models inside that folder will be displayed.

Item No	Original File Name	New File Name	Is Top Level?	Include?
1	BT0XXX_ALU SELF CLOSING GATE.SLDASM	BT00000-AS-000.sldasm	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
2	BT0XXX_ENTRANCE POST.SLDPRT	BT00000-PT-001.sldprt	<input type="checkbox"/>	<input checked="" type="checkbox"/>
3	BT0XXX_ALU GATE FRAME.SLDPRT	BT00000-PT-002.sldprt	<input type="checkbox"/>	<input checked="" type="checkbox"/>

4.1.8. Change Drawing No and Model Instance No accordingly.

Drawing No : BT 08888 Model Instance No: 800 (Example: 150, 200, 250, etc)

4.1.9. You may notice the new name column will change accordingly.

Item No	Original File Name	New File Name	Is Top Level?	Include?
1	BT0XXX_ALU SELF CLOSING GATE.SLDASM	BT08888-AS-800.sldasm	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
2	BT0XXX_ENTRANCE POST.SLDPRT	BT08888-PT-801.sldprt	<input type="checkbox"/>	<input checked="" type="checkbox"/>
3	BT0XXX_ALU GATE FRAME.SLDPRT	BT08888-PT-802.sldprt	<input type="checkbox"/>	<input checked="" type="checkbox"/>

4.1.10. You can override the Name as well.

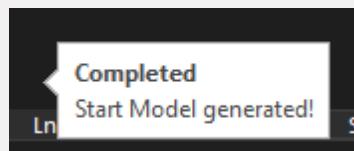
Item No	Original File Name	New File Name	Is Top Level?	Include?
1	BT0XXX_ALU SELF CLOSING GATE.SLDASM	BT08888-AS-800.sldasm	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
2	BT0XXX_ENTRANCE POST.SLDPRT		<input type="checkbox"/>	<input type="checkbox"/>
3	BT0XXX_ALU GATE FRAME.SLDPRT	BT08888-PT-805.sldprt	<input type="checkbox"/>	<input checked="" type="checkbox"/>

4.1.11. Browse for output location.

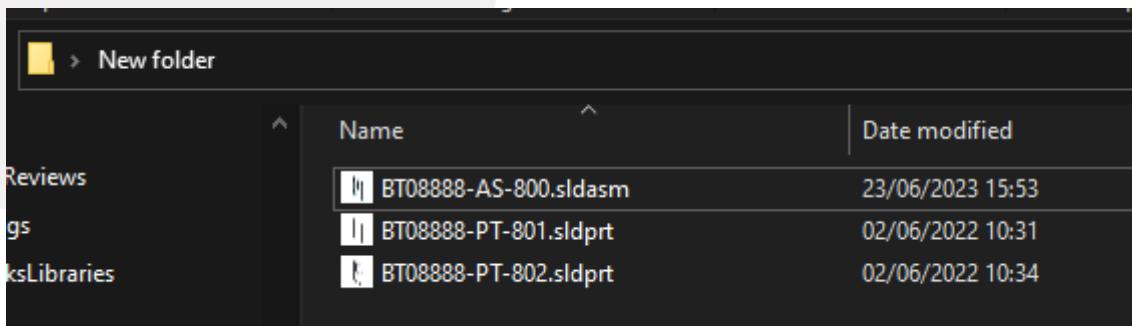
Output Location : C:\Users\MaxwellWG\Desktop\New folder

4.1.12. Press "Generate" button.

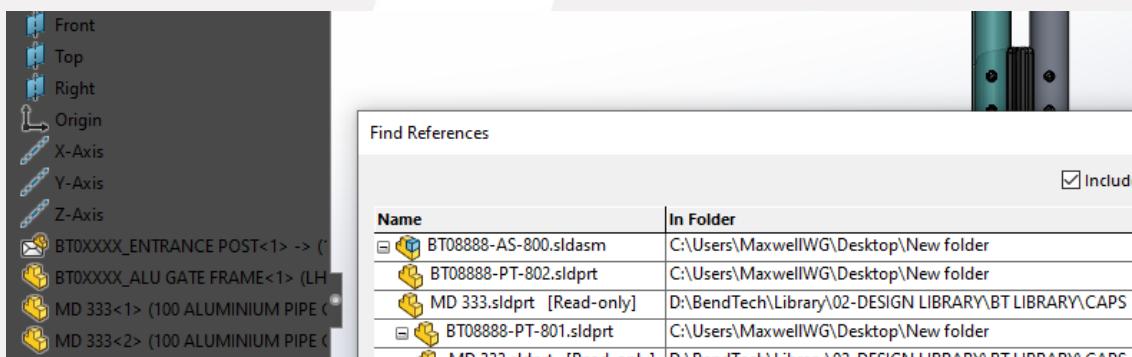
4.1.13. A bubble tooltip will appear at your cursor to indicate that the start model has been generated.



4.1.14. Go to the output folder and you can see your model there.

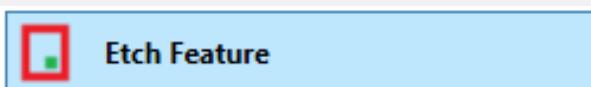


4.1.15. In the feature tree it will still be showing the old model's name until the next rebuild. However, the reference is already updated.

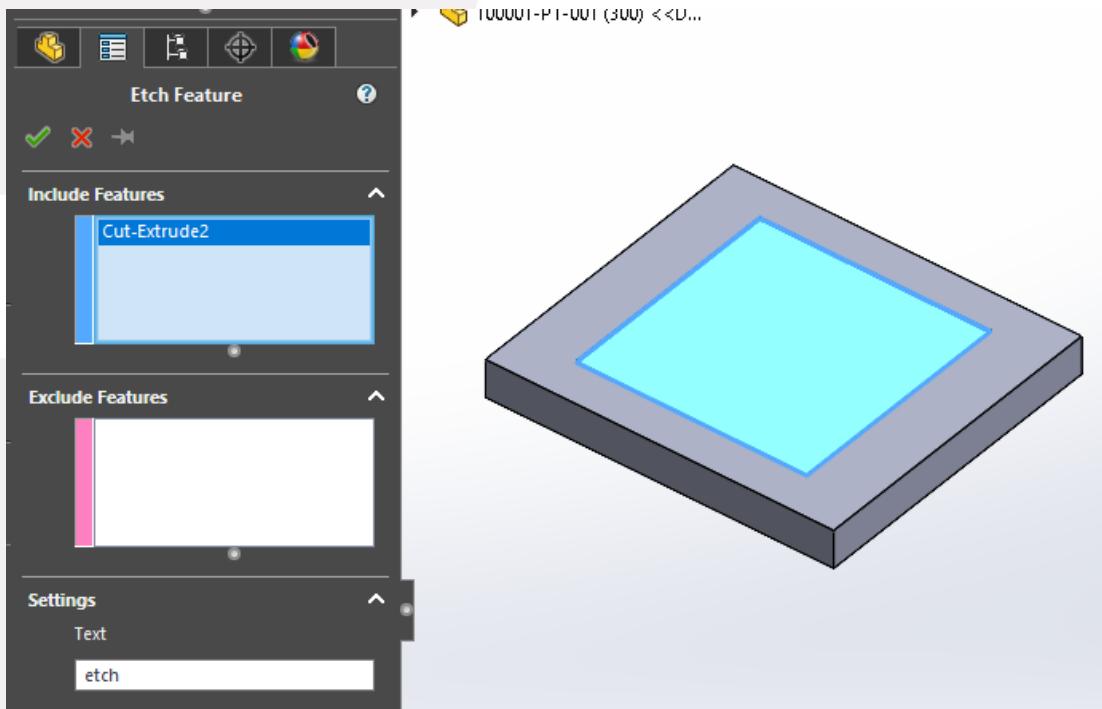


4.1.16. To update, just rebuild the model.

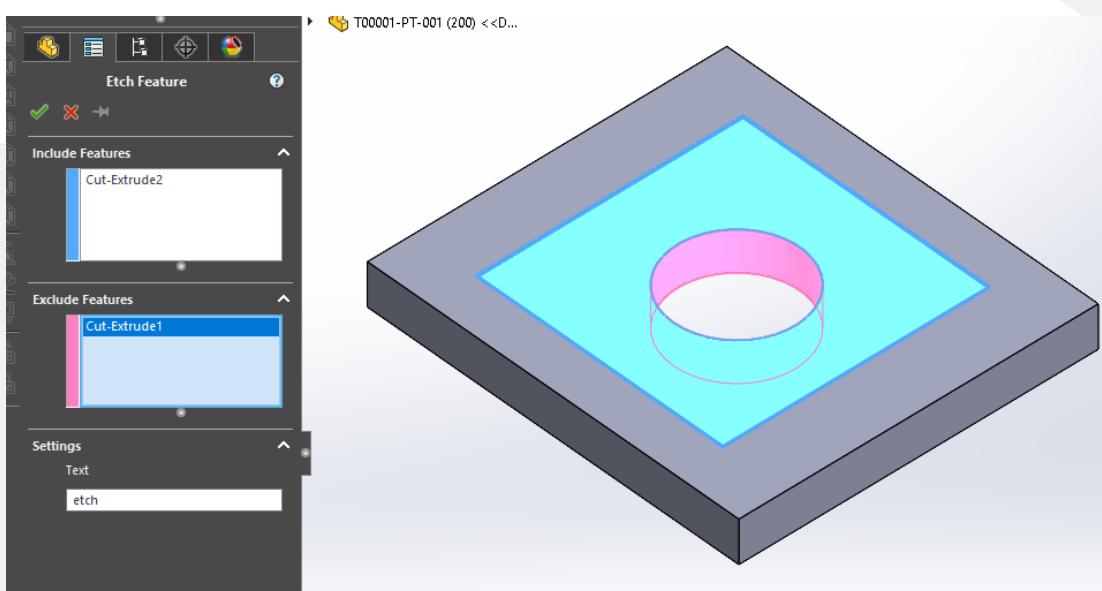
4.2. Etch Feature



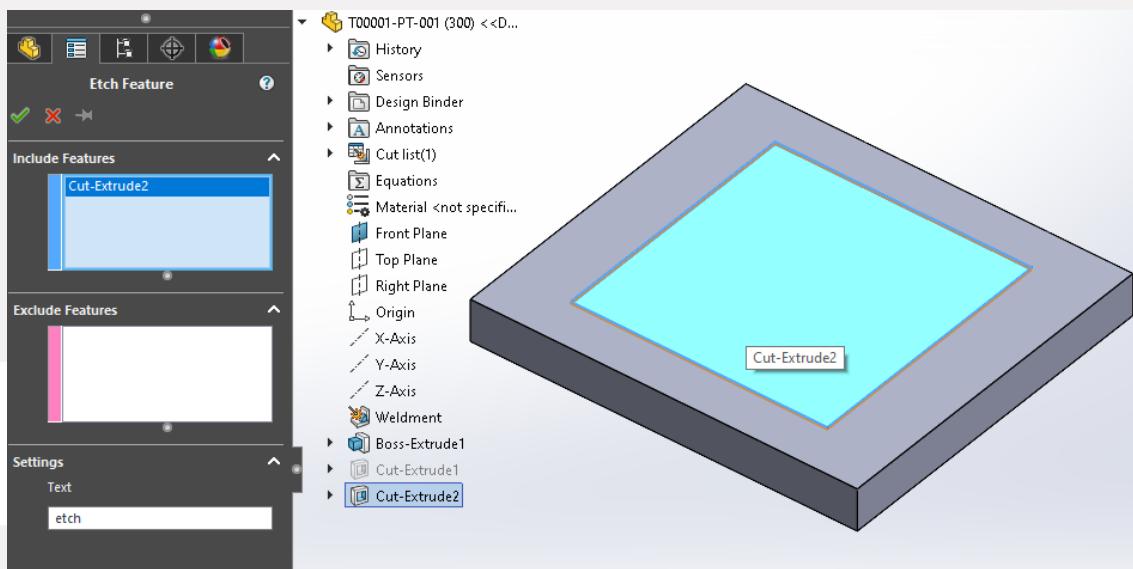
- 4.2.1. A SolidWorks macro feature that marks the edge of selected feature for drawing processing. It only supports part file.
- 4.2.2. Video: <https://youtu.be/dOhFuD62B4w>
- 4.2.3. Click the macro. Select features or faces for Include Features.



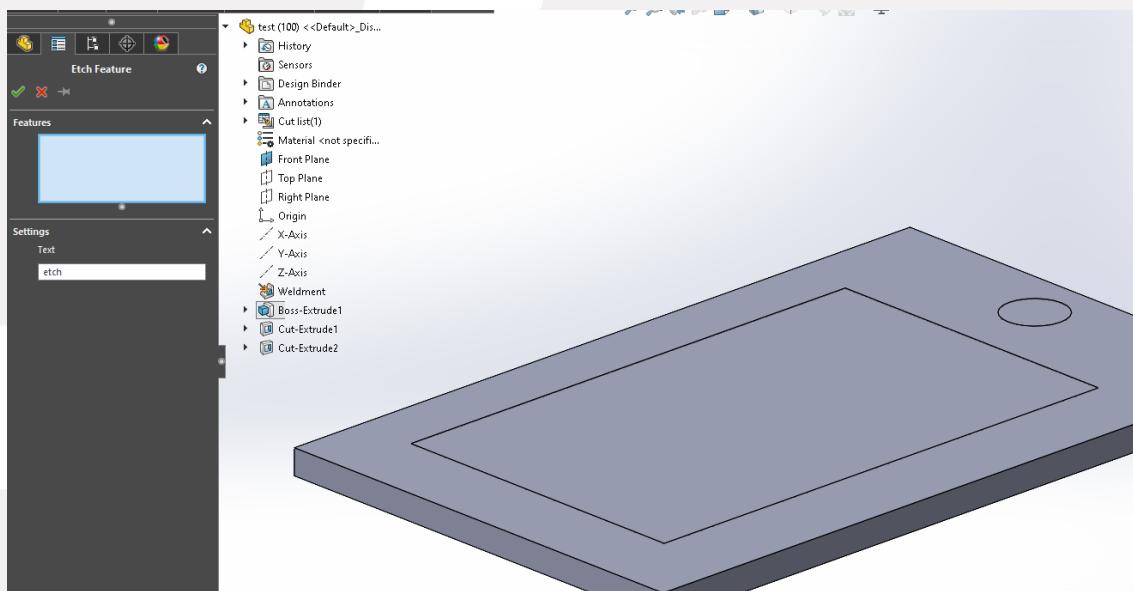
- 4.2.4. If there is a hole in the middle, you might need to select the hole as exclude feature since the edge of the hole should not be coloured red.



4.2.5. Expand the Flyout Feature Manager Design Tree to select the feature. Notice the highlighted faces in graphics area, it indicates the selected feature and affected faces. You can select the feature through graphics area as well. Try to any face and you can notice the underlying feature will be selected as well.

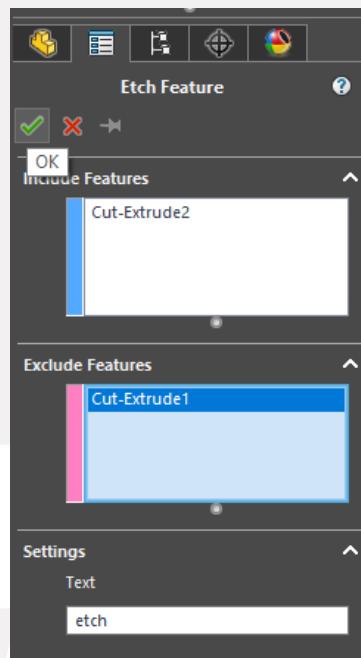


4.2.6. Select the feature again to deselect it.

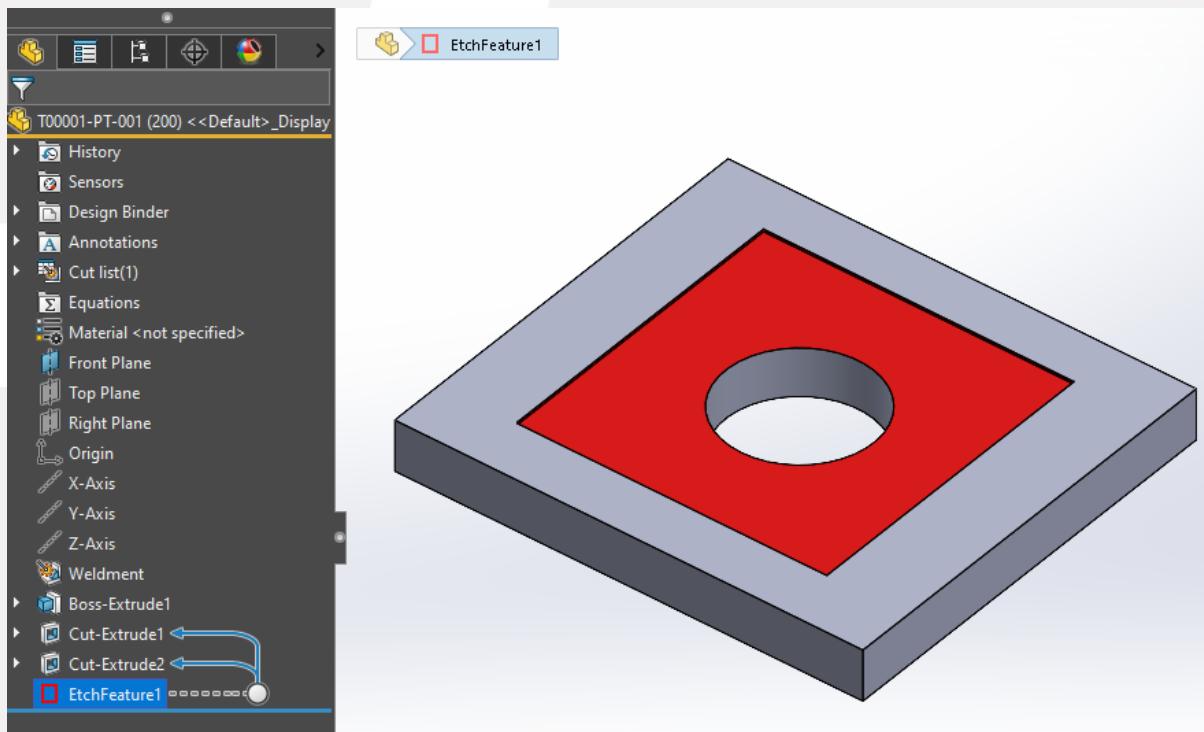


4.2.7. Text field in the settings group let user to customize the base name for entity naming.

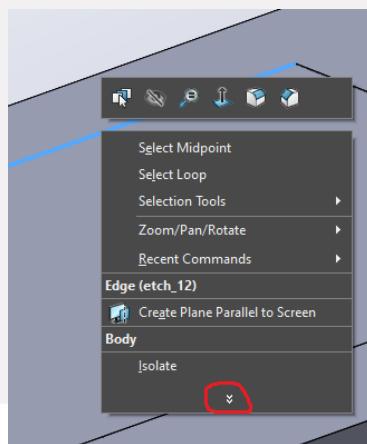
4.2.8. Once confirmed your selection, click the green tick or OK button. Tips: you can press shortcut key D to bring the ok button to your cursor.



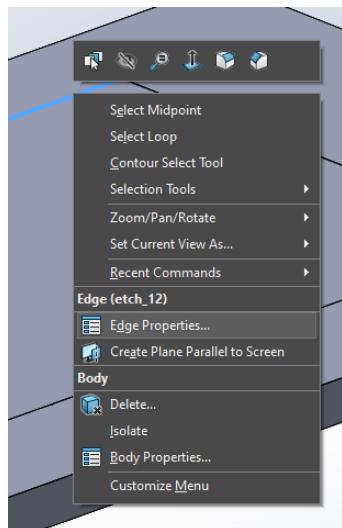
4.2.9. A feature with the name EtchFeature1 is created at the bottom of your Feature Manager Design Tree. The faces associated with the feature are coloured red to indicate that feature has been applied.



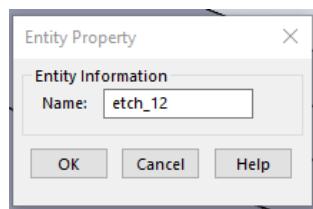
4.2.10. Now, go to right click the edge of the feature that you have defined as etching. You can notice that the edge you selected has a property of etch XX. Click on the double down arrow button to expand for more options.



4.2.11. Click on Edge Properties...

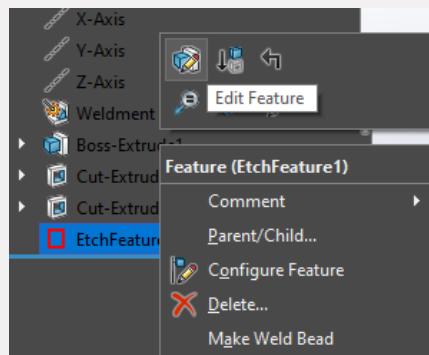


4.2.12. [Prep DXF Export](#) macro will colour any edge that has "etch" text in the Entity Property.

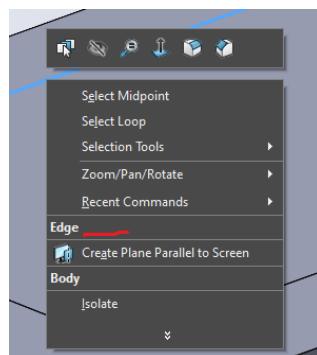


4.2.13. The edge that is selected in Exclude Features selection box will not have the "etch" text associate to it.

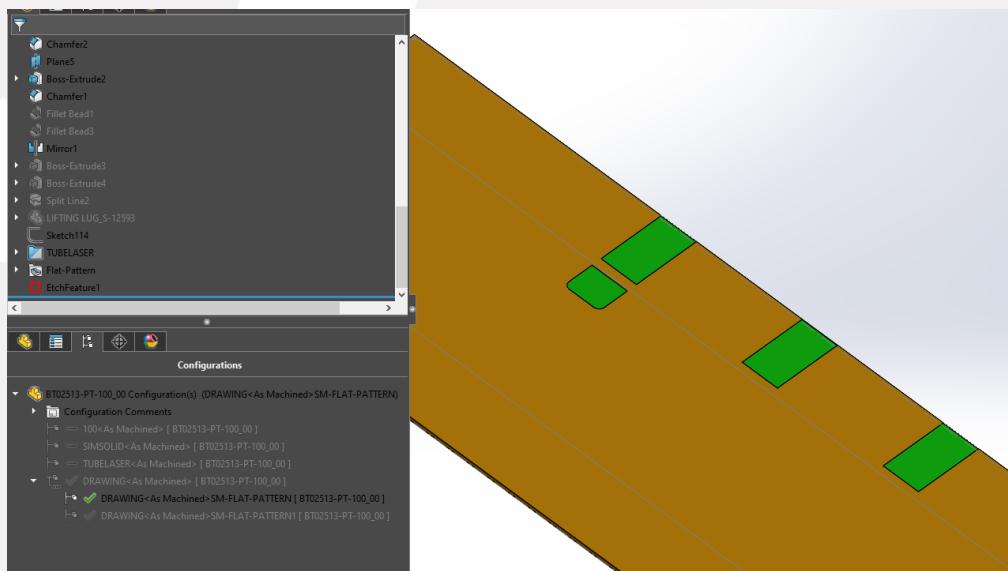
4.2.14. In case you have selected the wrong feature, you can go back to edit your Etch Feature and remove or add on more feature as your wish.



4.2.15. Once removed the feature, the Entity Properties on the edge will be removed as well.



4.2.16. If the etching is on the model that is pressed or flattened, add the Etch Feature in flatten configuration so that the result will be consistent.





4.2.17. When removing the feature from Etch Feature, sometimes it will set different colour on the feature. This is due to the render material used having a secondary colour. In that case, you can set the secondary colour same as the dominant colour, then run the Macro again.



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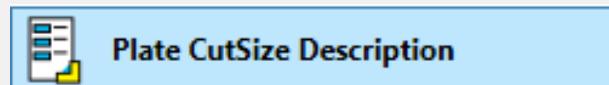
sales@bendtechgroup.com.au

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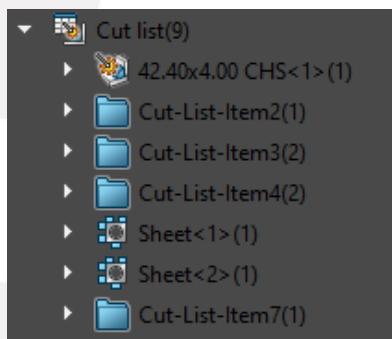
2-10 Kewdale Road, Welshpool WA 6106



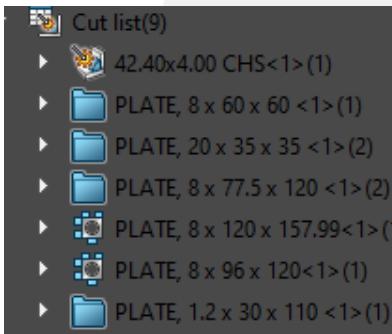
4.3. Plate Cut Size Description



- 4.3.1. Create a 3D Bounding Box and add Description according to a specific format for each cut list folder that has either "cut-list-item" or "sheet" in the name. It only supports part file.
- 4.3.2. Open the part you wish to add 3D Bounding Box and click on Plate Weld Note button.
- 4.3.3. Initially the body folder structure is as below:



- 4.3.4. After running the Macro:



- 4.3.5. Sheet metal body description filled with same syntax as cut list item.

PLATE, 8 x 120 x 157.99<1>	4	Bounding Box Area	Text	'SW-Bounding Box Area@@@PLATE, 8 x 120 x 157.99<1> @BT07714-PT-104_00,	1
PLATE, 8 x 96 x 120<1>	5	Bounding Box Area-Blank	Text	'SW-Bounding Box Area-Blank@@@PLATE, 8 x 120 x 157.99<1> @BT07714-PT-1	1
PLATE, 1.2 x 30 x 110 <1>	6	Cutting Length-Outer	Text	'SW-Cutting Length-Outer@@@PLATE, 8 x 120 x 157.99<1> @BT07714-PT-104_	6
	7	Cutting Length-Inner	Text	'SW-Cutting Length-Inner@@@PLATE, 8 x 120 x 157.99<1> @BT07714-PT-104_0	3
	8	Cut Outs	Text	'SW-Cut Outs@@@PLATE, 8 x 120 x 157.99<1> @BT07714-PT-104_00.SLDPR	2
	9	Bends	Text	'SW-Bends@@@PLATE, 8 x 120 x 157.99<1> @BT07714-PT-104_00.SLDPR'	1
	10	Bend Allowance	Text	'SW-Bend Allowance@@@PLATE, 8 x 120 x 157.99<1> @BT07714-PT-104_00.SLD	0
	11	MATERIAL	Text	'SW-Material@@@PLATE, 8 x 120 x 157.99<1> @BT07714-PT-104_00.SLDPR'	G
	12	Mass	Text	'SW-Mass@@@PLATE, 8 x 120 x 157.99<1> @BT07714-PT-104_00.SLDPR'	0
	13	Description	Text	PLATE, 'SW-Sheet Metal Thickness@@@PLATE, 8 x 120 x 157.99<1> @BT07714-P	P

4.4. Delete Tube Laser Body

Delete Tube Laser Body

4.4.1. Delete all body in weldment cut list that match the pre-defined string in setting. It only supports Part and Assembly.

4.4.2. Make sure the location of the string text files in setting are correct.

Delete Tube Laser Body (Part)	B:\Macros\mStructural\Settings\dtlbstring.txt	<input type="button" value="Browse"/>	<input type="button" value="Open"/>
Delete Tube Laser Body (Assembly)	B:\Macros\mStructural\Settings\adtlbstring.txt	<input type="button" value="Browse"/>	<input type="button" value="Open"/>

4.4.3. You can click on the “Default” button at the setting to set it to Bend Tech default location of this setting file. You might need to change the front part of the default string if it is different from your own setting.

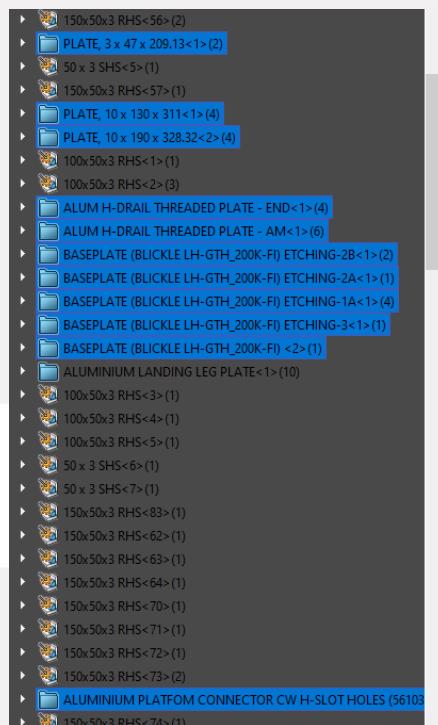
4.4.4. If you cannot access to the default setting file, you may create a similar text file and point the setting to this setting text file. For Part, each string is separated by “;” symbol, any cut list in the folder that matches any of the strings will be deleted. New line and the last “;” are not necessary, just for formatting purposes only.

```
dtlbstring.txt - Notepad
File Edit Format View Help
FB;
BASEPLATE;
THREADED PLATE;
PLATE,;
CONNECTOR;
```

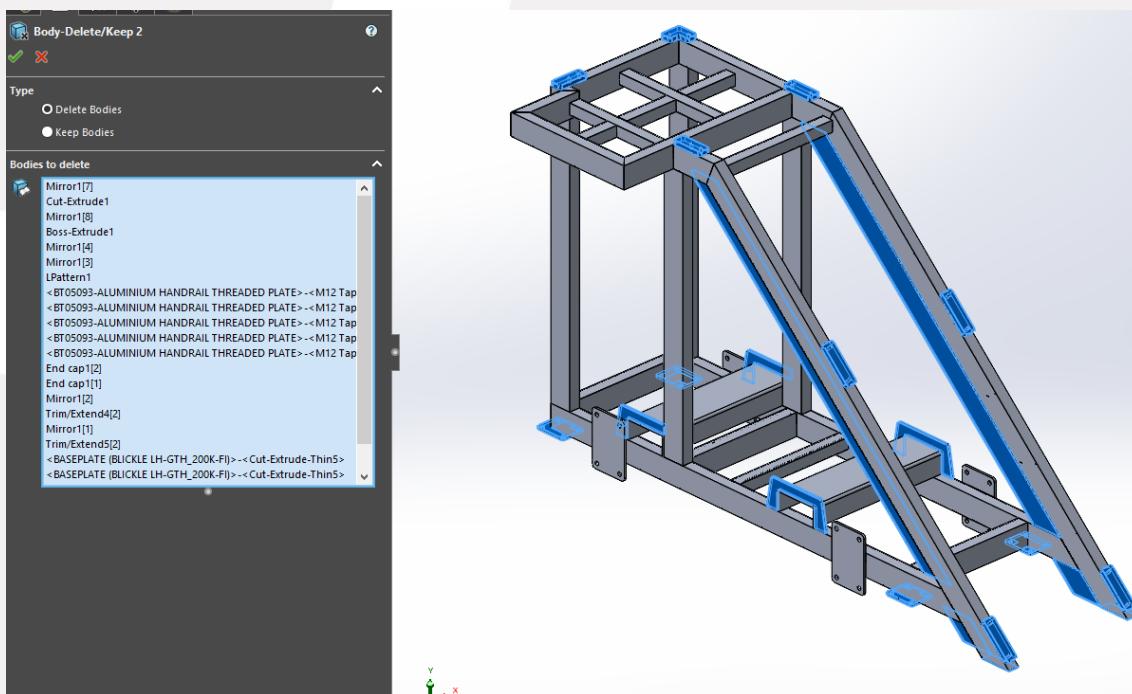
4.4.5. For Assembly, each string is separated by “;” symbol and enclosed by * symbol. * Symbol is a wildcard which represents any characteristic. All line should be enclosed by * symbol to cater for the parent component string.

```
adtlbstring.txt - Notepad
File Edit Format View Help
*-FT-*;
*-FS-*;
*LH-GTH_200K-FI*;
*MD 333*;
*DANGER SIGN*;
*TRUCLOSE HINGE*;
*SPRING BOLT LATCH*;
*GFM-5055-25*;
```

4.4.6. Open the document and press the Delete Tube Laser Body macro. If the document is Part, All bodies in the cut list folder that match the string will be selected.

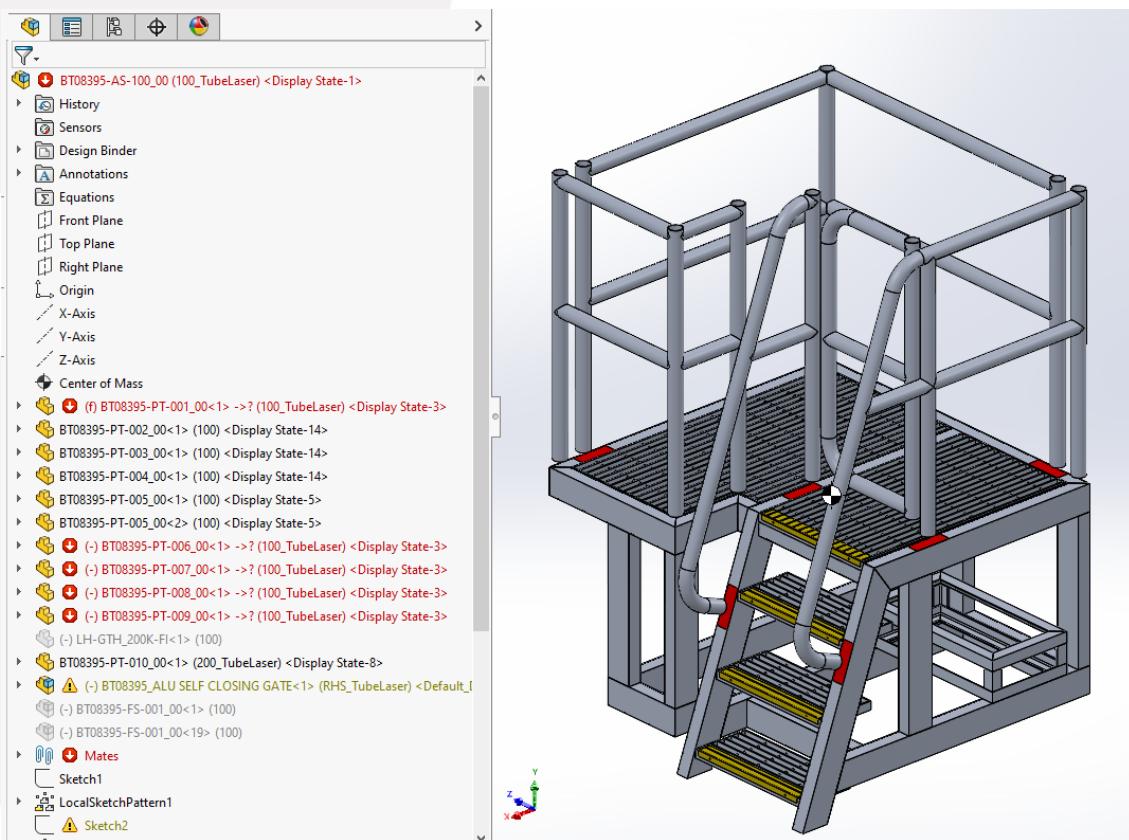


4.4.7. A Body-Delete/Keep feature will be created.



4.4.8. If the document is Assembly, any component that matches the condition in the Assembly setting will be suppressed and any component that matches the condition in the Part setting will have the bodies deleted. Any affected document will have a configuration created automatically with a suffix of “_TubeLaser”.

4.4.9. Errors and warnings on the model are expected since new configurations are created with bodies deleted. You shall check the result and save the model accordingly, the macro will not save the model.



4.5. Naming Project

Naming Project

4.5.1. A Macro to quickly rename your assembly structure according to the standard as stated below:

BT number: 99999

Main Assembly: BT99999-AS-100

Part file in main assembly: BT99999-PT-101, BT99999-PT-102...

Sub assembly file: BT99999-AS-150, BT99999-AS-200...

Part file in sub assembly BT99999-AS-150: BT99999-PT-151, BT99999-PT-152

Fastener assembly: BT99999-FT-110, BT99999-FT-120...

Any read-only, envelope or suppressed component will be omitted. Rename will fail if the component is renamed to a component which already exists physically on your disk.

4.5.2. Video: <https://youtu.be/9mqfVOBWfds>

4.5.3. Click on the Naming Project button. A tree view will be displayed.

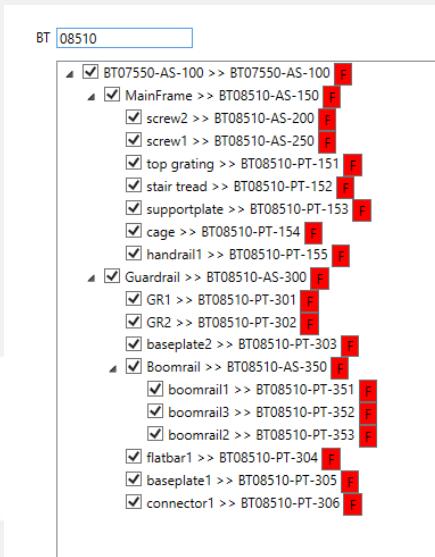


4.5.4. Each item in the tree view contains the following elements from left to right:

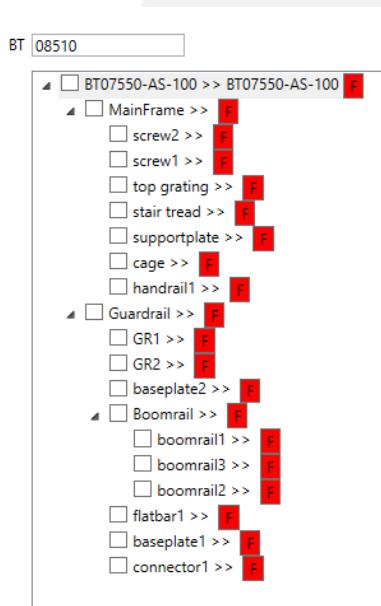
- 4.5.4.1. A check box to include or exclude the component.
- 4.5.4.2. Original component name.
- 4.5.4.3. ">>" symbol to separate original name with new name.
- 4.5.4.4. New component name.
- 4.5.4.5. A toggle button to set fastener flag for this component.

connector1 >> BT07550-PT-306 F

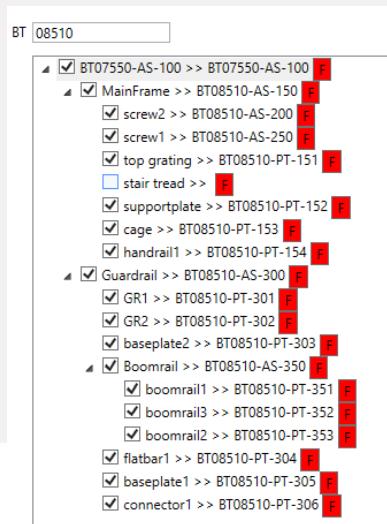
4.5.5. BT number is at the top of the interface, it is extracted from main assembly file name. Users may change the BT number manually. You can notice the new component will change dynamically with BT number. However, the main assembly's name will not change.



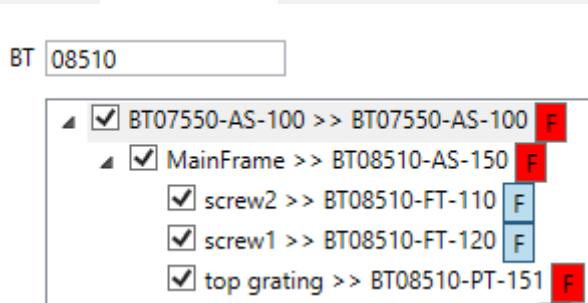
4.5.6. Unchecked/Checked on the parent node will do the same to its child node as well.



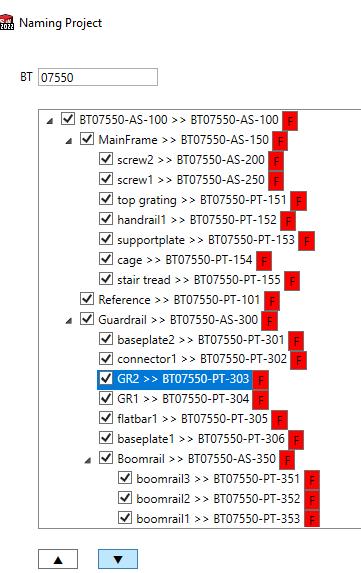
4.5.7. Unchecked/Checked on a component will trigger the rename Macro. The new component name will be assigned according to the new configuration.



4.5.8. Toggle on fastener toggle button will mark that component as fastener. The new component name will be re-assigned according to the new configuration.



4.5.9. Select any item that you wish to move up or down in the list using the ▲ or ▼ button.



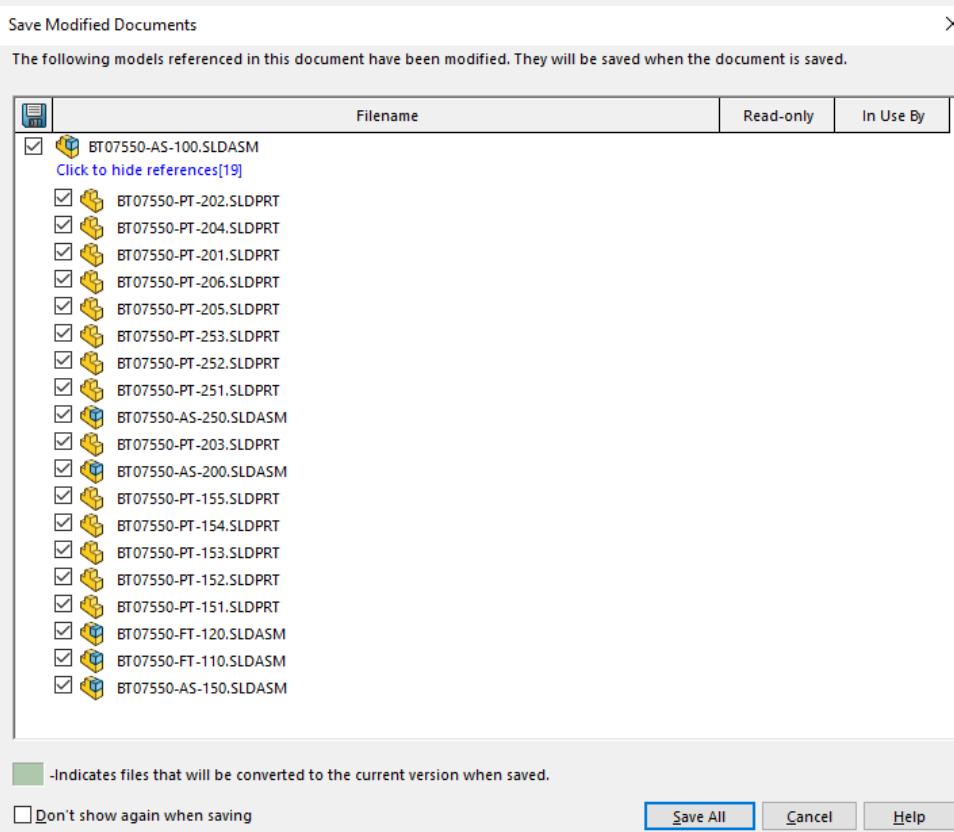
4.5.10. Once everything is set, press the Confirm button.



4.5.11. A message will be displayed to show rename status.



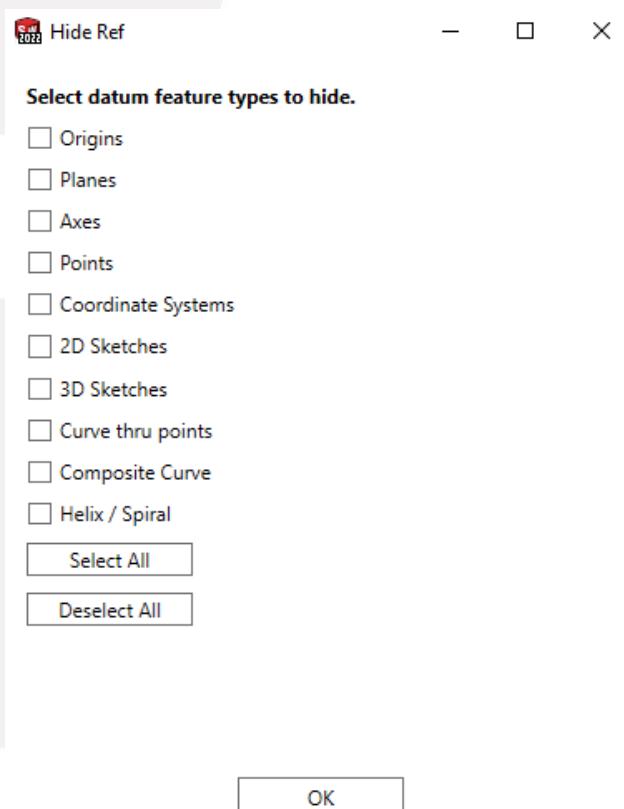
4.5.12. Double check your model if further rename is required. Once everything is confirmed, save your file to finalize the rename operation.



4.6. Hide Ref (Model)

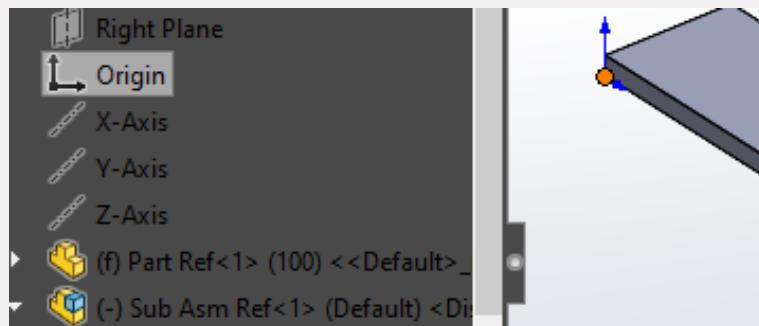


- 4.6.1. A macro to hide any selected feature found in the active document. This section explains the Macro when running in Part or Assembly document. For Drawing document, refer to [Hide Ref \(Drawing\)](#)
- 4.6.2. Open any document that you want to hide specific features. Click on the Hide Ref Macro button.
- 4.6.3. An interface listing all the supported feature types will be shown.

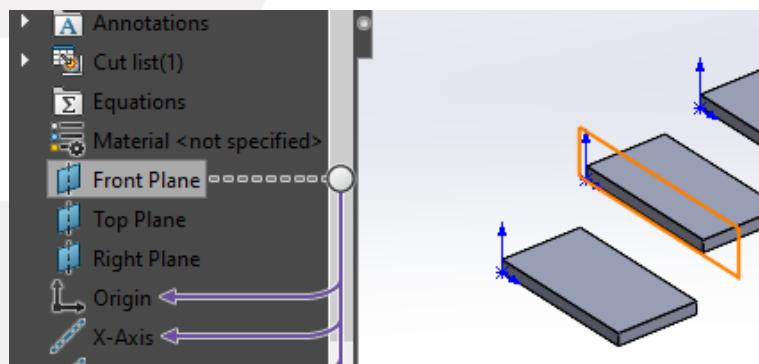


4.6.4. Below are the supported feature types:

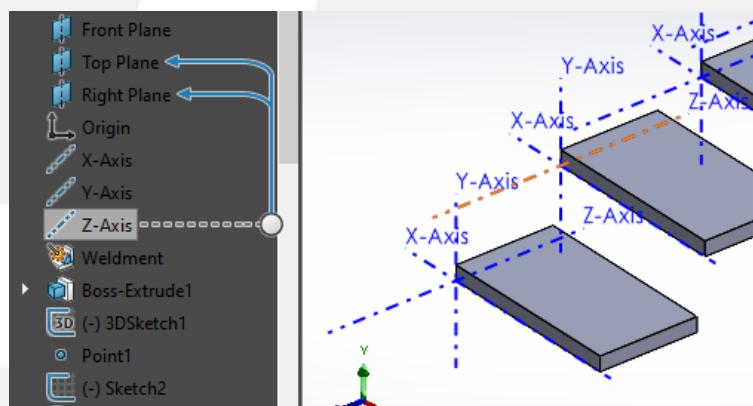
4.6.4.1. Origins



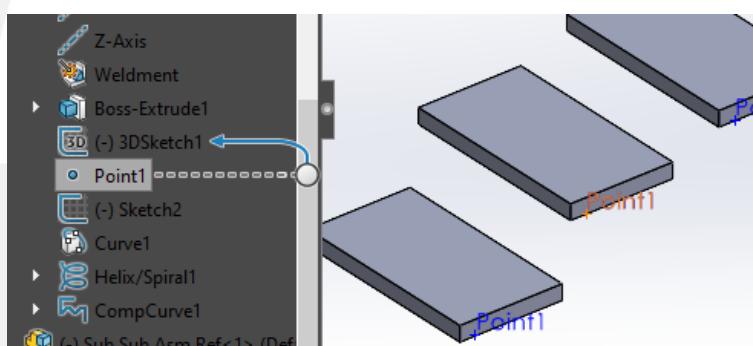
4.6.4.2. Planes



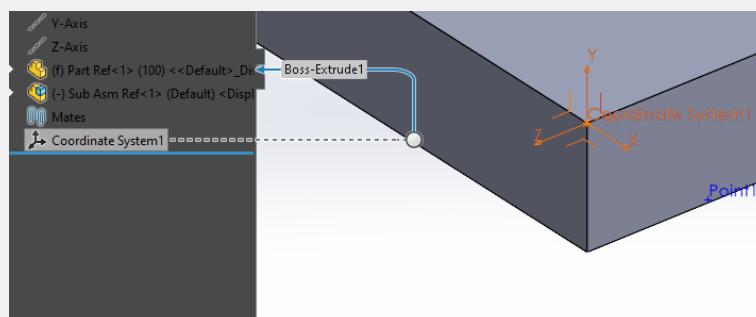
4.6.4.3. Axes



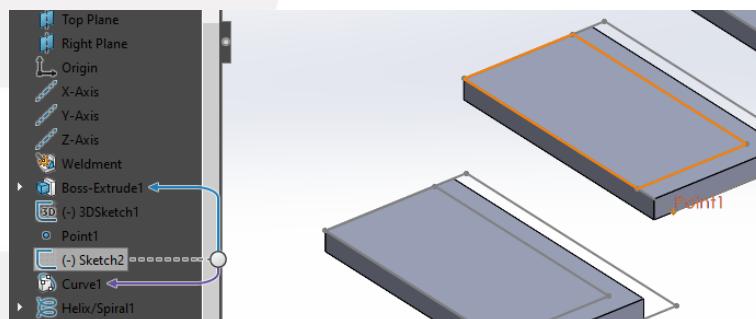
4.6.4.4. Points



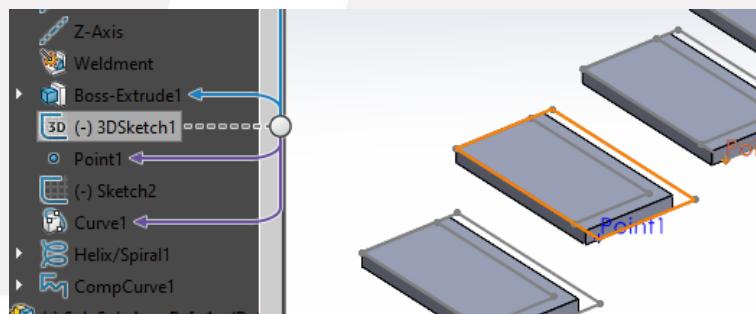
4.6.4.5. Coordinate Systems



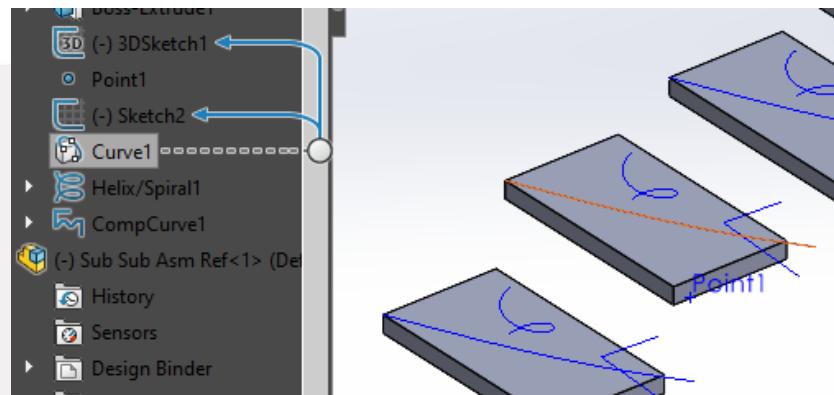
4.6.4.6. 2D Sketches



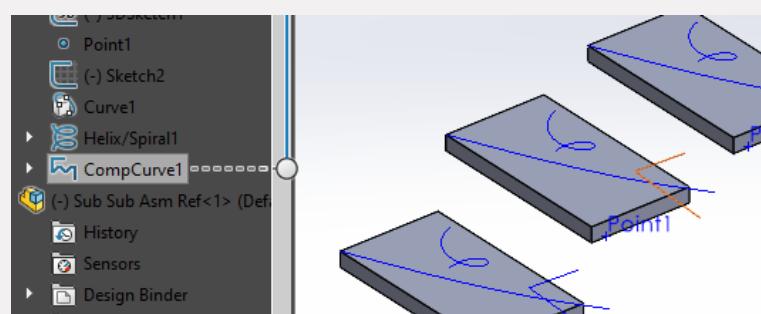
4.6.4.7. 3D Sketches



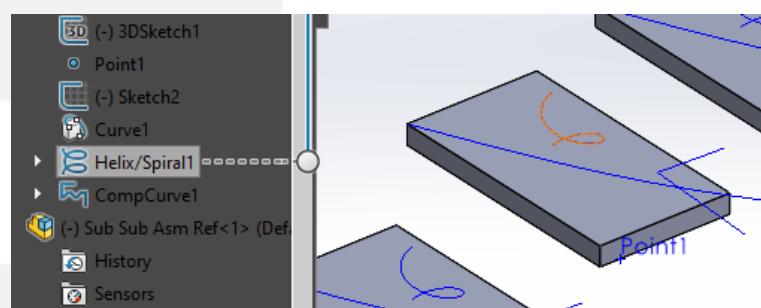
4.6.4.8. Curve thru points



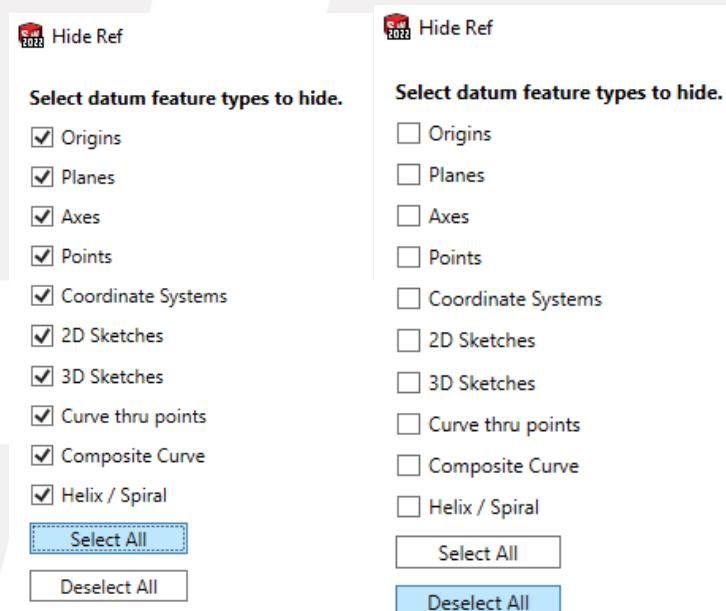
4.6.4.9. Composite Curve



4.6.4.10. Helix / Spiral

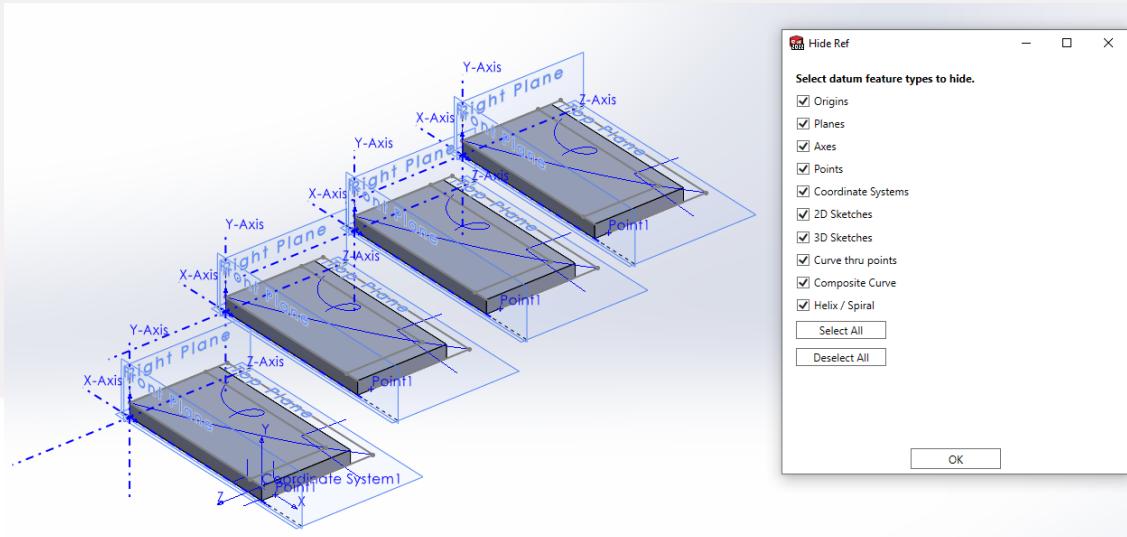


4.6.5. “Select All” button checks all the feature types while “Deselect All” button unchecks all the feature types.

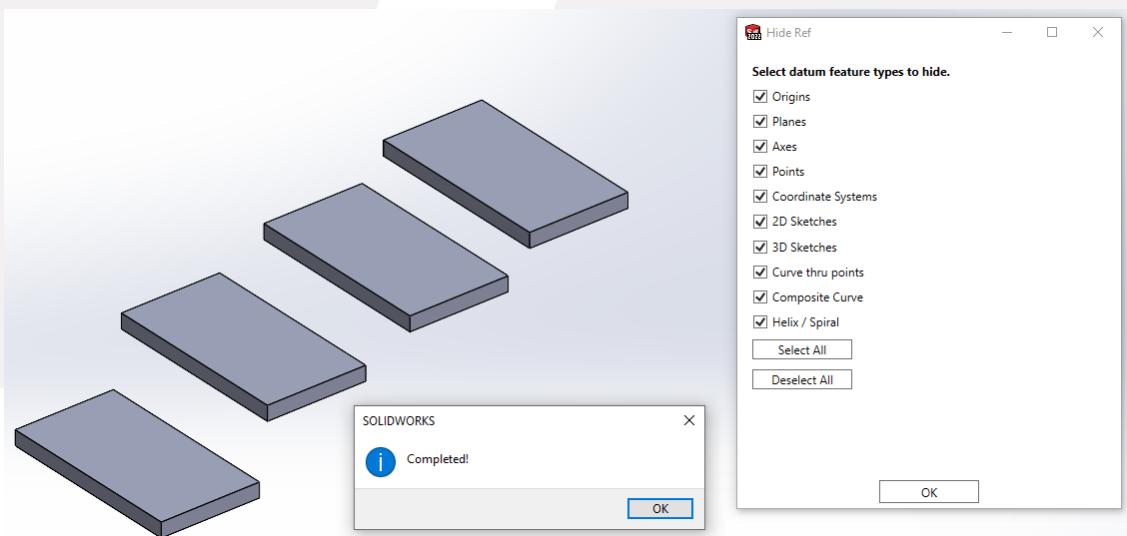


4.6.6. Use the Select All button, Deselect All button and manual check or uncheck to select any feature that you wish to hide. Then Click OK.

4.6.6.1. Before:



4.6.6.2. After:



4.7. Fast Cap



4.7.1. A macro to insert MD 333 library part as derived part into tube. It only supports Part.

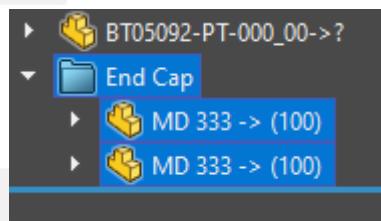
4.7.2. Make sure your MD 333 library part path is set correctly in the setting.



4.7.3. Open your Part that contains tube with 50mm diameter and 3mm thickness.

Then click the macro.

4.7.4. MD 333 will be inserted into the end of the tube accordingly.



4.8. Close Clean



- 4.8.1. This macro will close all opened documents except the dirty one (unsaved).
- 4.8.2. Click the macro and all the saved documents will be closed immediately.



PERTH
(08) 9459 8600

ADELAIDE
(08) 8231 1513

DARWIN
(08) 8963 5621

KALGOORLIE
(08) 9021 8300

NEWMAN
(08) 9175 1164

MT ISA
(07) 4743 6458

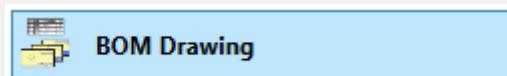
sales@bendtechgroup.com.au

www.bendtechgroup.com.au

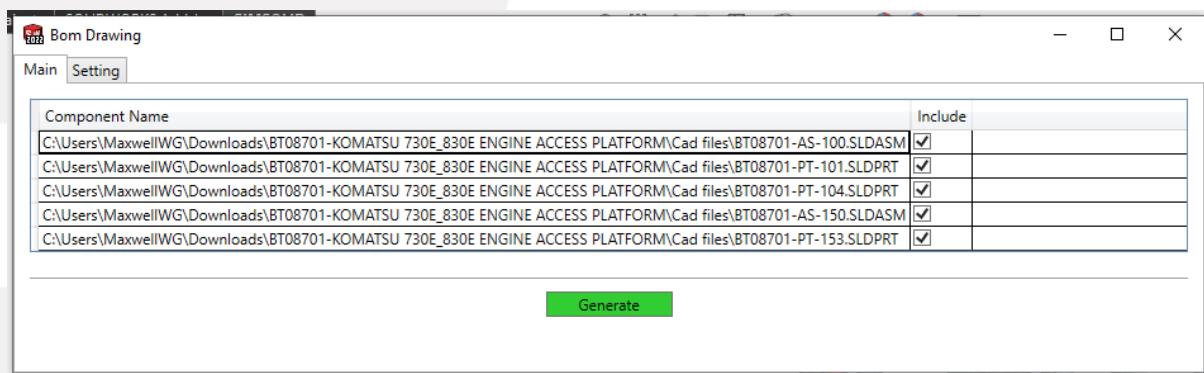
2-10 Kewdale Road, Welshpool WA 6106



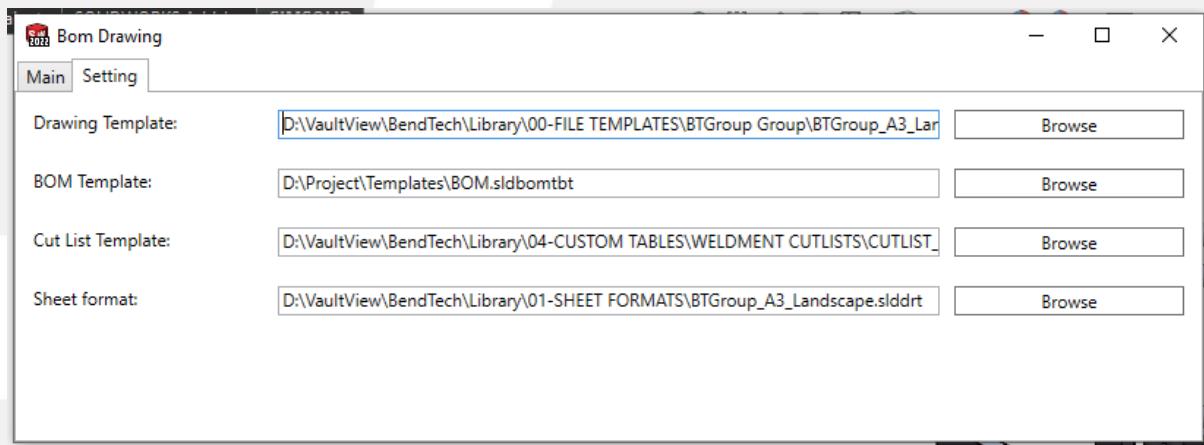
4.9. Bom Drawing



- 4.9.1. Macro that will create a view for each body and sub assembly from an opened assembly or single view if it is a part.
- 4.9.2. Open an assembly or part and click on the macro.
- 4.9.3. An interface will be shown after clicked on the macro. There are 2 tabs, Main tab and Setting tab. User may check on uncheck any component that you which to include or exclude respectively.

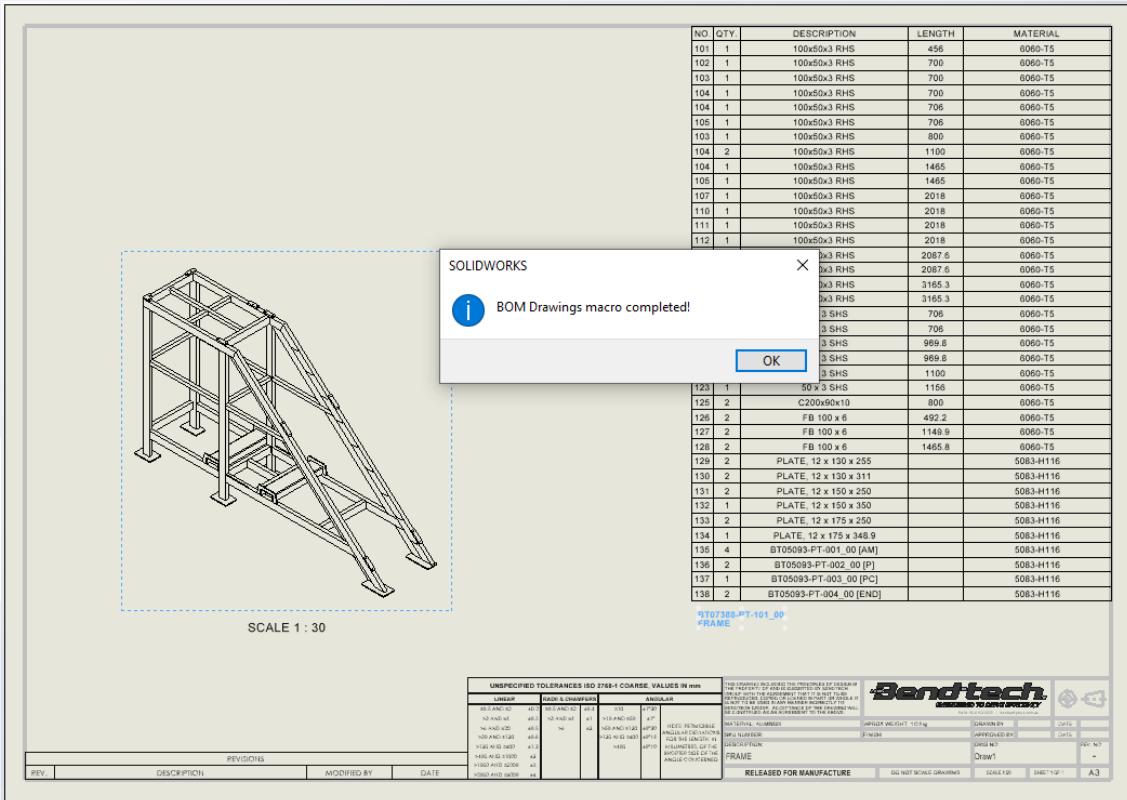


- 4.9.4. Before click generate, make sure the drawing template, BOM template, Cut List Template and Sheet format are not empty at Setting tab.



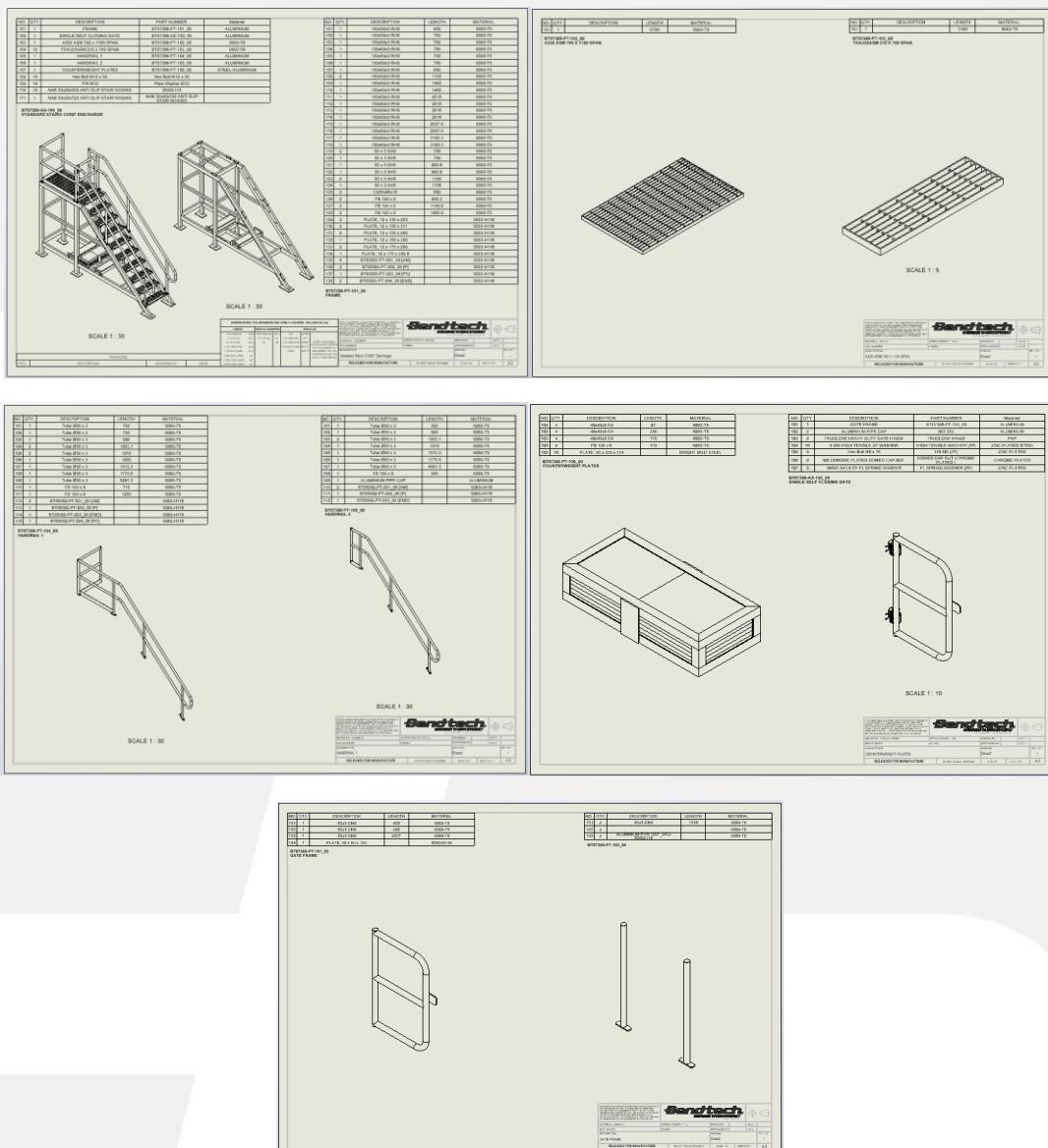
- 4.9.5. Click Generate.

4.9.6. If opened file is a part, a drawing file along with the part's view and cut list table will be created.



4.9.7. If opened file is an assembly, a drawing file will be created with the following views:

- 4.9.7.1. Main assembly view with BOM
- 4.9.7.2. Subassembly view with BOM
- 4.9.7.3. Component view with Cut List Table



- 4.9.8. The macro is designed to create views for any part or assembly that has a naming format of “ProjectNumber-XX-PartNumber”. The macro will skip some of the view that does not follow the naming format.
- 4.9.9. The macro will also skip the model that matches the naming format but having different project number.
- 4.9.10. The macro will use last saved configuration instead of the referenced configuration from assembly if the part has “welded” configuration. Usually, this configuration will be automatically created when the part has the following option turned on:

Document Properties - Weldments

System Options **Document Properties**

Drafting Standard

- Annotations
- Dimensions
- Virtual Sharps
- Tables
- DimXpert

Detailing

Grid/Snap

Units

Model Display

Material Properties

Image Quality

Sheet Metal MBD

Sheet Metal

Weldments (selected)

Plane Display

Configurations

Cut list options

- Automatically create cut lists
- Automatically update cut lists (may affect performance with many bodies)
- Rename cut list folders with Description property value
- Collect identical bodies
- Use English Description property name in Weldment Cut list

Weldment options

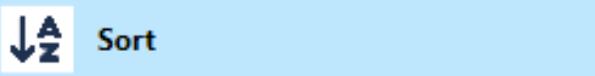
- Create derived configurations (highlighted)
- Assign configuration Description strings

Bounding Box Properties

Solid Bodies Description:

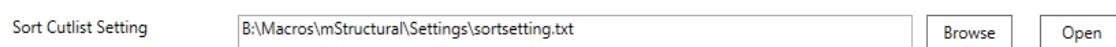
5. Automation (Drawing)

5.1. Sort Cutlist



5.1.1. This macro will sort selected weldment cut list description column according to setting file. This macro only supports drawing file.

5.1.2. Make sure setting file is set correctly (Might be different from picture below).



5.1.3. This macro will sort the weldment cut list according to the setting file (Top to bottom). "*" is wildcard and "%" is digit. For instance, "*rhs" refers to the description that ending with "rhs" (case insensitive); "tube*" refers to the description starting with "tube"; "c%*" refers to the description starting with c and following by a digit; "%%%ub*" refers to the description starting with 3 digits and following by "ub".

```
*rhs
*shs
*chs
tube*
i beam*
angle*
*ea
machine rod*
round bar*
*pfc
000ub*
000uc*
*cds
c0*
fb*
*flat
plate*
checkered plate*
```

5.1.4. Select a weldment cut list by left clicking the top left move symbol of a cut list and run the macro. Below is the result (left before, right after)

NO.	QTY.	DESCRIPTION	LENGTH	MATERIAL	NO.	QTY.	DESCRIPTION	LENGTH	MATERIAL
1	1	60x40x3 RHS	2500	Material <not specified>	1	1	60x40x3 RHS	1759.5	Material <not specified>
2	1	25 x 3 SHS	2569.2	Material <not specified>	2	1	60x40x3 RHS	2105.7	Material <not specified>
3	1	Tube Ø25 x 3	2334.5	Material <not specified>	2	1	60x40x3 RHS	2500	Material <not specified>
4	1	Tube Ø25 x 3	813.9	Material <not specified>	4	1	25 x 3 SHS	1242.3	Material <not specified>
5	1	60x40x3 RHS	2106.7	Material <not specified>	5	1	25 x 3 SHS	1675.7	Material <not specified>
6	1	ROUND BAR Ø20	1643.2	Material <not specified>	5	1	25 x 3 SHS	2569.2	Material <not specified>
7	1	25 x 3 SHS	1676.7	Material <not specified>	7	1	Tube Ø25 x 3	813.9	Material <not specified>
8	1	ROUND BAR Ø20	1253.5	Material <not specified>	8	1	Tube Ø25 x 3	1505.6	Material <not specified>
9	1	25 x 3 SHS	1242.3	Material <not specified>	8	1	Tube Ø25 x 3	2334.5	Material <not specified>
10	1	ROUND BAR Ø20	985.8	Material <not specified>	10	1	I BEAM 101.5 x 76.2	747.2	Material <not specified>
11	1	Machine Rod Ø8	1731.1	Material <not specified>	11	1	I BEAM 101.5 x 76.2	810.3	Material <not specified>
12	1	60x40x3 RHS	1759.5	Material <not specified>	11	1	I BEAM 101.5 x 76.2	1311.5	Material <not specified>
13	1	Machine Rod Ø8	1008.3	Material <not specified>	13	1	ANGLE 16x16x1.6	983.8	Material <not specified>
14	1	I BEAM 101.6 x 76.2	1311.5	Material <not specified>	14	1	ANGLE 16x16x1.6	1270	Material <not specified>
15	1	FB 12 x 3	1248.4	6060-T5	14	1	ANGLE 16x16x1.6	1725.5	Material <not specified>
16	1	I BEAM 101.6 x 76.2	747.2	Material <not specified>	16	1	Machine Rod Ø8	716.6	Material <not specified>
17	1	FB 12 x 3	1318.4	6060-T5	17	1	Machine Rod Ø8	1008.3	Material <not specified>
18	1	200UB29.8	1231.8	300PLUS	17	1	Machine Rod Ø8	1731.1	Material <not specified>
19	1	C200x90x10	1787.6	Material <not specified>	19	1	ROUND BAR Ø20	985.8	Material <not specified>
20	1	PLATE, 10 x 127.6 x 159.8		Material <not specified>	20	1	ROUND BAR Ø20	1253.5	Material <not specified>
21	1	Machine Rod Ø8	716.6	Material <not specified>	20	1	ROUND BAR Ø20	1643.2	Material <not specified>
22	1	Tube Ø25 x 3	1505.6	Material <not specified>	22	1	200UB29.8	960.5	300PLUS
23	1	C200x90x10	1145	Material <not specified>	23	1	200UB29.8	1231.8	300PLUS
24	1	150UC30.0	1863.4	300PLUS	24	1	200UB29.8	2044.5	300PLUS
25	1	I BEAM 101.6 x 76.2	810.3	Material <not specified>	25	1	150UC30.0	1351.7	300PLUS
26	1	PLATE, 10 x 341.1 x 604.6		Material <not specified>	26	1	150UC30.0	1863.4	300PLUS
27	1	C200x90x10	850.2	Material <not specified>	27	1	150UC30.0	2322.5	300PLUS
28	1	PLATE, 3 x 222.1 x 268.4		Material <not specified>	28	1	C200x90x10	850.2	Material <not specified>
29	1	ANGLE 16x16x1.6	1725.5	Material <not specified>	29	1	C200x90x10	1145	Material <not specified>
30	1	150UC30.0	1351.7	300PLUS	29	1	C200x90x10	1787.6	Material <not specified>
31	1	PLATE, 3 x 203.6 x 268.4		Material <not specified>	31	1	FB 12 x 3	613.2	6060-T5
32	1	PLATE, 3 x 231.4 x 472		Material <not specified>	32	1	FB 12 x 3	1248.4	6060-T5
33	1	ANGLE 16x16x1.6	983.8	Material <not specified>	33	1	FB 12 x 3	1318.4	6060-T5
34	1	FB 12 x 3	613.2	6060-T5	34	1	PLATE, 3 x 203.6 x 268.4		Material <not specified>
35	1	200UB29.8	2044.5	300PLUS	35	1	PLATE, 3 x 222.1 x 268.4		Material <not specified>
36	1	PLATE, 10 x 387.6 x 496.1		Material <not specified>	36	1	PLATE, 3 x 231.4 x 249.9		Material <not specified>
37	1	ANGLE 16x16x1.6	1270	Material <not specified>	37	1	PLATE, 3 x 231.4 x 472		Material <not specified>
38	1	PLATE, 10 x 434.6 x 705.4		Material <not specified>	38	1	PLATE, 10 x 127.6 x 159.8		Material <not specified>
39	1	200UB29.8	960.5	300PLUS	39	1	PLATE, 10 x 341.1 x 604.6		Material <not specified>
40	1	PLATE, 10 x 379.8 x 643.4		Material <not specified>	40	1	PLATE, 10 x 344.6 x 590.8		Material <not specified>
41	1	150UC30.0	2322.5	300PLUS	41	1	PLATE, 10 x 379.8 x 643.4		Material <not specified>
42	1	PLATE, 10 x 344.6 x 590.8		Material <not specified>	42	1	PLATE, 10 x 387.6 x 496.1		Material <not specified>
43	1	PLATE, 3 x 231.4 x 249.9		Material <not specified>	43	1	PLATE, 10 x 434.6 x 705.4		Material <not specified>

5.1.5. The weldment cut list will be sorted accordingly. All categories except plate and etc will be sorted according to group followed by length. Plate and etc will be sorted in ascending order.



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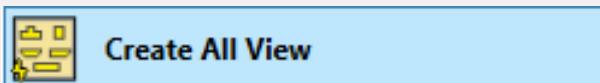
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5.2. Create All View

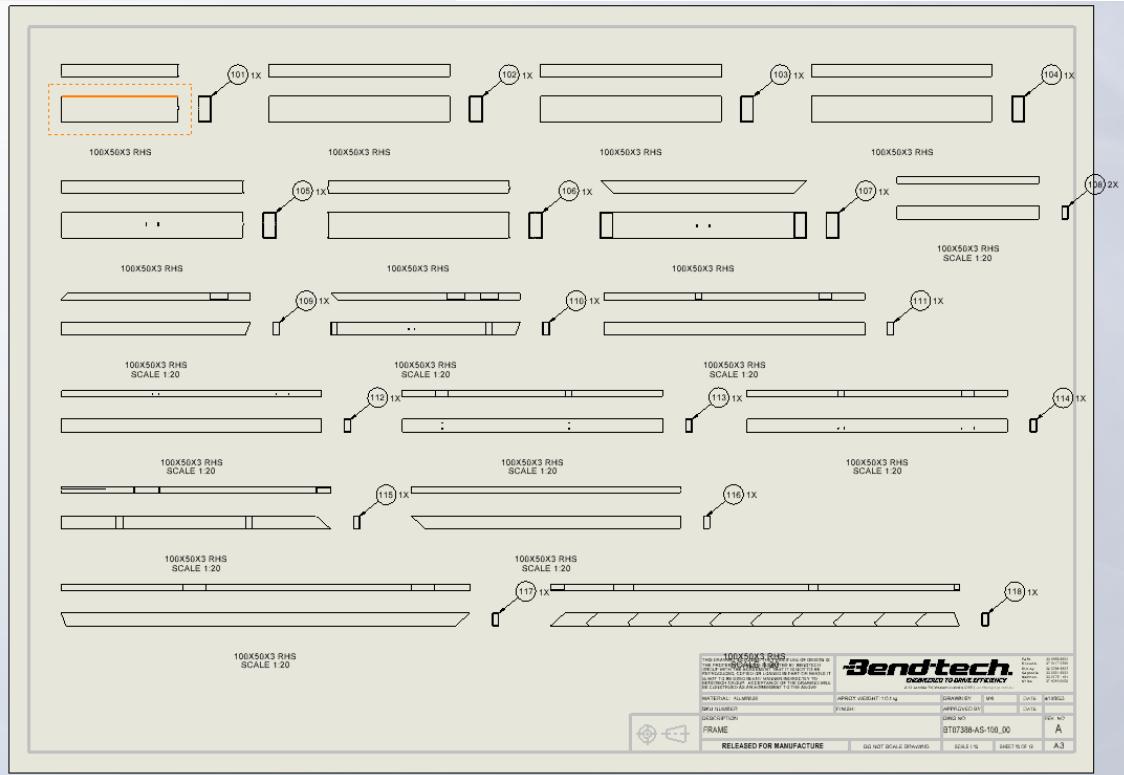


5.2.1. Macro to generate drawing view for each row of the selected weldment cut list. It only supports Drawing document and a weldment cut list must be selected before running the Macro.

5.2.2. Open a drawing and select a weldment cut list. Then, click on the Create All View button.

A	B	C	D	E
NO.	QTY.	DESCRIPTION	LENGTH	MATERIAL
1	2	460UB67.1	3000	GRADE 300 STEEL AS 3679.1
2	12	PLATE, 16 x 90.8 x 358		GRADE 350 STEEL AS 3678
3	2	PLATE, 16 x 90.8 x 426.6		GRADE 350 STEEL AS 3678
4	2	PLATE, 20 x 100 x 512.9		GRADE 350 STEEL AS 3678
5	2	200.0x100.0x9.0 RHS	144.9	GRADE 350 STEEL AS 1163
6	4	150UC37.2	854.4	GRADE 300 STEEL AS 3679.1
7	2	PLATE, 10 x 350 x 441		GRADE 350 STEEL AS 3678
8	3	200UC46.2	931.5	GRADE 300 STEEL AS 3679.1
9	4	PLATE, 16 x 150 x 181		GRADE 350 STEEL AS 3678
10	2	75.0x75.0x4.0 SHS	1163.4	GRADE 350 STEEL AS 1163
11	4	75.0x75.0x4.0 SHS	553.6	GRADE 350 STEEL AS 1163
12	4	PLATE, 10 x 328.3 x 363.5		GRADE 350 STEEL AS 3678
13	2	PLATE, 10 x 260 x 1430		GRADE 350 STEEL AS 3678
14	2	PLATE, 10 x 507.1 x 1130		GRADE 350 STEEL AS 3678
15	2	PLATE, 10 x 260 x 1130		GRADE 350 STEEL AS 3678
16	2	PLATE, 16 x 147 x 259.4		GRADE 350 STEEL AS 3678
17	2	PLATE, 16 x 147 x 217.2		GRADE 350 STEEL AS 3678
18	2	PLATE, 20 x 490 x 1254		GRADE 350 STEEL AS 3678
19	2	PLATE, 10 x 507.1 x 1430		GRADE 350 STEEL AS 3678
20	2	PLATE, 16 x 190 x 454		GRADE 350 STEEL AS 3678

5.2.3. This Macro will create a drawing view for each row of the selected weldment cut list together with top and right projected view. New drawing sheets will be added to the drawing and the views will be generated in sequence from left to right and top to bottom. Once the drawing sheet is filled, a new sheet will be created, and the remaining drawing view will be generated on that sheet. The process will be repeated until all rows of the weldment cut list has been processed.



5.2.4. A note linked to the description of the cut list body also will be added to the parent view.

5.2.5. A balloon will be added to the right view.

5.2.6. Once completed, a message will be prompted to indicate all views have been generated successfully.

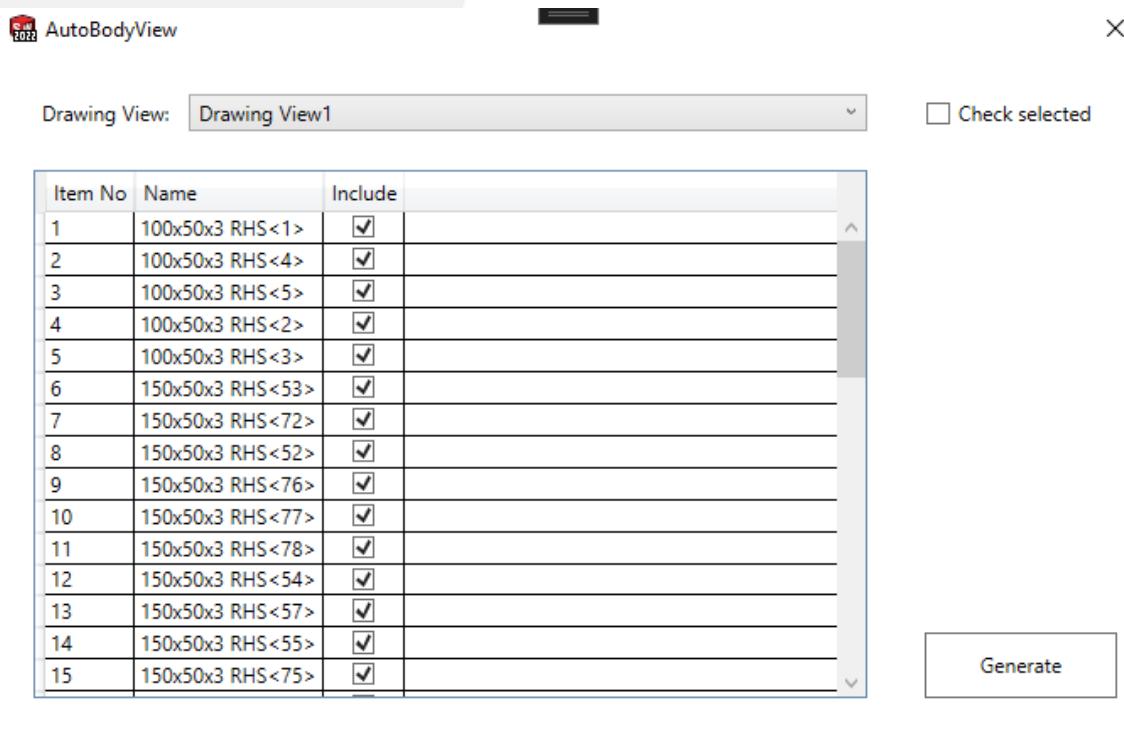
5.3. Auto View



5.3.1. To generate drawing views for each row of the selected weldment cut list. It only supports drawing files and a weldment cut list must be selected before running the Macro.

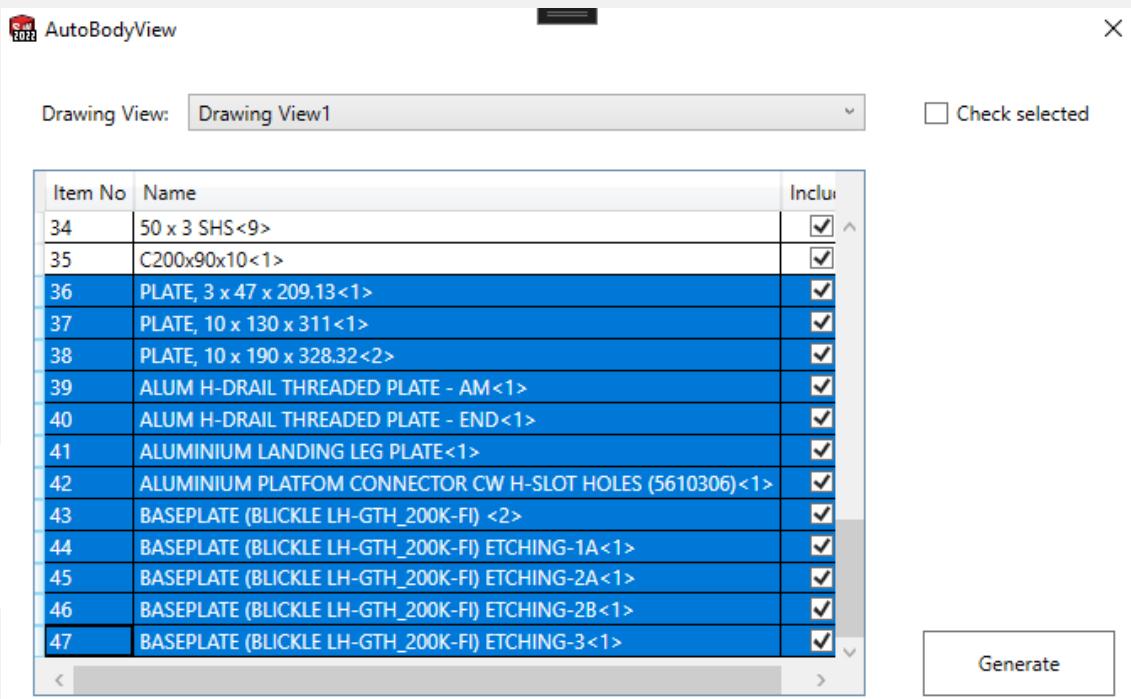
5.3.2. Video: <https://youtu.be/RU7rS64NSao>

5.3.3. Select a weldment cut list and Auto View Macro button. A window will be prompted for the user to select which item to generate view.

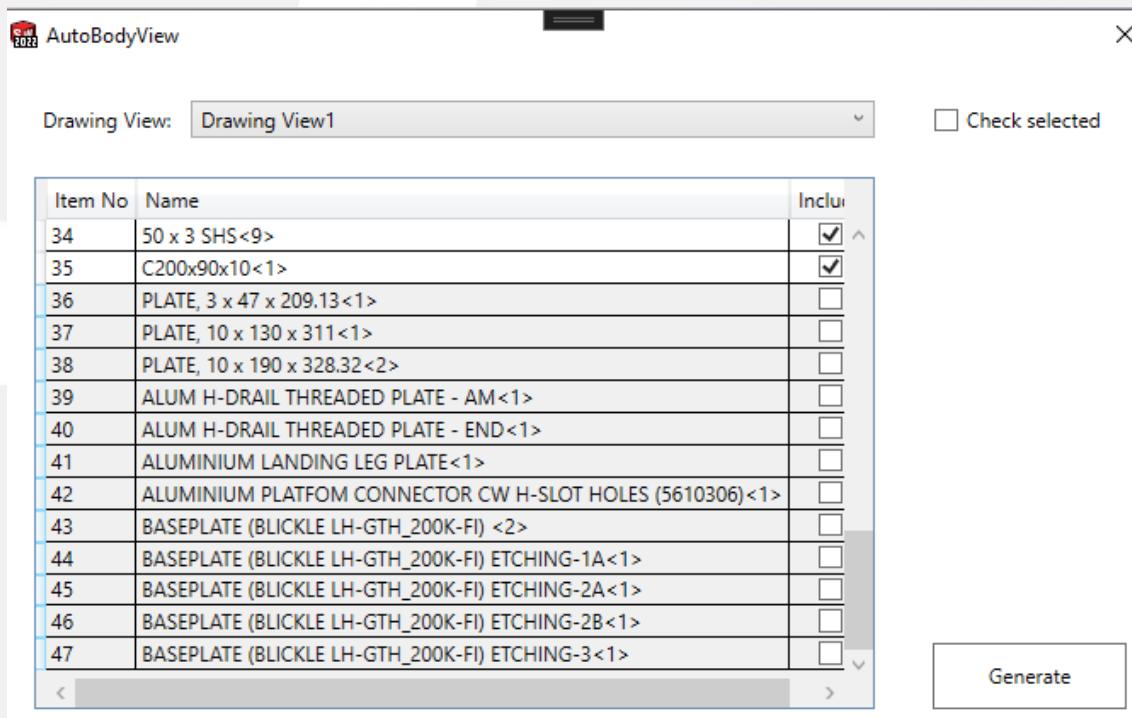


5.3.4. If the list is empty or the list is different from your cut list, try to select a different drawing view from the Drawing View dropdown.

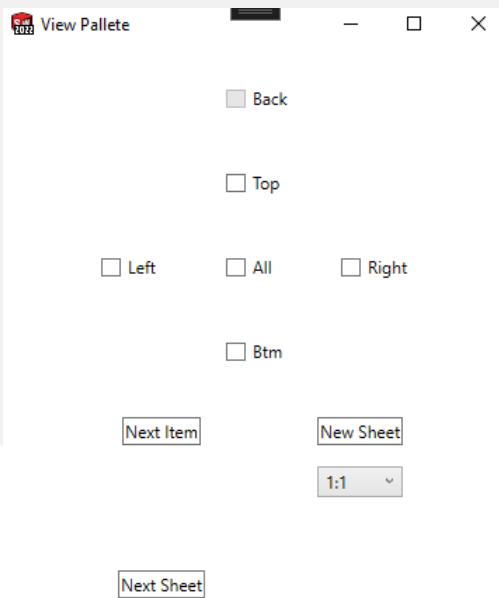
5.3.5. You can multi-select items using shift or ctrl button same as how you perform selection in window environment.



5.3.6. If you wish to exclude certain items, multi-select them and click twice on the "Check selected" to exclude the selected item.



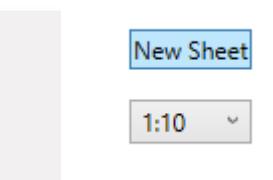
- 5.3.7. Click on the Generate button to create a view.
 5.3.8. Another window will be prompted to manipulate the generated view.



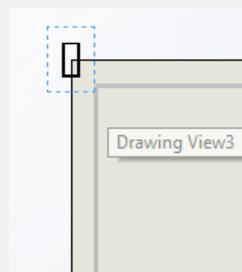
- 5.3.9. All generated views will be placed at the top left corner of your drawing sheet.

NO.	QTY.	DESCRIPTION
1	1	100x50x3 RHS
2	1	100x50x3 RHS
3	1	100x50x3 RHS
4	3	100x50x3 RHS
5	1	100x50x3 RHS
6	1	150x50x3 RHS
7	1	150x50x3 RHS
8	2	150x50x3 RHS

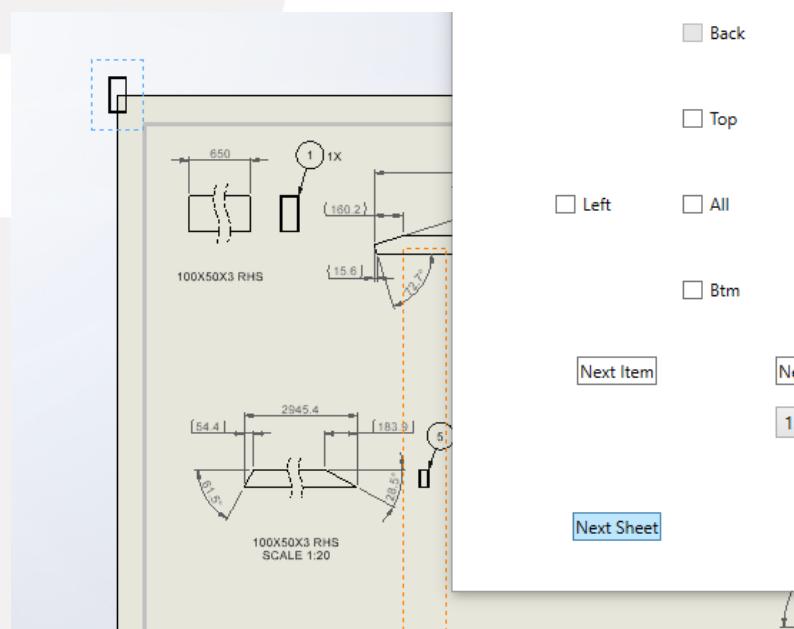
- 5.3.10. To move it to the new sheet, select your desired sheet format and click on the New Sheet button.



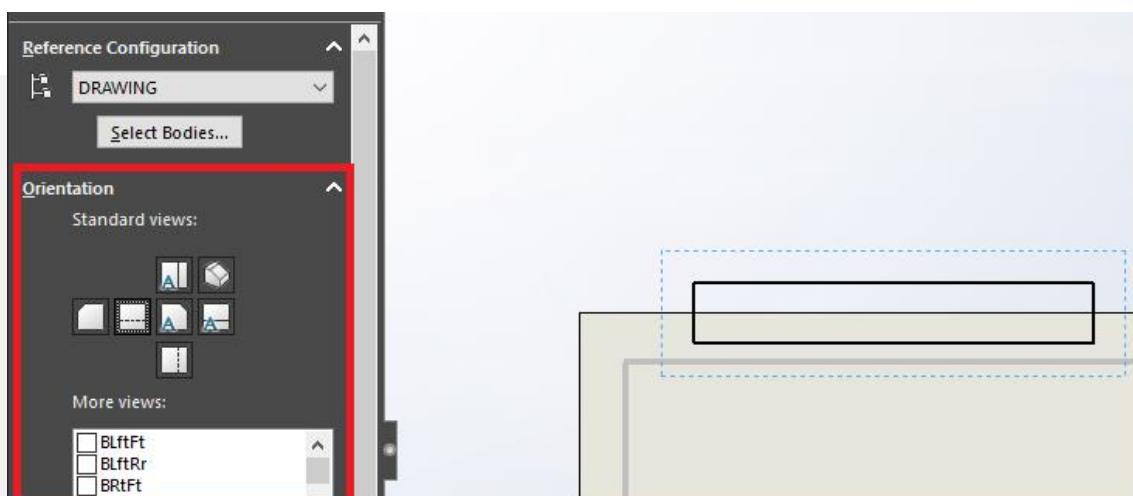
5.3.11. A new sheet will be created, and the view will be deleted and recreated on this sheet.



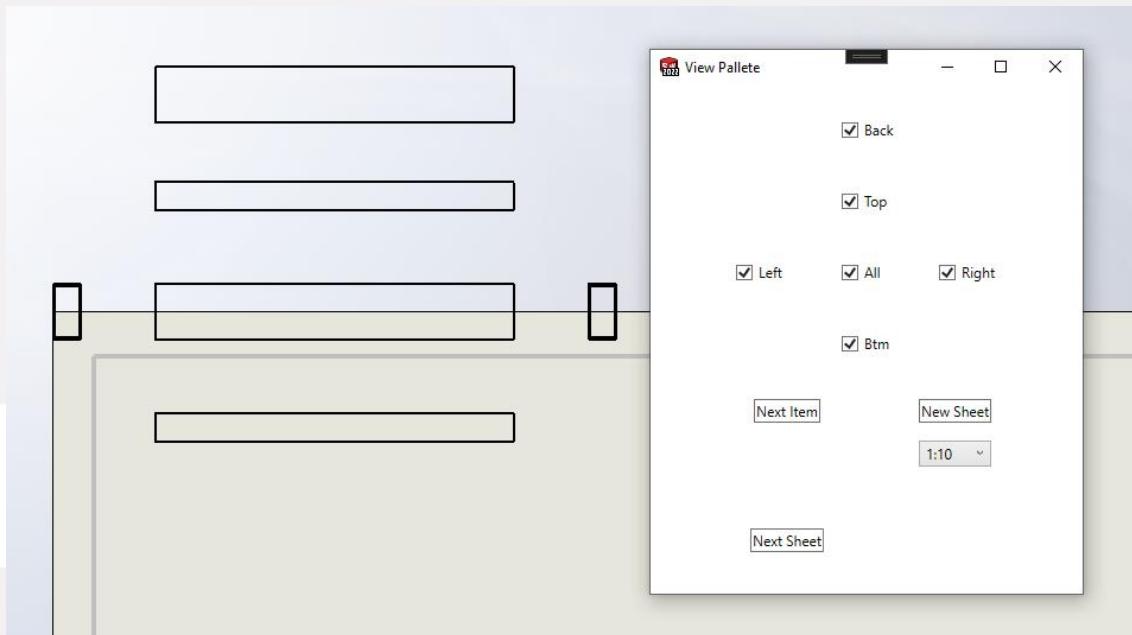
5.3.12. To move it to the next sheet, click on the Next Sheet button. The view will be deleted and recreated on the next sheet.



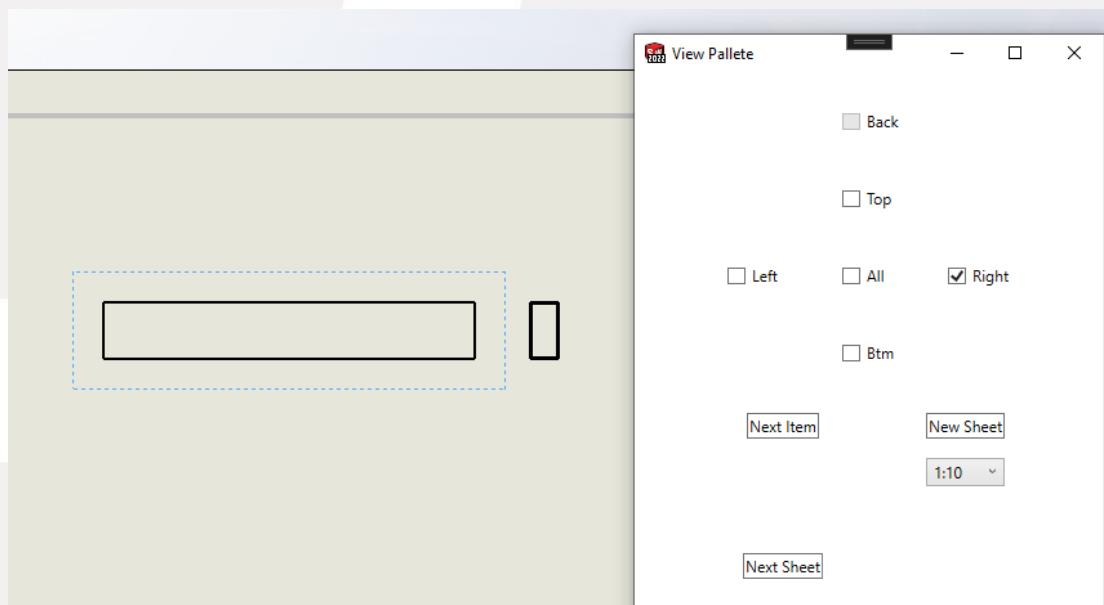
5.3.13. Try to choose the best parent view from SolidWorks function before proceeding.



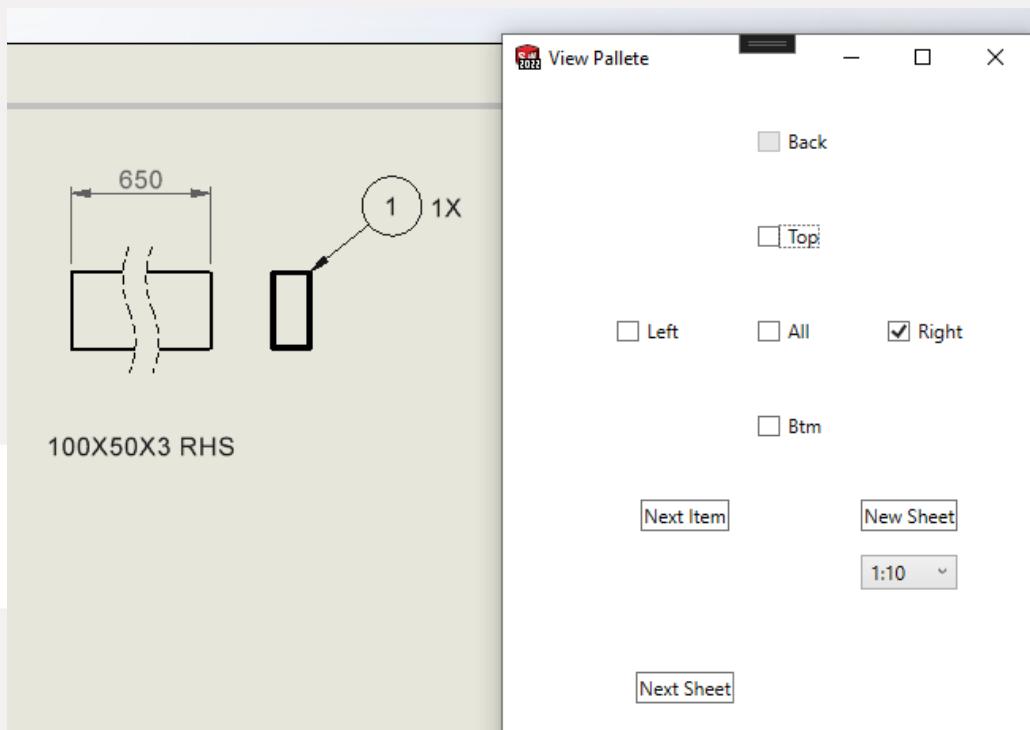
5.3.14. Once selected the best parent view, click on All checkbox to preview all the possible views that may be useful to present your dimension.



5.3.15. Uncheck any of the redundancy views and position the view accordingly.

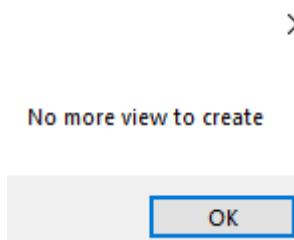


5.3.16. Detail the view using standard SolidWorks function or using another Macro from BTG SOLIDWORKS Add-In.

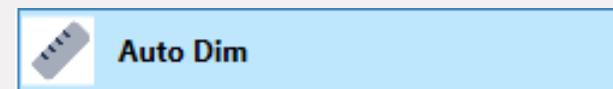


5.3.17. Click Next Item button to create another view.

5.3.18. Once all view has been processed, a message will be shown to indicate that no more view left to process.



5.4. Auto Dim

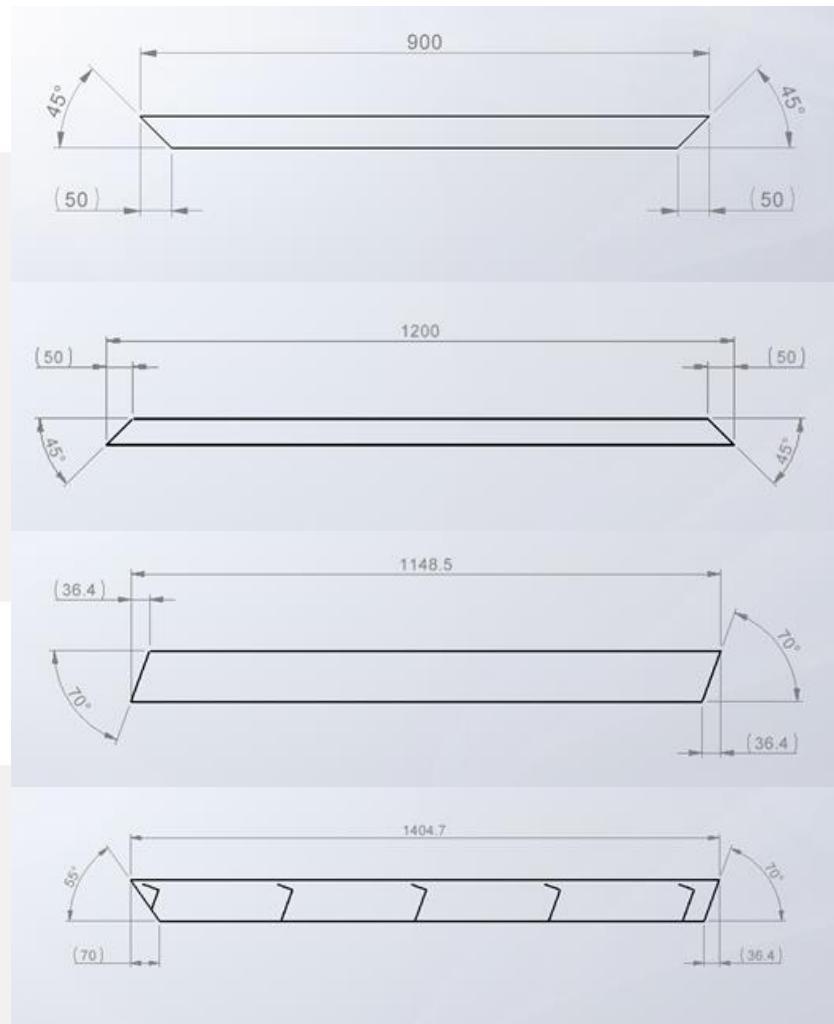


5.4.1. A Macro to dimension weldment member automatically. It only supports drawing files, and a view must be selected before running the Macro.

5.4.2. Supported profile:

- 5.4.2.1. Straight
- 5.4.2.2. Single angle
- 5.4.2.3. Dual angle
- 5.4.2.4. Tab from side view
- 5.4.2.5. Tab from top view

5.4.3. Single angle at both sides



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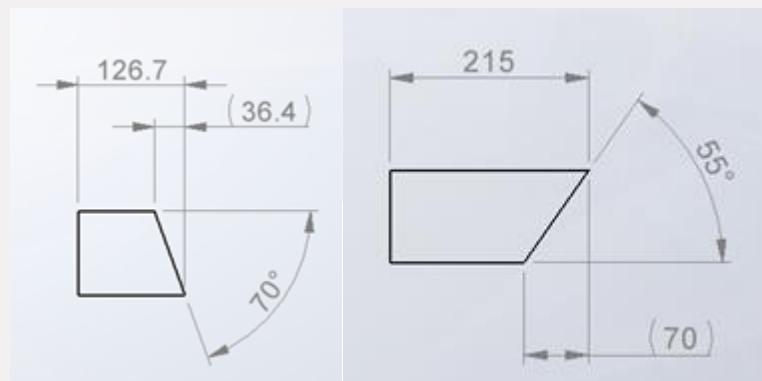
sales@bendtechgroup.com.au

www.bendtechgroup.com.au

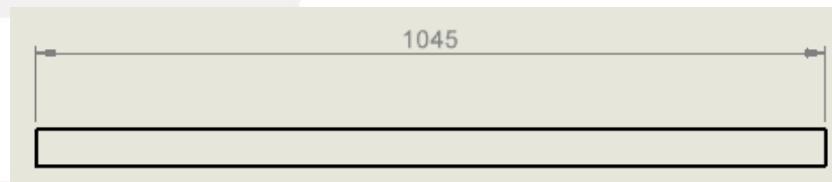
2-10 Kewdale Road, Welshpool WA 6106



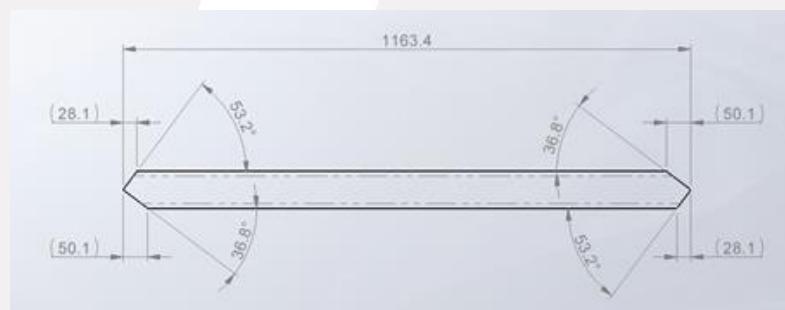
5.4.4. Single angle at one side and straight at another.



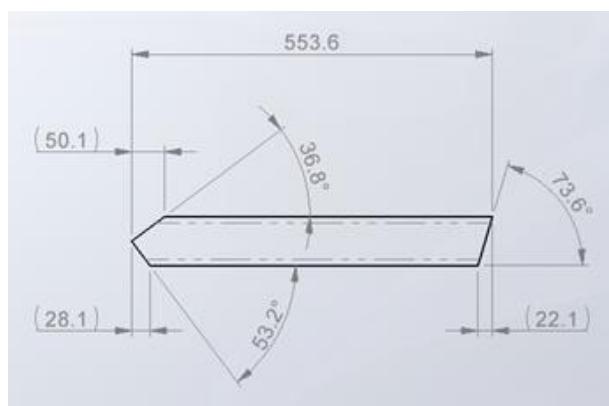
5.4.5. Straight at both side



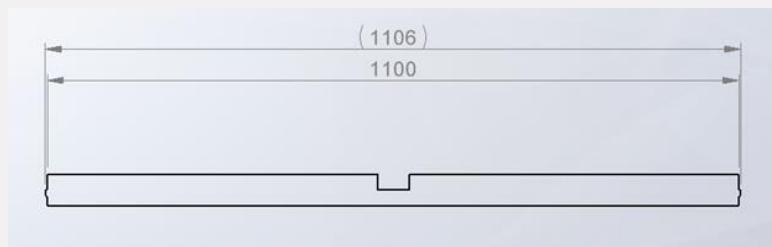
5.4.6. Dual angle at both side



5.4.7. Dual angle at one side, single angle at one side



5.4.8. Tab from side view.



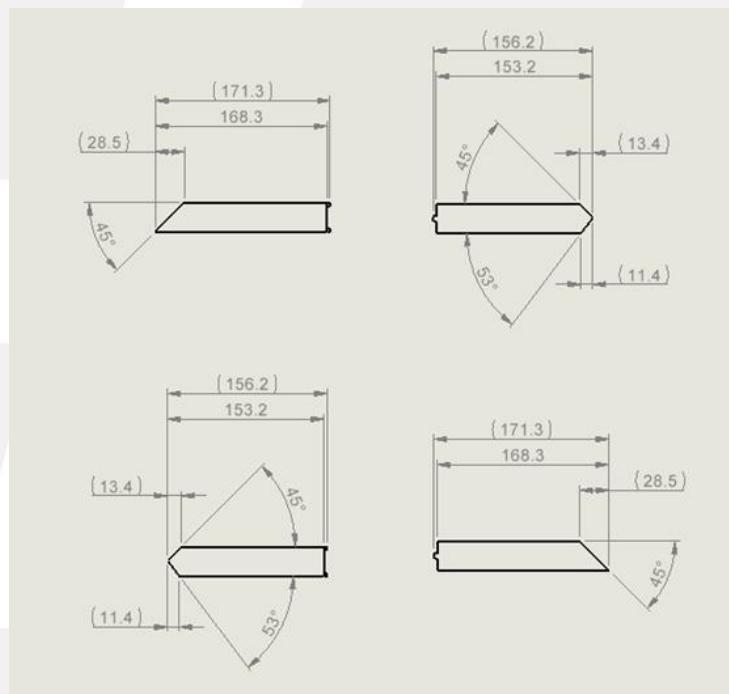
5.4.9. Tab from top view.



5.4.10. Tab one side and straight other side



5.4.11. Tab one side and single or dual angle other side



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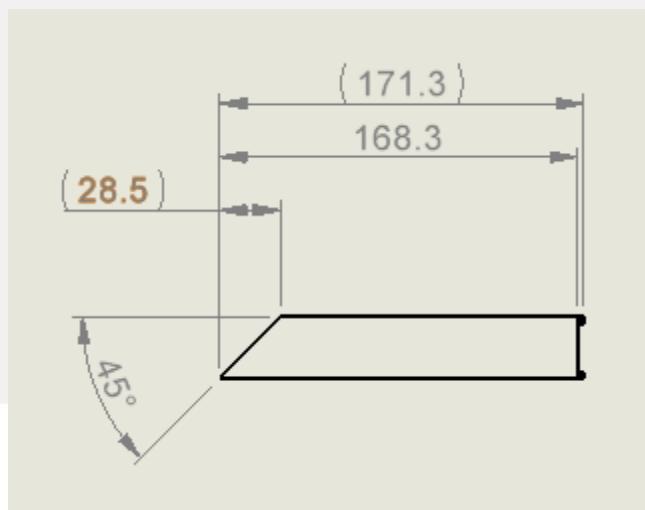
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- 5.4.12. Open a drawing and select a view. Then click on Auto Dim button at BTG SOLIDWORKS Add-In task pane.
5.4.13. Dimension will be added automatically for supported type.



5.5. Balloon

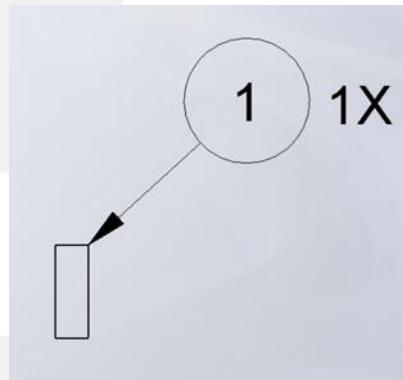


Balloon

5.5.1. A Macro to add balloon to a view automatically. It only supports drawing files, and a view must be selected before running the Macro.

5.5.2. Open a drawing and select a view. Then click on Auto Dim button at BTG SOLIDWROKS Add-In task pane.

5.5.3. Balloons with quantity will be added automatically to the selected view.



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5.6. Quick Note

Aa Quick Note

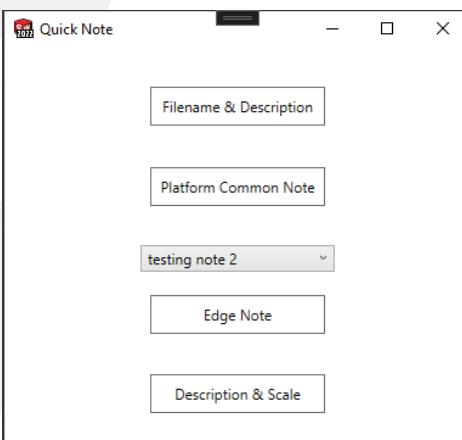
5.6.1. A Macro to quickly create notes on drawing view or entities. It only supports drawing files.

5.6.2. Set the list of string for edge note Macro in setting. Each string is separated by ";" symbol.

Edge Note

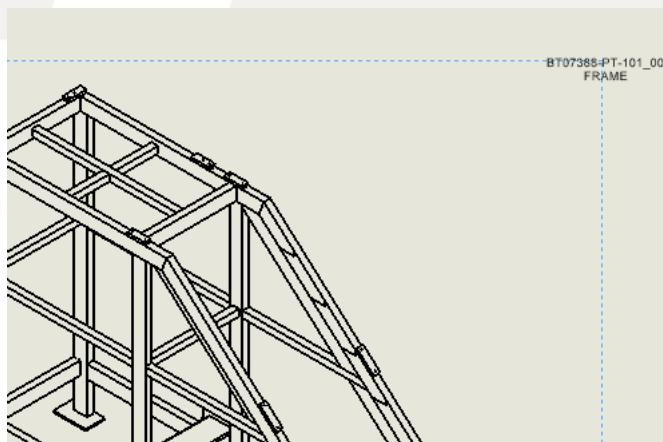
ETCHING;ETCHING (TYP);LINE ETCHING; LINE ETCHING (TYP);SLOT;2x SLOTS

5.6.3. Click on Quick Note button, another window will be prompted.



5.6.4. For the first button, "Filename & Description", select any view (can be multiple) and press the button.

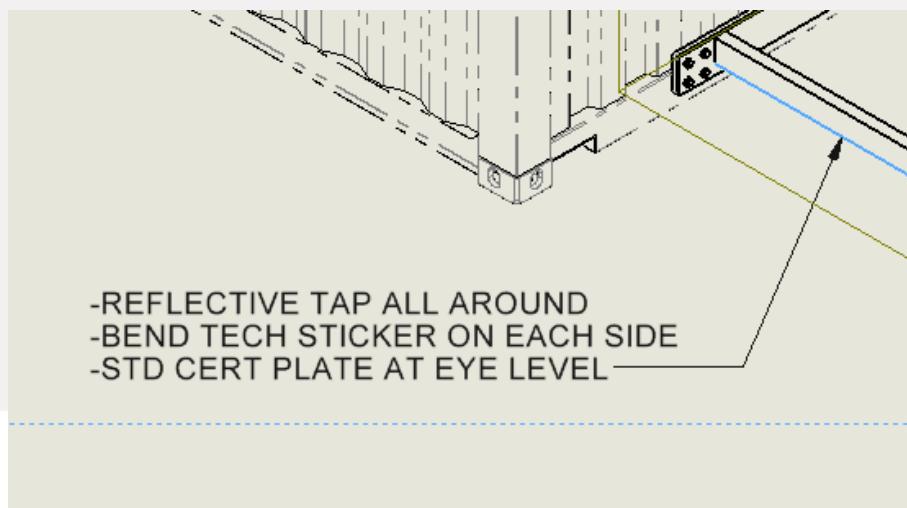
5.6.5. A note will be created at the top right of the view outline.



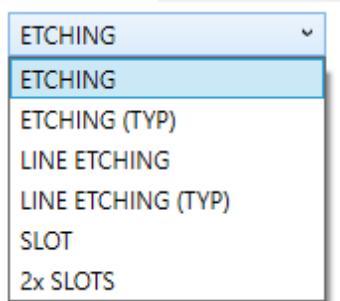
5.6.6. The content is linked to file:

`$PRPVIEW:"SW-File Name(File Name)"
$PRPVIEW:"Description"`

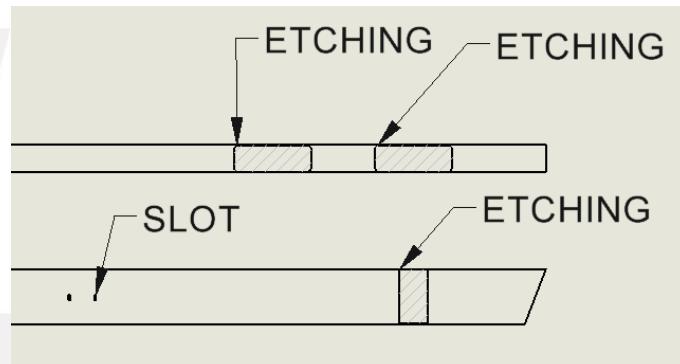
- 5.6.7. For the second button, "Platform Common Note", select any edge or vertex and press the button.
- 5.6.8. A note with a common platform description will be attached to the edge.



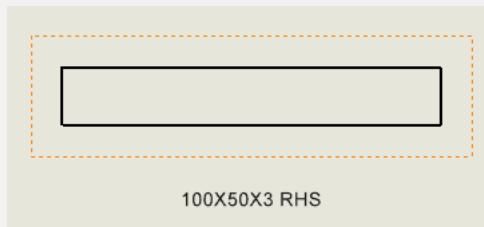
- 5.6.9. For the third button, "Edge Note", choose a text from the dropdown before proceeding.



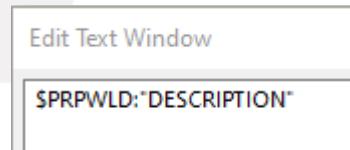
- 5.6.10. The list of text in the dropdown is populated from setting in step1.
- 5.6.11. try to select any edge or vertex (can be multiple) and press the button.
- 5.6.12. A note with ladder will be created with default position and attached to the selected entity.



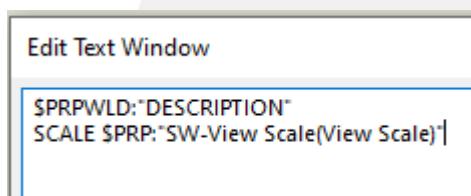
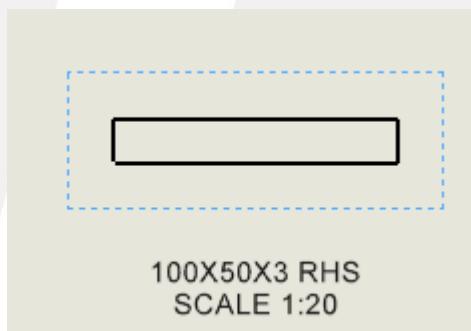
5.6.13. For the fourth button, "Description & Scale", select a view and click on the button.



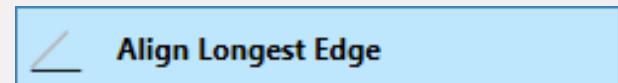
5.6.14. A note with following content will be created at the middle bottom of the view:



5.6.15. If the view does not use sheet scale, another note which linked to scale will be added at the bottom of description:



5.7. Align Longest Edge

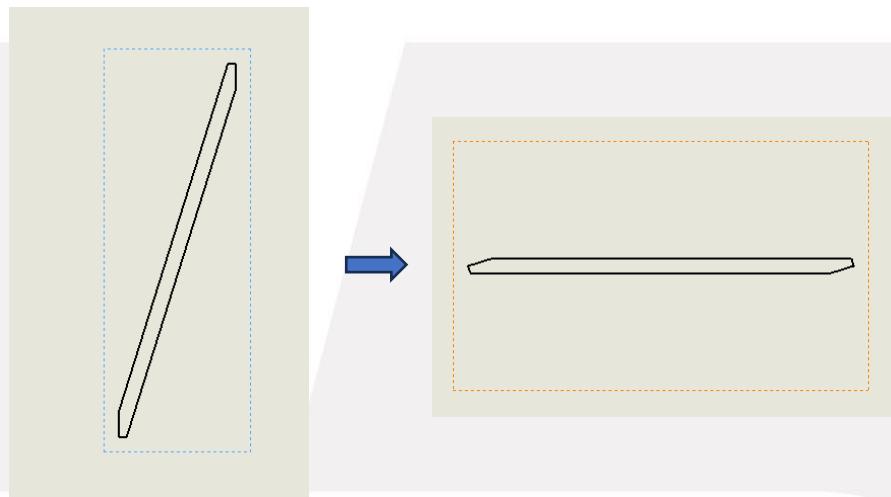


5.7.1. To align the drawing view horizontal to the longest edge. It only supports drawing files, and a view must be selected before running the Macro.

5.7.2. Select a view and click on Align Longest Edge Macro button.



5.7.3. The view will rotate according to the angle between the longest edge and horizontal.



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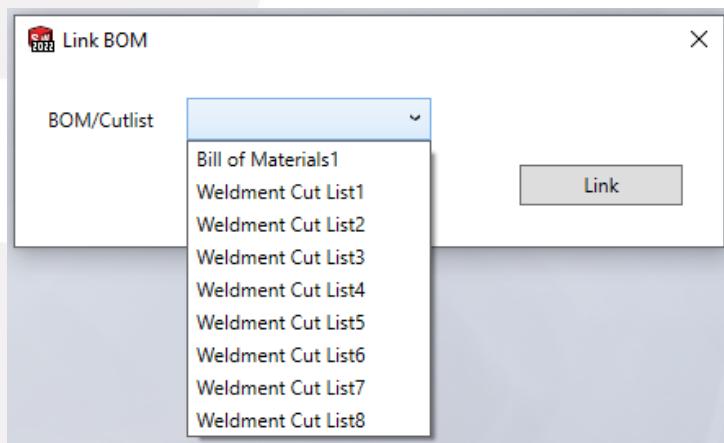
MT ISA
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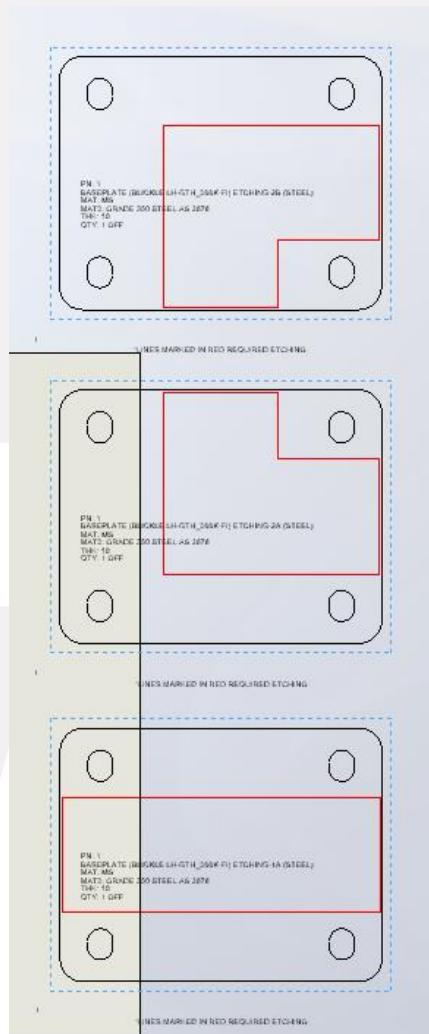
5.8. Link BOM



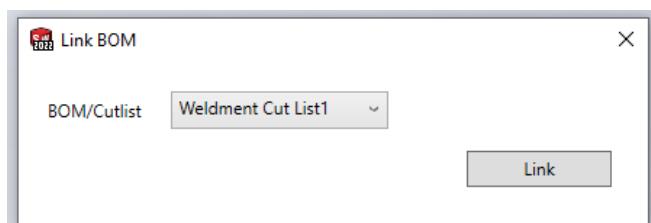
- 5.8.1. Link the balloon text of all the selected views to the BOM or Cut list selected in the interface. It only supports drawing files.
- 5.8.2. Open any drawing that you wish to link the balloon text to a specific BOM or Cut list.
- 5.8.3. Click on the Link BOM Macro button, an interface with a BOM / Cut list dropdown will appear on your screen. The dropdown lists all the BOM and Cut list in the drawing.



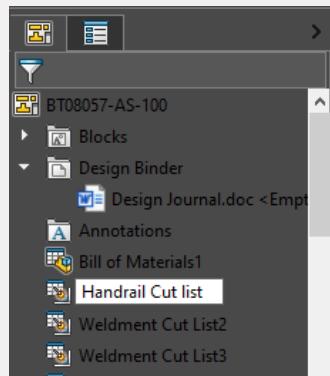
5.8.5. Select the views that you want to link the balloon text to a specific BOM or Cut list. You may use CTRL button to perform multi select.



5.8.6. Select the BOM or Cut list you want to link with all these selected views.

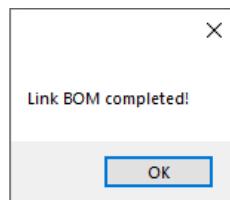


5.8.8. Tips: For ease of selection, you may rename the BOM or Cut list at feature manager design tree

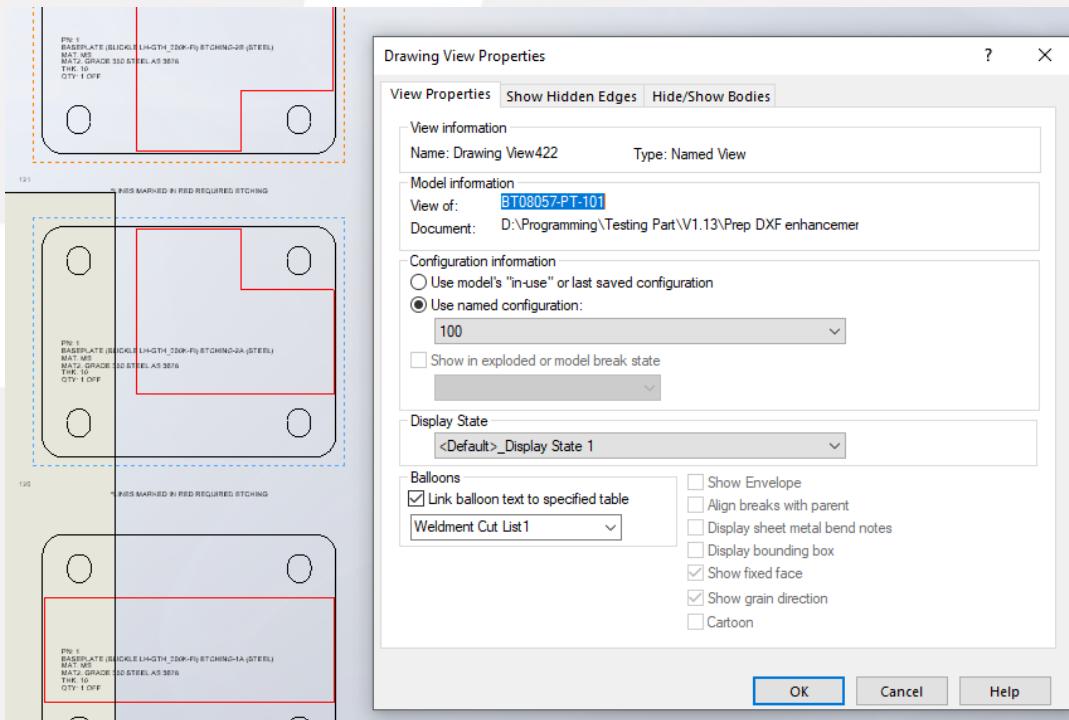


5.8.9. Click the Link button.

5.8.10. A message will be prompted to indicate the operation is completed. Click OK.



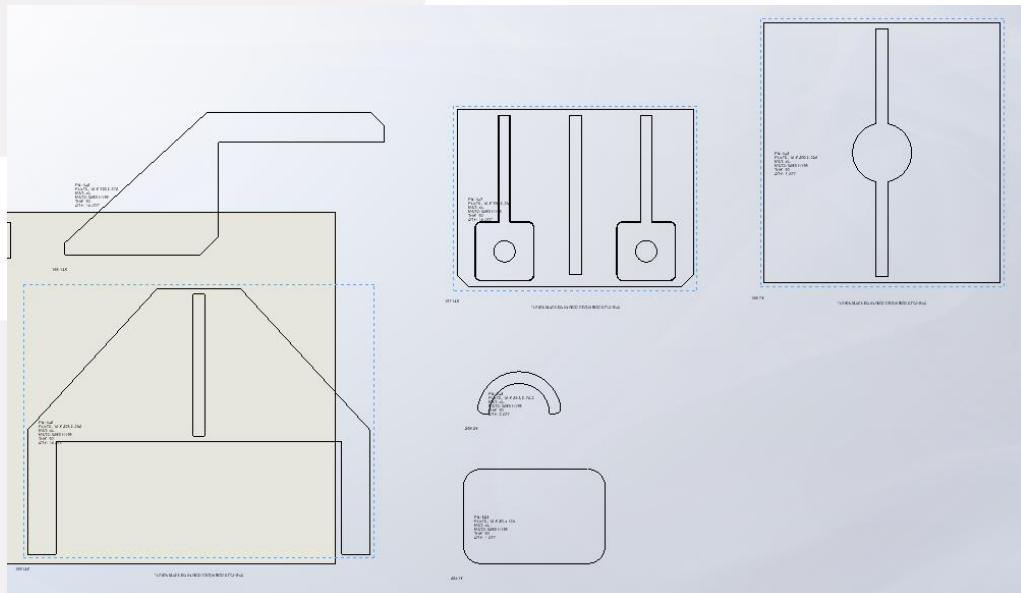
5.8.11. Check the properties of the drawing view, you can see that the balloons are now linked with the selected BOM or Cut list.



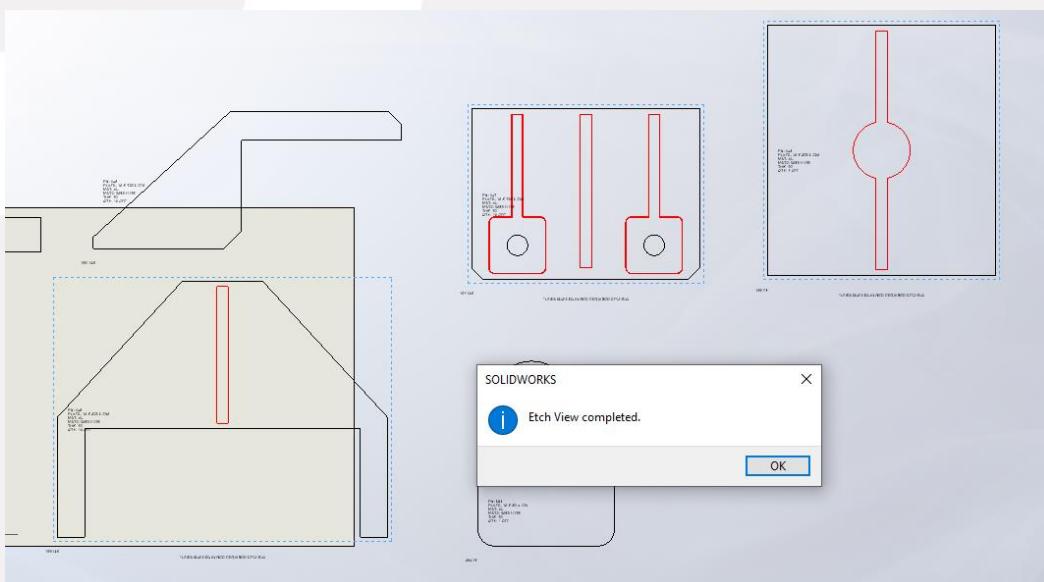
5.9. Etch View



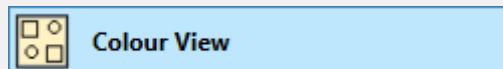
- 5.9.1. A Macro to colour any visible edge of the selected view that has “etch” in model name. You can mark the edge by using [Etch Feature Macro](#). This macro only supports Drawing document.
- 5.9.2. Open a drawing and select any view that you want to colour. Multi-select is allowed.



- 5.9.3. Click on the Etch View button, any visible edge that has “etch” in the model’s name will be coloured red.

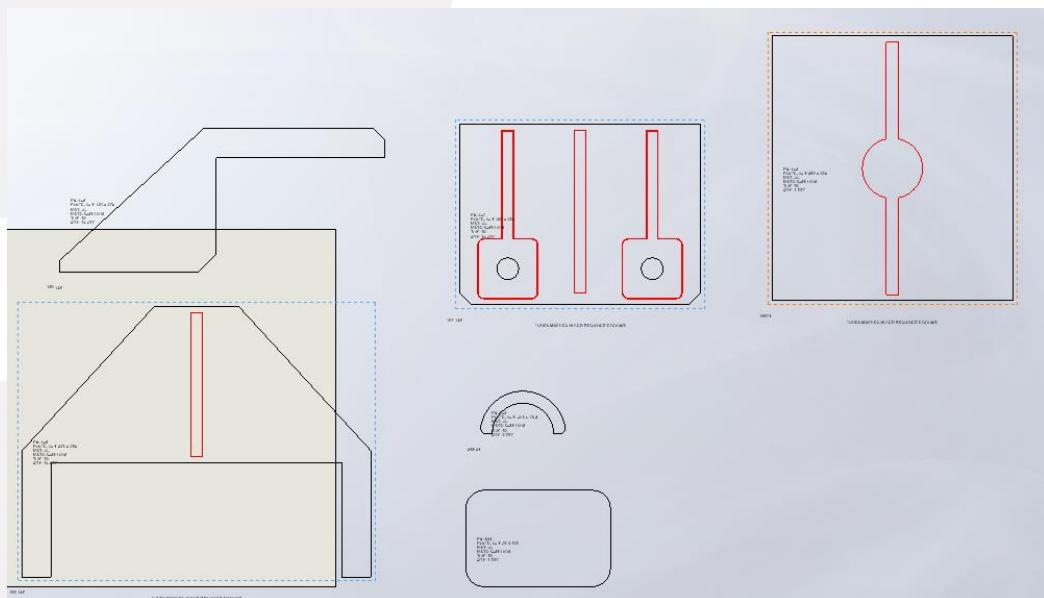


5.10. Colour View

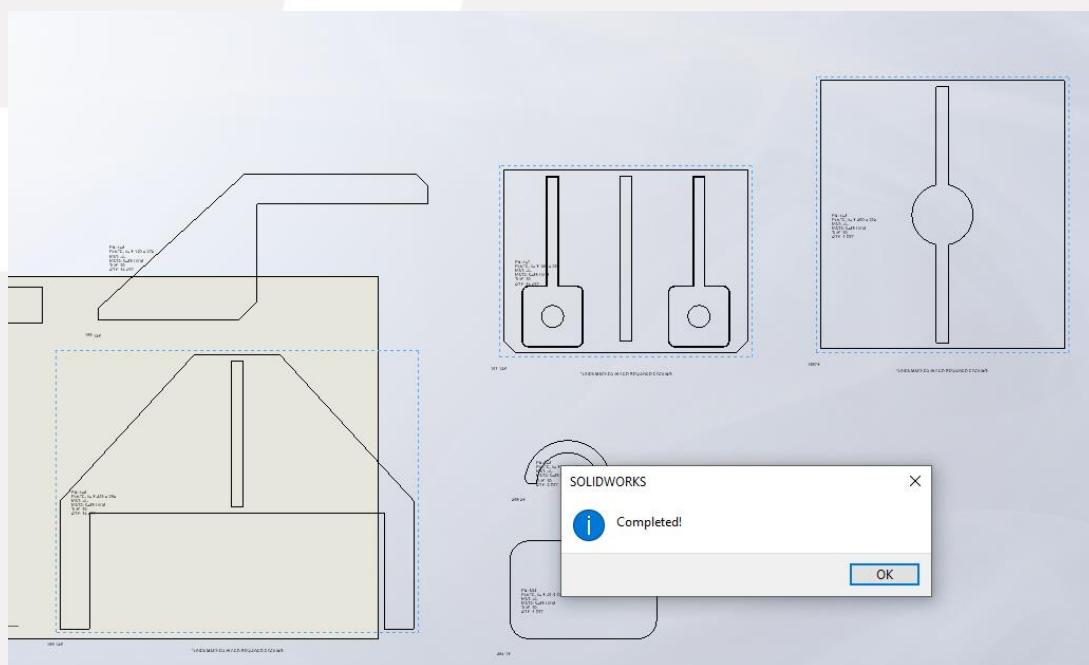


5.10.1. A Macro to colour all visible edges in the selected view to black colour. This Macro only supports Drawing document.

5.10.2. Open a drawing and select any view that you want to run the Macro. Multi-select is allowed.



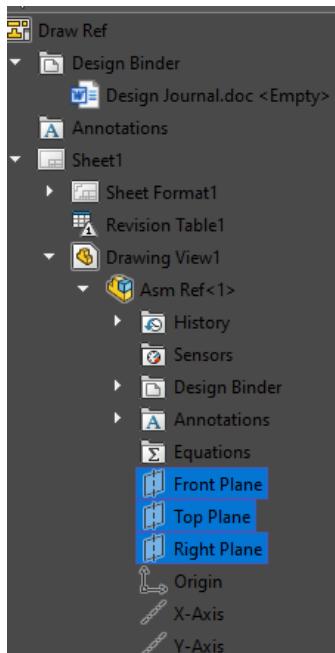
5.10.3. Click on Colour View button, all visible edges in the selected view are coloured black.



5.11. Hide Ref (Drawing)



- 5.11.1. A macro to hide any selected feature found in the active document. This section explains the Macro when running in Drawing document. For Part or Assembly document, refer to [Hide Ref \(Model\)](#).
- 5.11.2. Open any document that you want to hide specific features and click on the Hide Ref Macro button at Automation (Drawing) session.
- 5.11.3. An interface listing all the supported feature types will be shown. Refer to [Hide Ref \(Model\)](#) for supported feature type.
- 5.11.4. Use the Select All button, Deselect All button and manual check or uncheck to select any feature that you wish to hide. Then Click OK.
- 5.11.5. The Macro will traverse through all the sheets, and then views to go through all the feature tree and hide any feature with matching type and hide it.



5.12. Auto Generate DXF Sheets

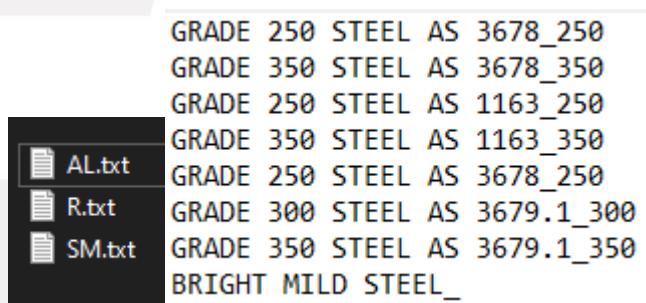


5.12.1. This macro generates all DXF sheets for all the unique bodies that contain “PLATE,” “RUBBER,” or “RUBBER SEAL,” in the name of the selected view model. This macro only supports drawing files.

5.12.2. Set configuration file location at setting.

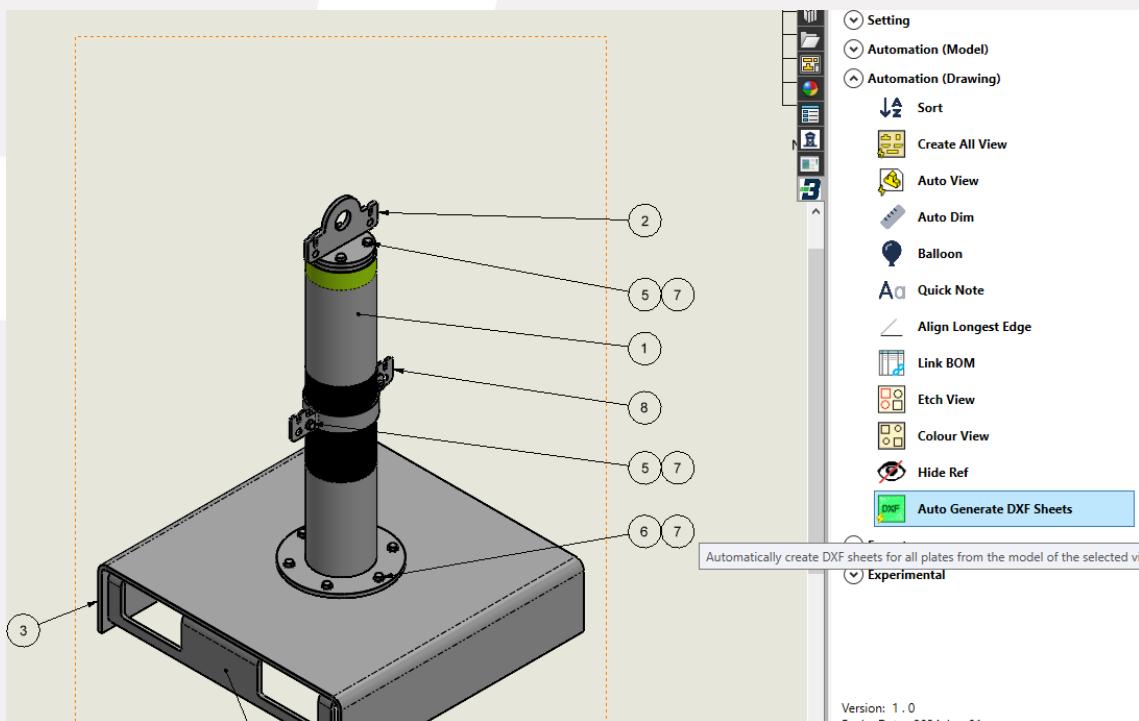


5.12.3. This setting should point to a folder path which contains the material configuration.

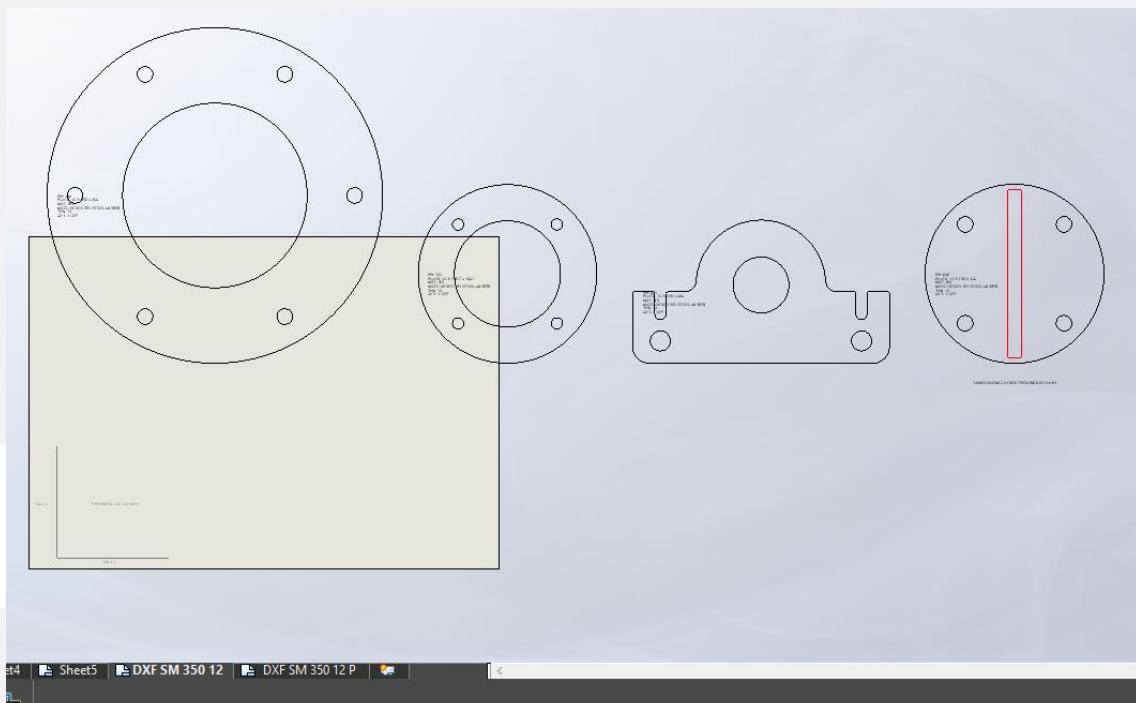


5.12.4. Open the drawing that you want to generate DXF sheets.

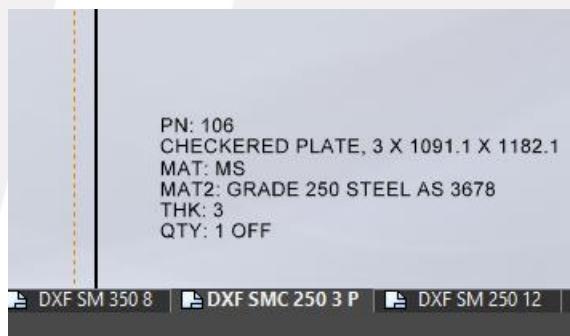
5.12.5. Select the view of the main assembly and click on “Auto Generate DXF Sheets” button.



5.12.6. All bodies that meet the criteria will have a view generated at the end of the drawing.

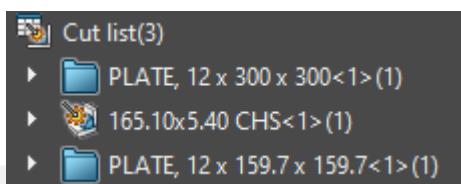


5.13. If the plate is "CHECKERED PLATE", "C" will be added at the back of material code.



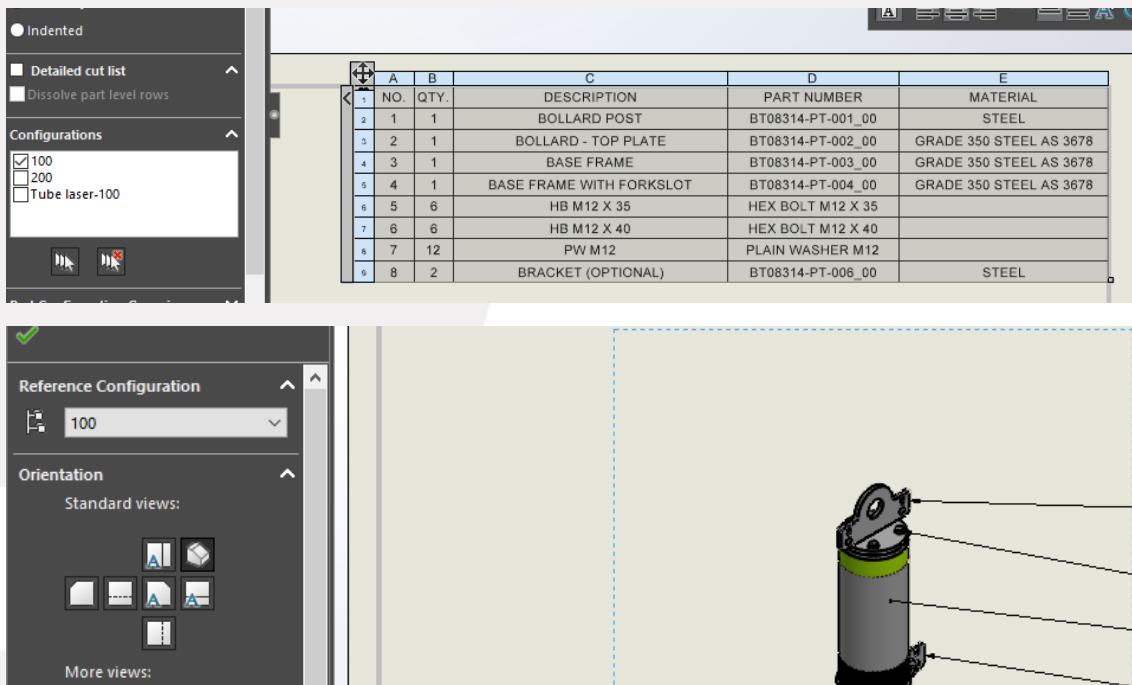
5.14. If failed to generate the DXF or view, double check the following:

5.14.1. Make sure the body folder contains either "PLATE," "RUBBER," or "RUBBER SEAL," (case insensitive).

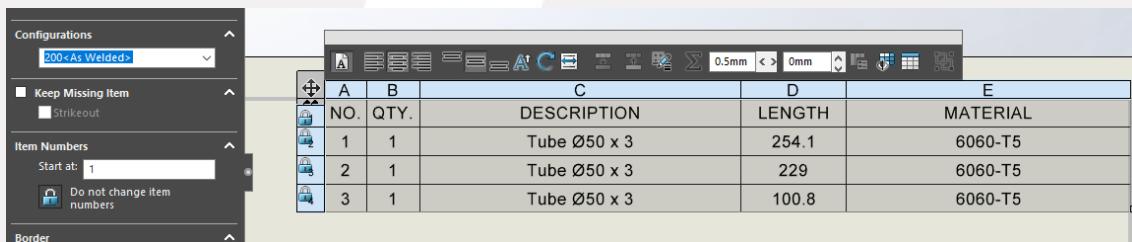


5.14.2. Make sure the drawing file contains all the BOM or Cut List. This macro does not support split drawing files (etc: BT99999-AS-001_00.SLDDRW, BT99999-AS-001_01.SLDDRW).

5.14.3. Table configuration and view configuration need to be the same.



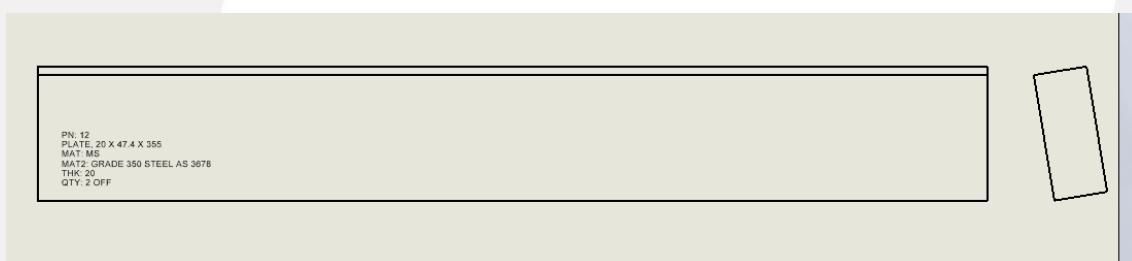
5.14.4. "As Welded" configuration needs to be deleted.



5.15. There are few known issues for the view created which might need manual adjustment from the user.

5.15.1. If there is a countersunk hole on the plate that you do not want to show it, flip the view manually.

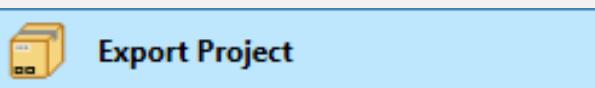
5.15.2. If there is a model that is created in an angled orientation, the view created will be in an angled orientation as well.



5.15.3. Any solid body created using Sheet Metal feature will be considered as Pressed Plate. You may need to move the view manually to a different sheet.

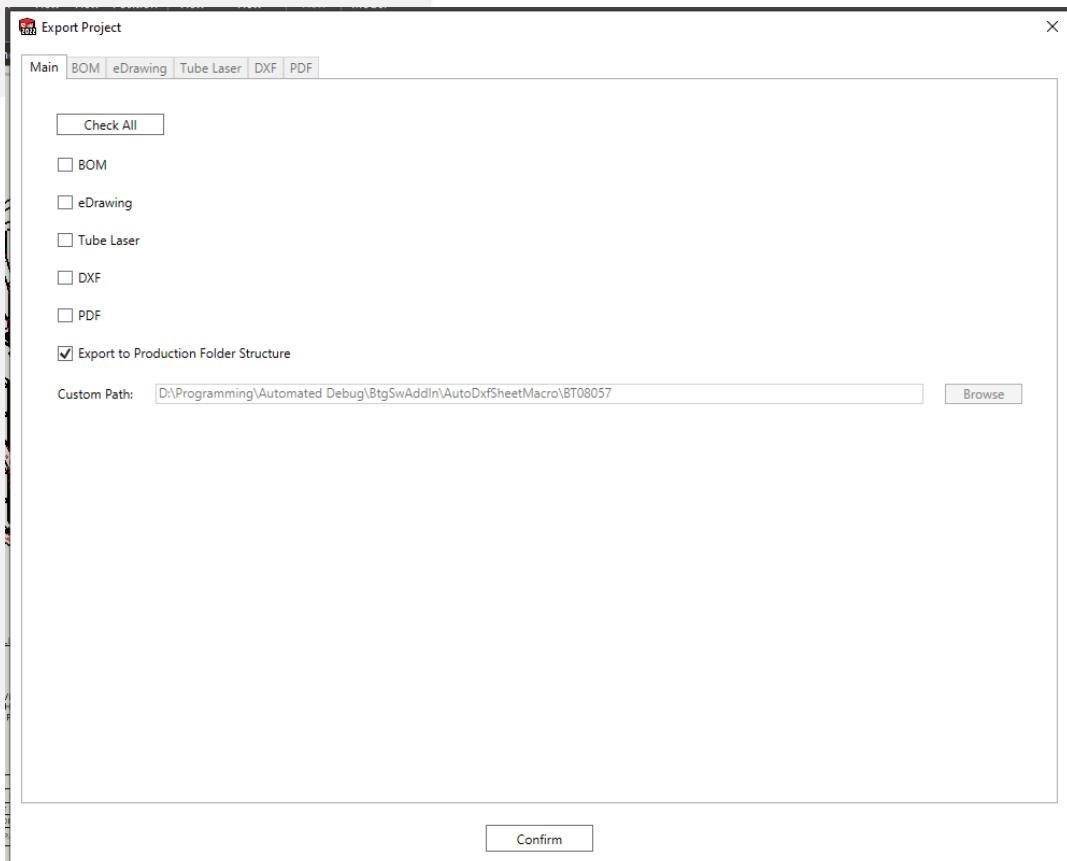
6. Export

6.1. Export Project

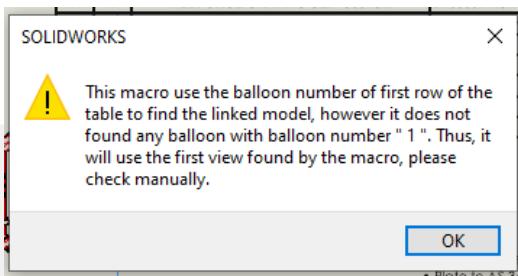


6.1.1. A Macro that exports BOM, eDrawing, Tube Laser, DXF and PDF. It only supports Drawing document.

6.1.2. Select the main assembly view in the drawing and click on the Export Project button. An interface will be prompted for the user to modify there are 6 tabs, namely Main, BOM, eDrawing, Tube Laser, DXF and PDF.



6.1.3. This macro uses the balloon number to link between the view and table. If one of your views does not have a balloon number of the first row in the same page as the table, this macro will show warning.



6.1.4. Main tab:



- 6.1.4.1. Check All button: Click this to check all the BOM, eDrawing, Tube Laser, DXF and PDF checkboxes.
- 6.1.4.2. BOM checkbox: Check this to export BOM.
- 6.1.4.3. eDrawing checkbox: Check this to export model as eDrawing.
- 6.1.4.4. Tube Laser checkbox: Check this to export model as Tube Laser STEP file.
- 6.1.4.5. DXF checkbox: Check this to export DXF.
- 6.1.4.6. PDF checkbox: Check this to export PDF.
- 6.1.4.7. Export to Production Folder Structure checkbox: Check on this if you want the exported file following folder structure (..\production). If uncheck it will save to custom path. You can set the default setting for this option in setting.
- 6.1.4.8. Custom Path textbox: Uncheck Export to Production Folder Structure to activate this field. This field represents a custom path if do not want to export to production folder structure. By default, it will point to the same folder as the active drawing file.
- 6.1.4.9. Browse button: A button to browse to a folder path for custom path textbox.
- 6.1.4.10. If you are running this macro on the file inside PDM or the Custom Path is pointing to PDM location, make sure the directory exists beforehand.



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2-10 Kewdale Road, Welshpool WA 6106



6.1.6. BOM tab:

Main	BOM	eDrawing	Tube Laser	DXF	PDF
Table Name	First Balloon No.	Rename Body?	Tube Laser Configuration		
Bill of Materials1	1	<input type="checkbox"/>	in use	▼	
Weldment Cut List1	101	<input checked="" type="checkbox"/>	tube laser	▼	
Weldment Cut List2	201	<input checked="" type="checkbox"/>	tube laser	▼	
Weldment Cut List3	301	<input checked="" type="checkbox"/>	tube laser	▼	
Weldment Cut List7	4	<input checked="" type="checkbox"/>	Default	▼	
Weldment Cut List4	401	<input checked="" type="checkbox"/>	Default	▼	
Weldment Cut List5	501	<input checked="" type="checkbox"/>	Default	▼	
Weldment Cut List6	601	<input checked="" type="checkbox"/>	TUBE LASER	▼	
Weldment Cut List8	10	<input checked="" type="checkbox"/>	100	▼	

6.1.6.1. Table Name column: Column listing all the BOM or Weldment cut list.

6.1.6.2. First Balloon No. column: Column showing the first balloon number in that table.

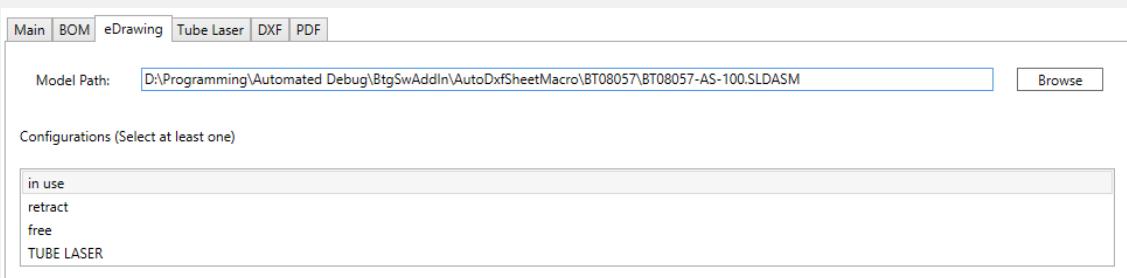
6.1.6.3. Rename Body? column: Checkbox column for you to choose whether you want to rename the body of the reference model of that cut list. This only support for weldment cut list, all checkboxes on BOM will be unchecked by default. The body will be renamed following the format:

ModelName_BalloonNo_Instance



6.1.6.4. Tube Laser Configuration column: Dropdown column for you to choose which configuration you want to run the body rename macro.

6.1.8. eDrawing Tab:

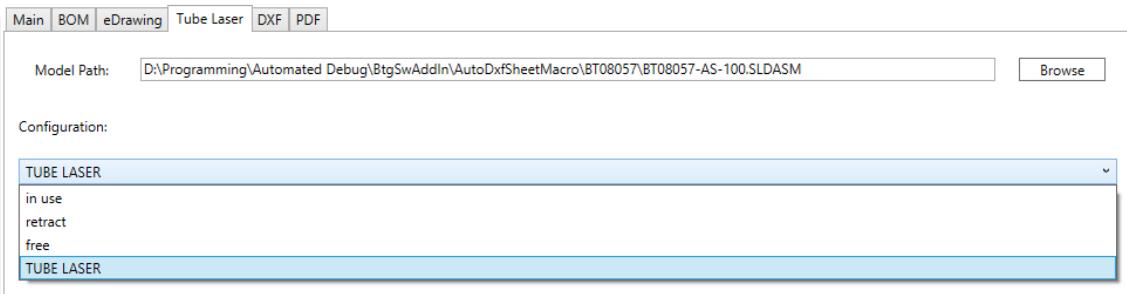


6.1.8.1. Model Path textbox: The path of the model to export as eDrawing for this project. It will show the path to the selected view model by default.

6.1.8.2. Browse button: A button to browse to the model to export, only supports Part or Assembly file.

6.1.8.3. Configuration listbox: A listbox to select which configuration to export. Must select at least one configuration.

6.1.9. Tube Laser Tab:



6.1.9.1. Model Path textbox: The path of the model to export as Tube Laser STEP file for this project. It will show the path to the selected view model by default.

6.1.9.2. Browse button: A button to browse to the model to export, only supports Part or Assembly file.

6.1.9.3. Configuration dropdown: Choose the configuration to export. It will select the first configuration which contains "LASER" in the name (case insensitive).

6.1.10. DXF Tab:

Main	BOM	eDrawing	Tube Laser	DXF	PDF
Sheet Name	Include				
DXF SM 350 5 P	<input checked="" type="checkbox"/>				
DXF SM 350 5	<input checked="" type="checkbox"/>				
DXF SM 350 10 P	<input checked="" type="checkbox"/>				
DXF SM 350 10	<input checked="" type="checkbox"/>				
DXF SM 350 3	<input checked="" type="checkbox"/>				
DXF SM 350 3 P	<input checked="" type="checkbox"/>				
DXF SM 350 20	<input checked="" type="checkbox"/>				

6.1.10.1. Sheet Name column: Column listing all sheets with DXF in the name.

6.1.10.2. Include column: Column for you to choose which DXF you want to export.

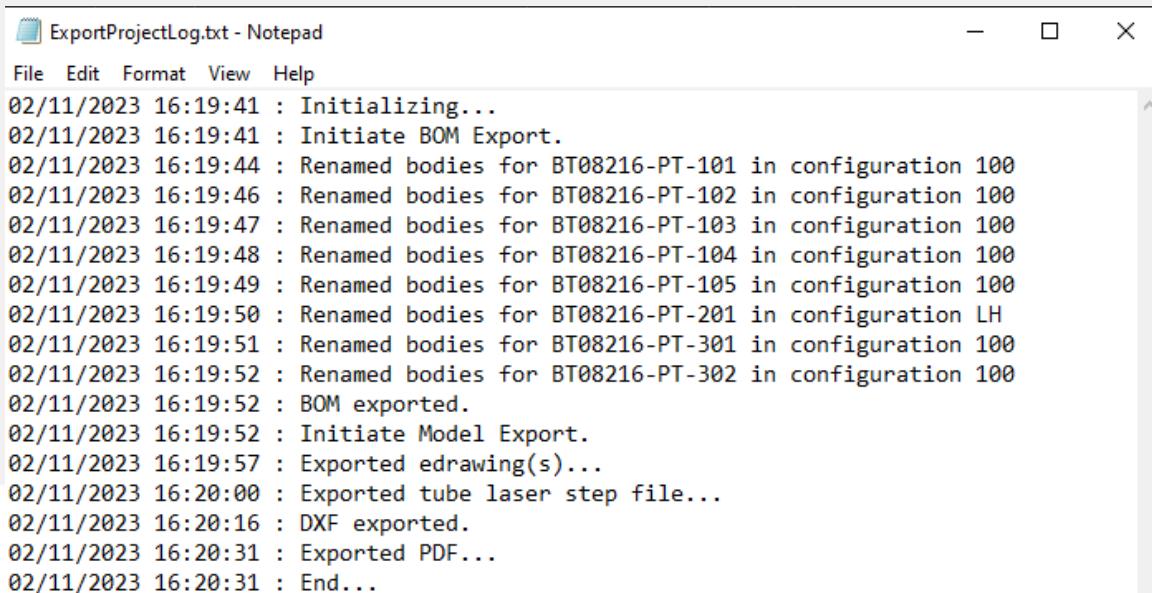
6.1.11. PDF Tab:

Main	BOM	eDrawing	Tube Laser	DXF	PDF
Sheet Name	Include				
Sheet1	<input checked="" type="checkbox"/>				
Sheet2	<input checked="" type="checkbox"/>				
Sheet3	<input checked="" type="checkbox"/>				
Sheet4	<input checked="" type="checkbox"/>				
Sheet5	<input checked="" type="checkbox"/>				
Sheet6	<input checked="" type="checkbox"/>				
Sheet7	<input checked="" type="checkbox"/>				
Sheet8	<input checked="" type="checkbox"/>				
Sheet9	<input checked="" type="checkbox"/>				
Sheet10	<input checked="" type="checkbox"/>				
Sheet11	<input checked="" type="checkbox"/>				
Sheet12	<input checked="" type="checkbox"/>				
Sheet13	<input checked="" type="checkbox"/>				
Sheet14	<input checked="" type="checkbox"/>				
Sheet15	<input checked="" type="checkbox"/>				
Sheet16	<input checked="" type="checkbox"/>				
Sheet17	<input checked="" type="checkbox"/>				
Sheet18	<input checked="" type="checkbox"/>				
DXF SM 350 5 P	<input type="checkbox"/>				
DXF SM 350 5	<input type="checkbox"/>				
DXF SM 350 10 P	<input type="checkbox"/>				
DXF SM 350 10	<input type="checkbox"/>				
DXF SM 350 3	<input type="checkbox"/>				
DXF SM 350 3 P	<input type="checkbox"/>				
DXF SM 350 20	<input type="checkbox"/>				

6.1.11.1. Sheet Name column: Column listing all sheets in the drawing.

6.1.11.2. Include column: Checkbox column for you to include or exclude the sheet for PDF export. Any sheet that has "DXF" in the sheet name is unchecked by default.

6.1.12. Make sure all settings are set accordingly, then press the Confirm button.



The screenshot shows a Windows Notepad window with the title "ExportProjectLog.txt - Notepad". The window contains a log of operations performed on 02/11/2023 at 16:19:41. The log includes:

- Initializing...
- Initiate BOM Export.
- Renamed bodies for BT08216-PT-101 in configuration 100
- Renamed bodies for BT08216-PT-102 in configuration 100
- Renamed bodies for BT08216-PT-103 in configuration 100
- Renamed bodies for BT08216-PT-104 in configuration 100
- Renamed bodies for BT08216-PT-105 in configuration 100
- Renamed bodies for BT08216-PT-201 in configuration LH
- Renamed bodies for BT08216-PT-301 in configuration 100
- Renamed bodies for BT08216-PT-302 in configuration 100
- BOM exported.
- Initiate Model Export.
- Exported edrawing(s)...
- Exported tube laser step file...
- DXF exported.
- Exported PDF...
- End...

6.1.13. A log file will be generated at {AppData}\BTG\BTG SolidWorks AddIn\Log location.

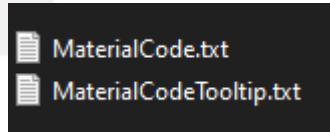
6.2. Prep DXF Export



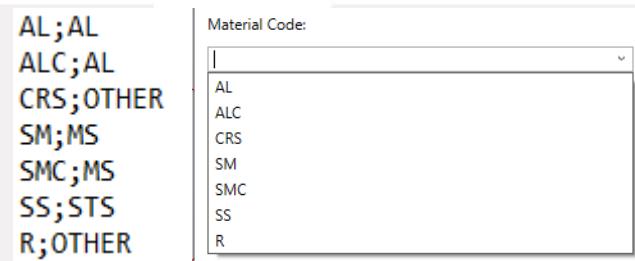
- 6.2.1. A Macro to prepare the drawing sheet for production. It only supports drawing files.
- 6.2.2. Before running the macro, check setting and make sure the path is referring to the correct folder.



- 6.2.3. In the setting folder, it should have 2 files.

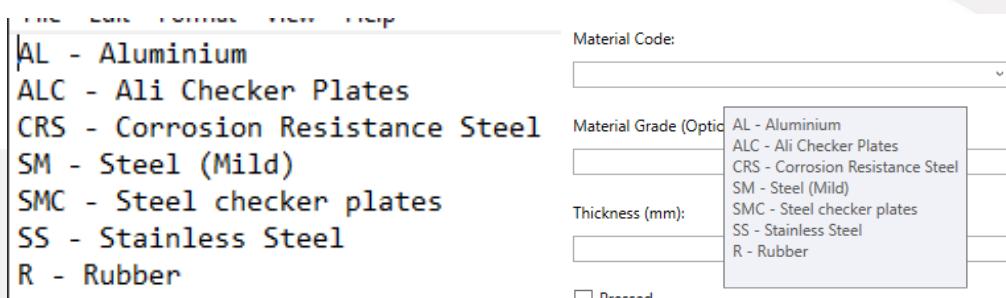


- 6.2.4. MaterialCode.txt is for material code drop down aliases.



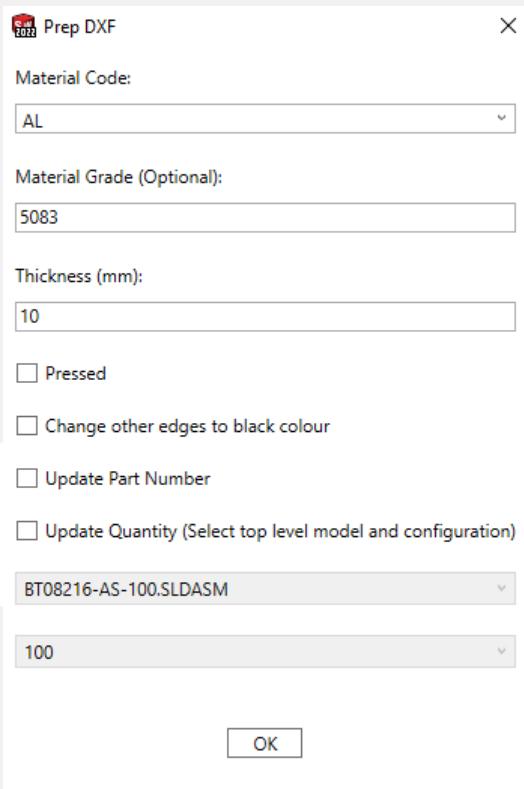
- 6.2.5. The character before ";" will appear at Material Code dropdown selection while the character after ";" will be used in the line "MAT" for description string.

- 6.2.6. MaterialCodeTooltip.txt is for the tooltip that appears when you hover over the material code drop down.



- 6.2.7. Make the DXF sheet active and press the Prep DXF Export button.

6.2.8. Another window will be prompted to enter plate thickness and select material code. You can type in customized material code as well.



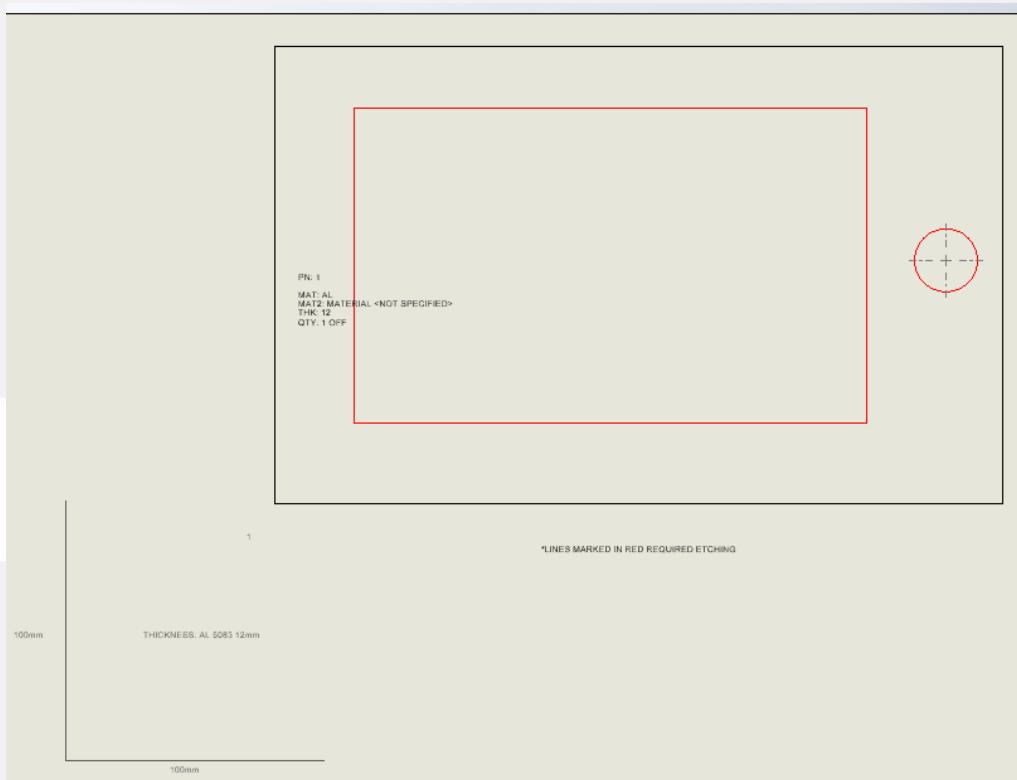
6.2.9. Check on Update Part Number for any view that is shared by different part number. This option will loop through each table and gather all part number that has same body as the body in the view.



- 6.2.10. Check on Update Quantity to calculate the total quantity of the body in the model selected in the dropdown below of the checkbox. It uses SolidWorks API GetCoincidenceTransform to check if any of the other bodies in the assembly returns a transformation matrix. There is another dropdown to select the reference configuration as well, make sure to select the right one.
- 6.2.11. After clicking ok, the sheet format will be removed, scale will change to 1:1. All visible edges in all drawing view will be coloured **BLACK** if the checkbox Change other edges to black colour is checked. If your model does not prepare with the Etch Feature macro, any visible edges with 0.5mm edge associated with it will be coloured RED. Additionally, few notes will be added to the view accordingly:
- 6.2.12. An etch note will be added at the bottom of the view if it is coloured.
- 6.2.13. Description notes.
- 6.2.14. A balloon without border and leader for cross-checking function in Export DXF Macro.
- 6.2.15. The format for the description notes is as follow:
- 6.2.15.1. Row 1: PN: <Balloon Number>
 - 6.2.15.2. Row 2: Cut List Property Description
 - 6.2.15.3. Row 3: MAT <Material Alias>
 - 6.2.15.4. Row 4: MAT2: <Cut List Property Material>
 - 6.2.15.5. Row 5: THK: <Thickness Input>
 - 6.2.15.6. Row 6: QTY: <Balloon Quantity> OFF

PN: 1
SHEET
MAT: AL
MAT2: 5083-H116
THK: 10
QTY: 1 OFF

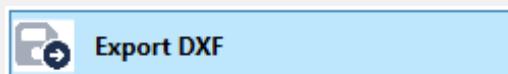
6.2.17. If your model is prepared using the Etch Feature Macro, any edge that contains "etch" in the model entity name will be coloured red. Below is the result from the Macro.



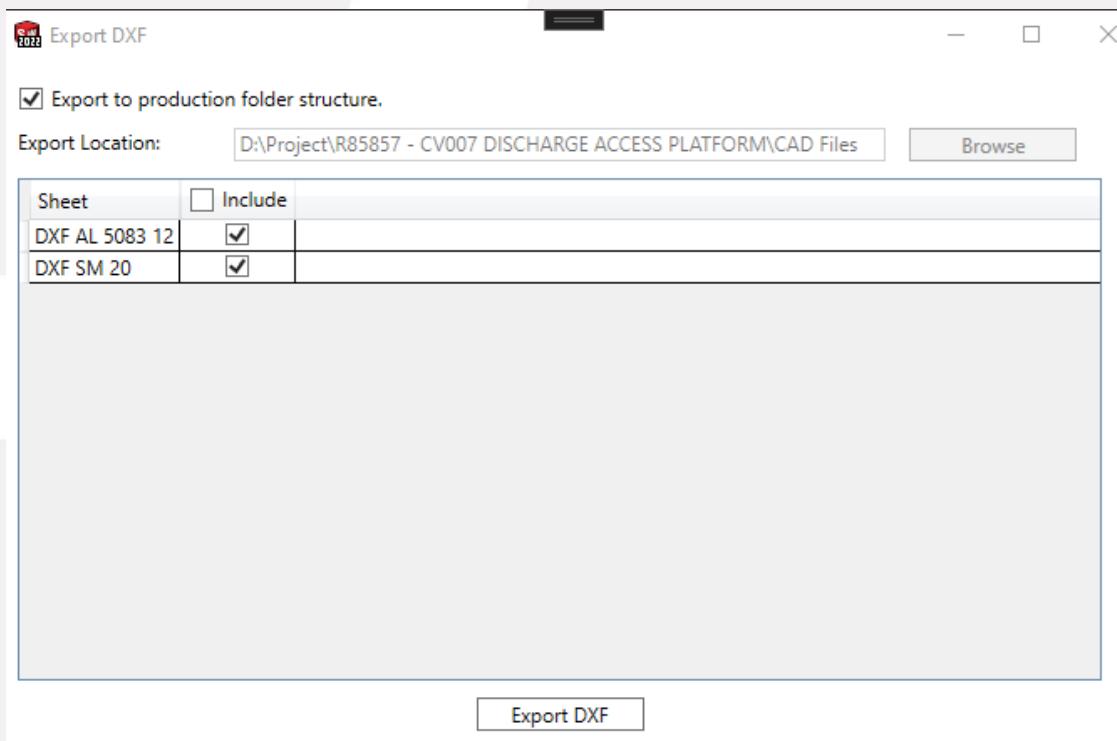
6.2.18. If the part number is different from your cut list or BOM, override the balloon instead of the text generated by this Macro.

6.2.19. If your balloon shows an asterisk mark (*), the macro will try to correct it for you. If the macro failed, you should override the quantity manually on the text generated.

6.3. Export DXF



- 6.3.1. A Macro to save every sheet that contains "dxf" in the sheet name (case insensitive) in Production\Flat Plates or Pressed Plates folder accordingly. It only supports drawing files. If you are working directly in PDM, make sure the output folder already exists. The name of the exported file will be in the format of FileName_Revision_SheetName.
- 6.3.2. Open the drawing you which to save all the sheet with "dxf" in the sheet name and press the Export DXF macro.
- 6.3.3. An interface will be prompted with a list of supported sheets. The following are the Macro for each control in the interface:
- 6.3.3.1. Export to production folder structure check box: Check to save at ...\\Production folder. You can set the default setting for this option in setting. Uncheck to export the file to custom Export Location, default will be the same folder as the drawing file.
- 6.3.3.2. Export Location textbox: Is disabled if Export to production folder structure is checked. Browse or key in the custom export location.
- 6.3.3.3. Include column: Uncheck to exclude that sheet from exporting. Check or uncheck the checkbox in the header will update all rows.



- 6.3.4. Press Export DXF to execute the macro. The Macro will run through all the sheets and save as DXF accordingly.



- 6.3.5. All the generated DXF without ending with "p" in the sheet name will be saved in the Production\Flat Plates directory.
- 6.3.6. If the sheet name file has "p" at the end, which indicates that it is pressed or flattened, the DXF will be saved at Production\Pressed Plates - external fab.



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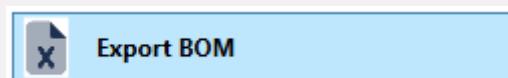
sales@bendtechgroup.com.au

www.bendtechgroup.com.au

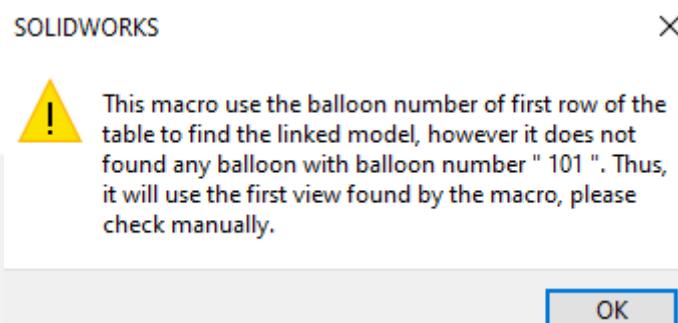
2-10 Kewdale Road, Welshpool WA 6106



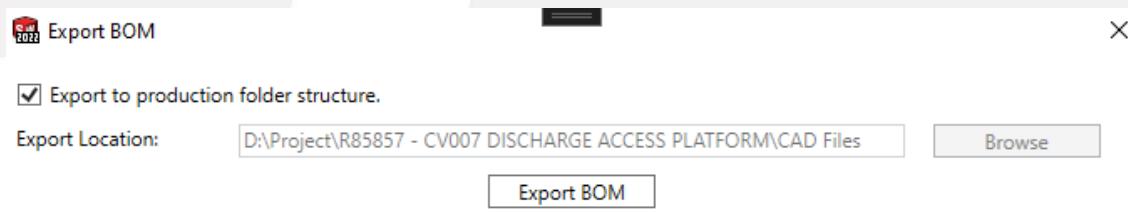
6.4. Export BOM



- 6.4.1. A Macro to save every BOM and Weldment Cut List to an excel file in Production folder. It only supports drawing files. If you are working directly in PDM, make sure the output folder already exists.
- 6.4.2. Open the drawing you which to save all the BOM to excel and press the Export BOM button.
- 6.4.3. If the macro is unable to find your view attached to the Weldment Cut List, it will add temporary balloon to each view to search for the view. If it still failed, it would prompt the warning to inform that it will take the first view in the sheet as the referenced view.



- 6.4.4. A window will be prompted, checked on Export to production folder structure will save the output file to ..\Production folder. You can set the default setting for this option in setting. Unchecked will activate custom Export Location textbox. You can either browse or key in the custom export location.

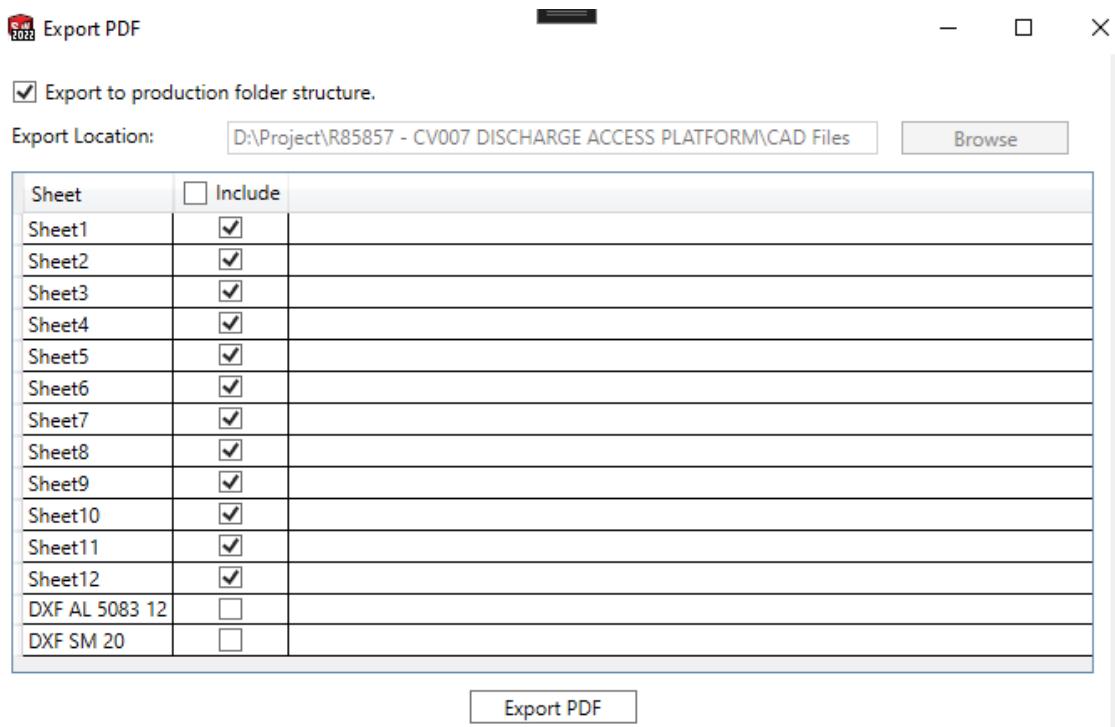


- 6.4.5. Click Export BOM to execute the macro, it will run through all BOM and Weldment Cut List and save in Excel accordingly.
- 6.4.6. Any existing excel in the export location with same name will be replaced.

6.5. Export PDF



- 6.5.1. A Macro to export drawing files as PDF to Production folder while excluding those prepared for DXF export. It only supports drawing files. If you are working directly in PDM, make sure the output folder already exists.
- 6.5.2. Open the drawing you wish to export as PDF and press the Export PDF macro.
- 6.5.3. A window will be prompted, checked on Export to production folder structure will save the output file to ..\Production folder. You can set the default setting for this option in setting. Unchecked will activate custom Export Location textbox. You can either browse or key in the custom export location. Choose which sheet to export by check uncheck the checkbox. Check or uncheck the checkbox in the header will update all rows. By default, all sheets with name contains “DXF” are excluded.



- 6.5.4. Click Export PDF to execute the macro.

6.6. Export Model

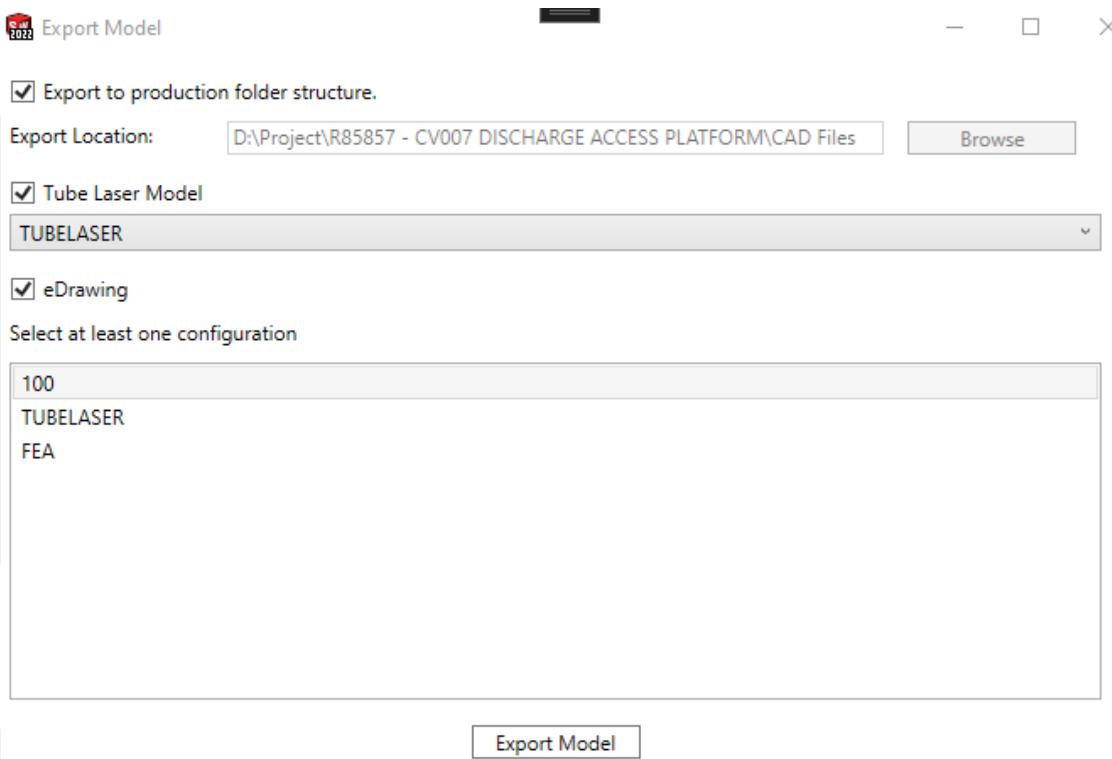


6.6.1. Macro to export SolidWorks part or assembly file as eDrawing and step file. If you are working directly in PDM, make sure the output folder already exists.

This Macro will fail if your model does not follow the format of BTXXXXX-XX-XXX.

6.6.2. Open the assembly or part file that you want to export as eDrawing or step file. Then press the Export Model macro.

6.6.3. A window will be prompted, checked on Export to production folder structure will save the output file to ..\Production folder. You can set the default setting for this option in setting. Unchecked will activate custom Export Location textbox. You can either browse or key in the custom export location.



6.6.4. If you wish to export as STEP file for tube laser, check on the Tube Laser Model checkbox. Then, select the configuration that you want to use as tube laser model. Only single selection is permitted.

6.6.5. If you wish to export as an eDrawing file, check on the eDrawing check box. Then, select whichever configuration that you want to include in this Edrawing file. Multiple selection is possible.

6.6.6. STEP naming format – BTXXXXX-AS100A.STEP / BTXXXXX-PT100A.STEP

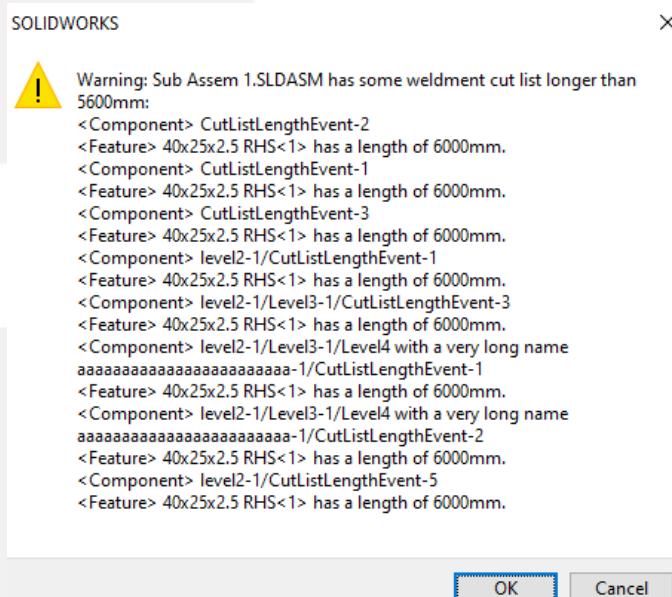
6.6.7. eDrawing naming format – BTXXXXX-AS-100_RevA.easm / BTXXXXX-PT-100_RevA.eprt

7. Events

7.1. Event to check for weldment cut list length

7.1.1. When this add-in is enabled, every change in document will attach an event to the document. This event only supports Part and Assembly.

7.1.2. Whenever the document with this event attached is saved, it will go through all the weldment properties (will traverse child component for assembly) and check if the length property is equal or longer than 5700mm. If it does, a warning message will be displayed.



7.1.3. You can turn off this event listener by going to setting and uncheck the Check Weldment Cut List Length checkbox. This setting will checked by default when start up SolidWorks.

Check Weldment Cut List Length

8. Design Automation

8.1. Platform Automation



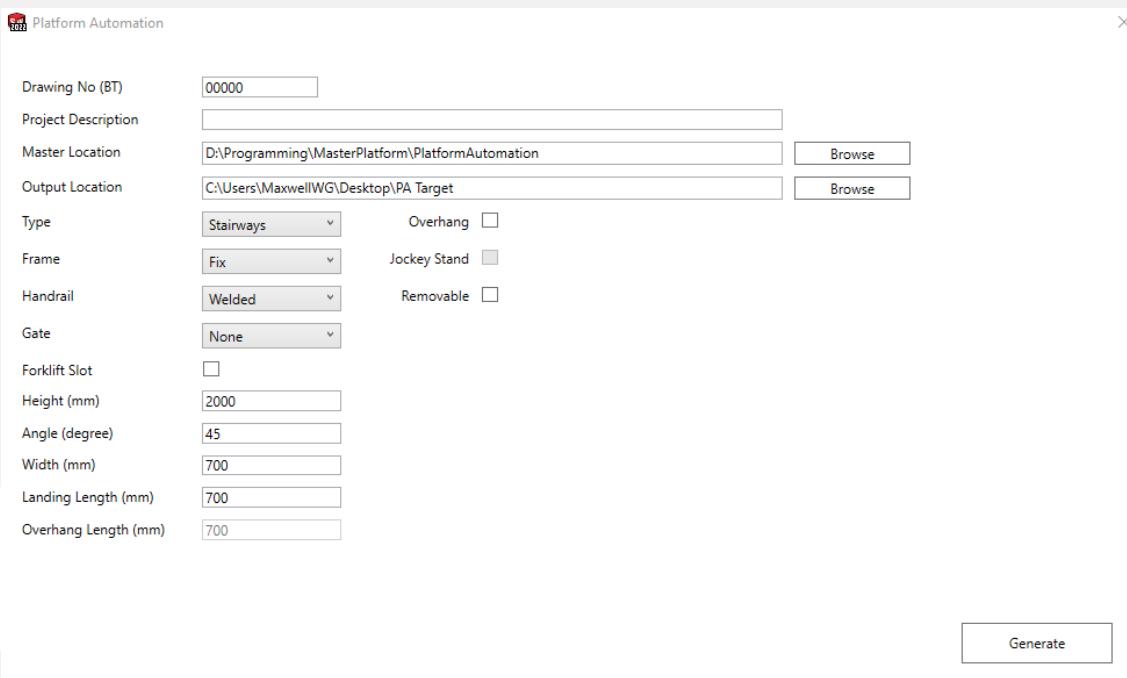
8.1.1. An experimental Macro to automatically generate Platform CAD models and drawings. This Macro generates models according to user input by using the premade model specifically for this Macro. Now only support standard platform as shown in picture below:



8.1.2. Close all open documents before running this Macro.

8.1.3. Click on Platform Automation button.

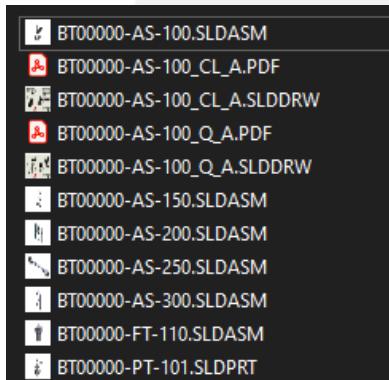
8.1.5. Interface will be shown for user to input the parameters of the platform.



8.1.6. Drawing No (BT): All generated models and drawings will be using this input as drawing number.

8.1.6.1. Sample Input: 00000

8.1.6.2. Sample assembly file name generated: BT00000-AS-100.SLDASM



8.1.7. Drawing number must fulfilled the following criteria:

8.1.7.1. Character length must be exactly 5.

8.1.7.2. Can contain character from 0 to 9 only.

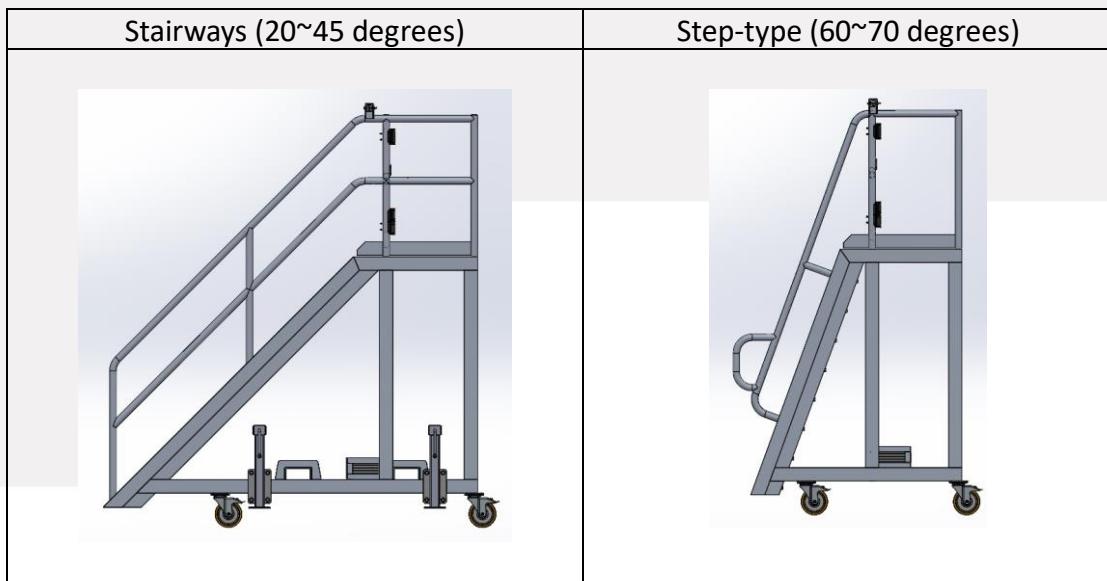
8.1.9. Project Description: The description field in the main assembly custom property. This input cannot be empty.

Properties			
	Property Name	Type	Value /
1	Weight	Text	"SW-Mass@BT00000-AS-100.SLDASM"
2	Description	Text	test
3	<Type a new property>		

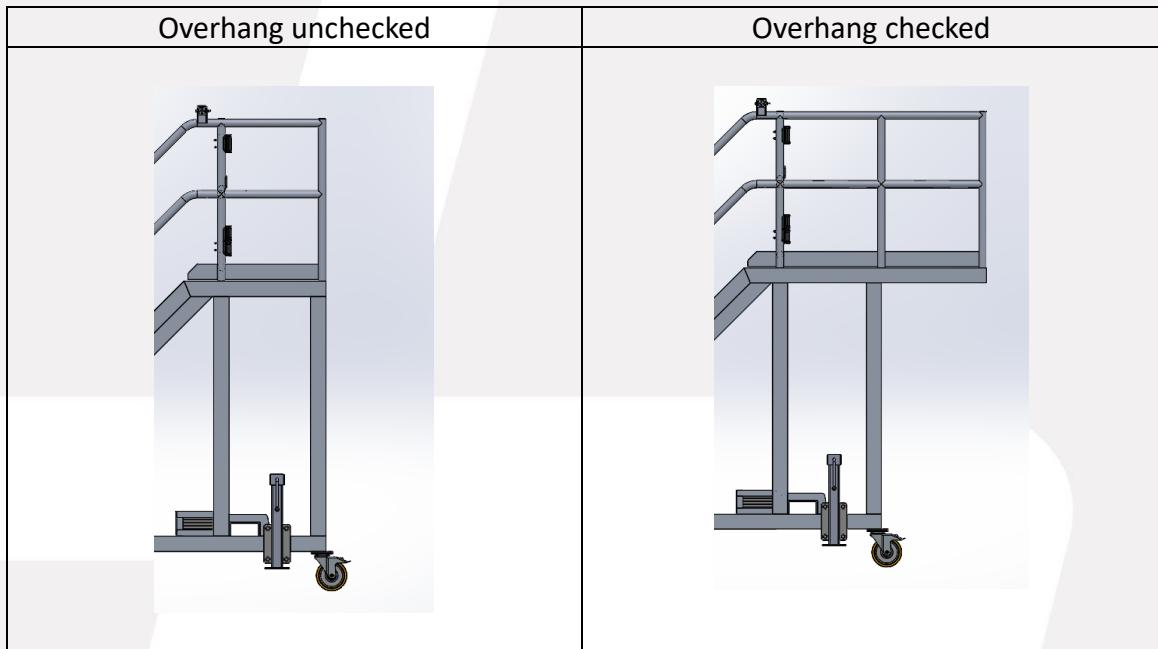
8.1.10. Master Location: Folder path to the specific models built for this Macro. If you have saved the models in a different location, make sure to set it first to avoid any unexpected error.

8.1.11. Output Location: Folder path to store all the generated models and drawings.

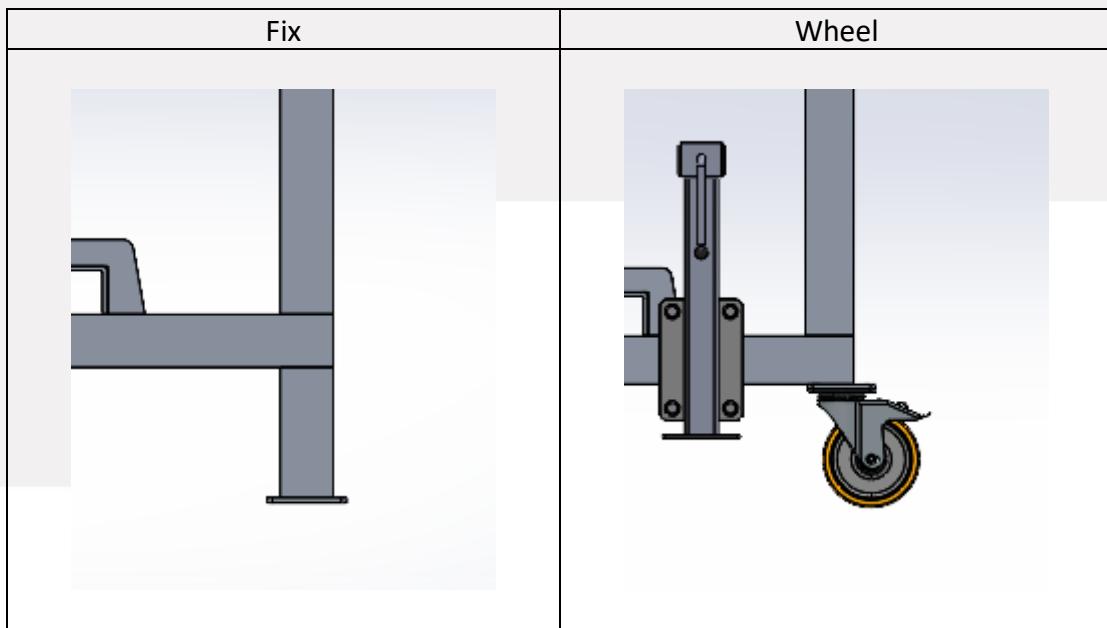
8.1.13. Type: There are 2 types of platforms, Stairways and Step-type.



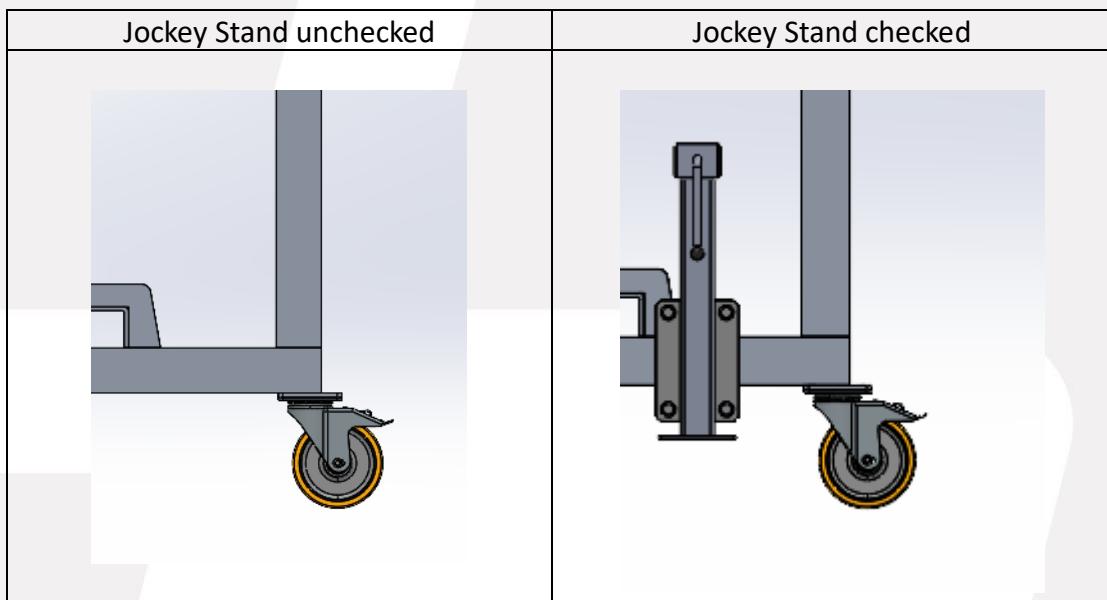
8.1.14. Overhang checkbox: Check this for platform that has overhang.



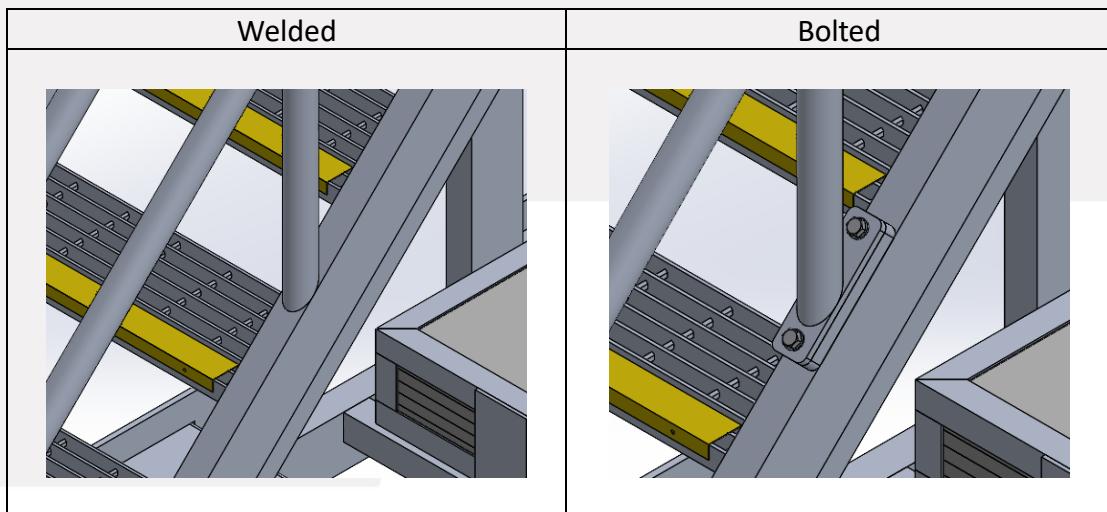
8.1.16. Frame: There are 2 types of frames, Fix and Wheel.



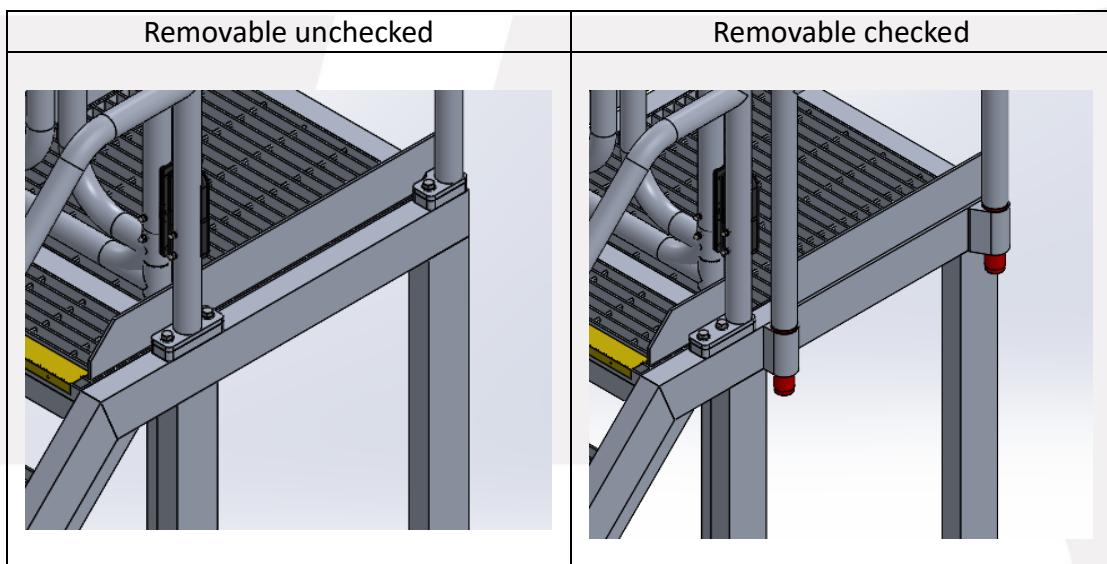
8.1.17. Jockey Stand checkbox: Check this to include the jockey stand. **Step-type platform and Fix frame do not support Jockey Stand.**



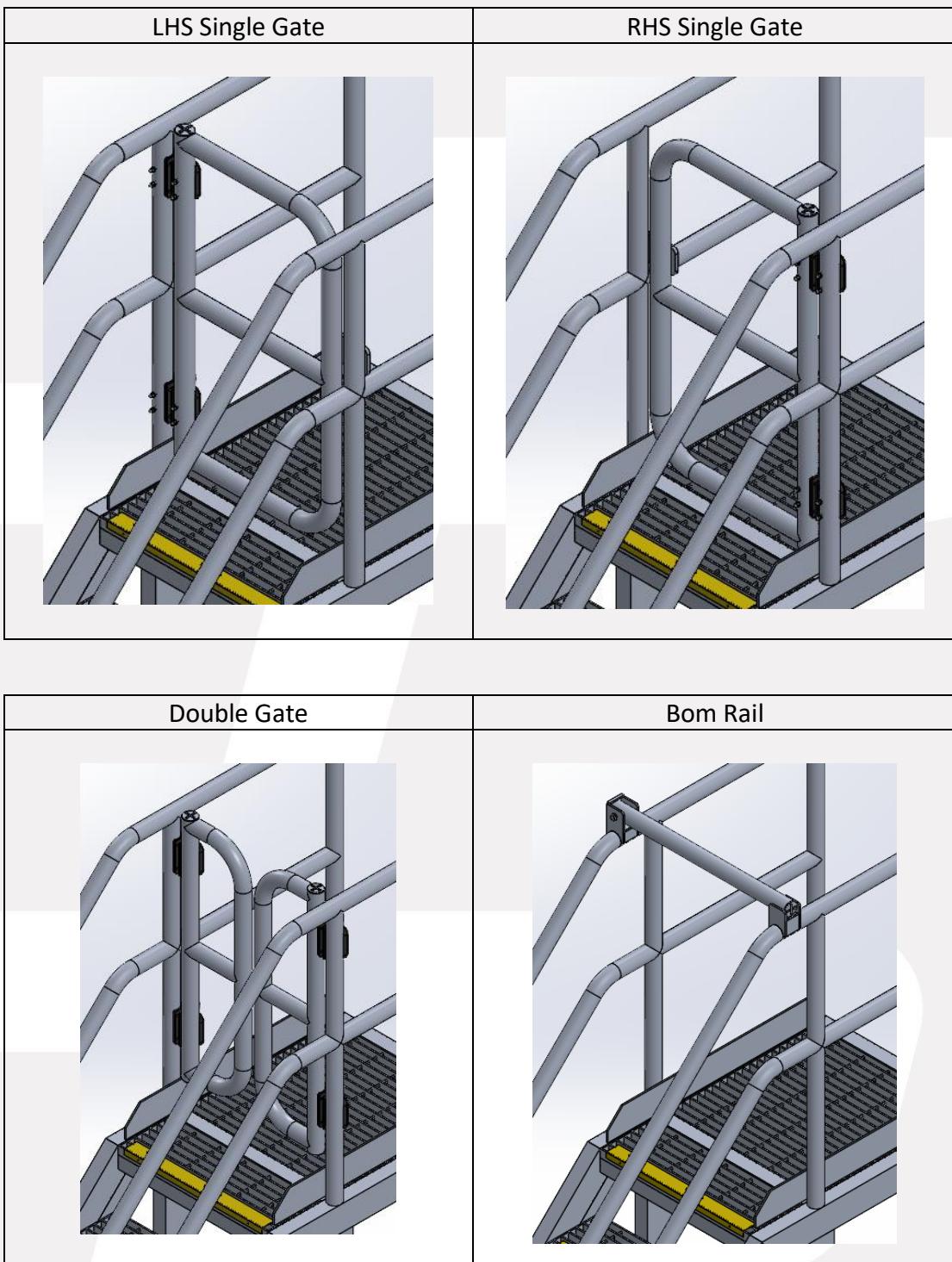
8.1.19. Handrail: There are 2 types of handrails, Welded and Bolted.



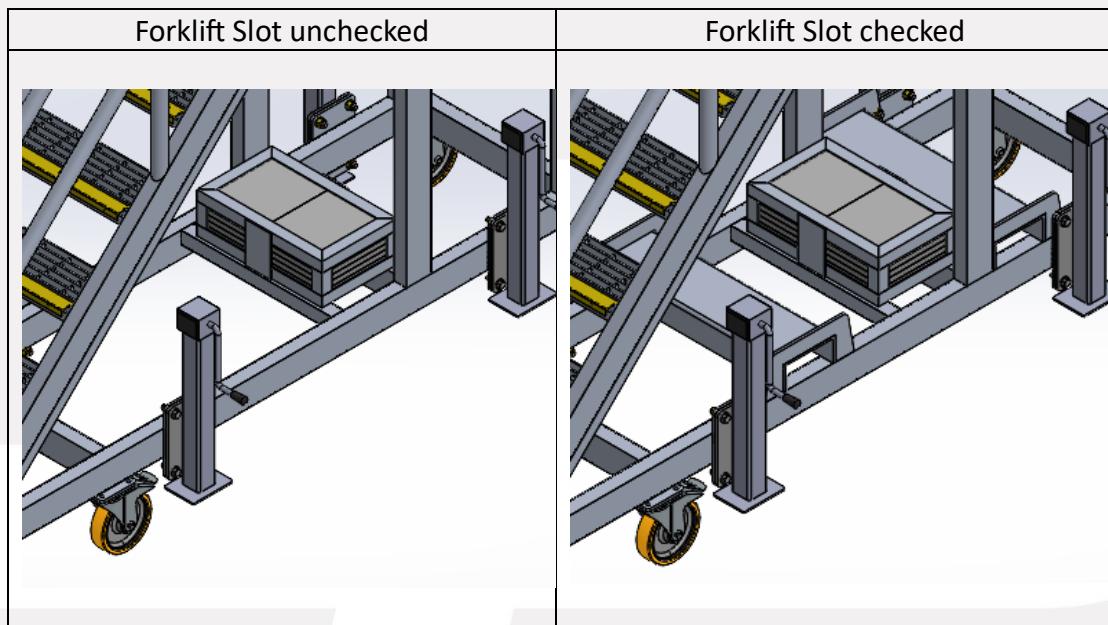
8.1.20. Removable: Check this for removable guardrail.



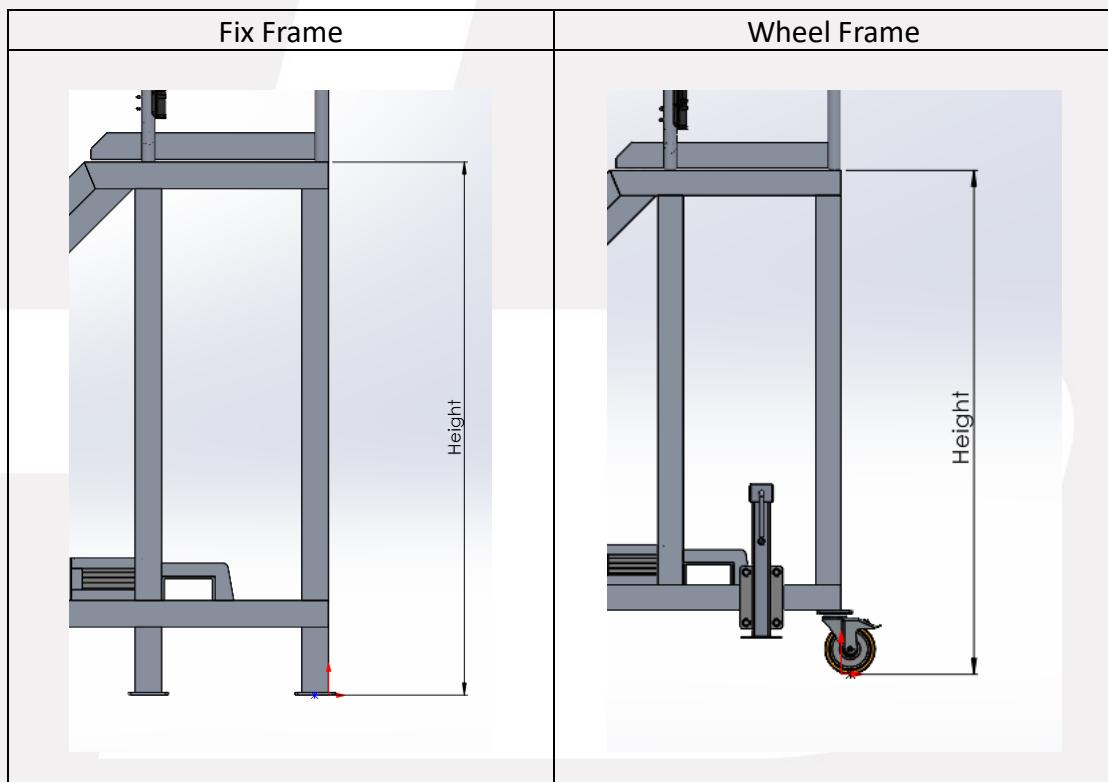
8.1.22. Gate: There are 5 options for gate, None, LHS Single Gate, RHS Single Gate, Double Gate and Bom Rail.



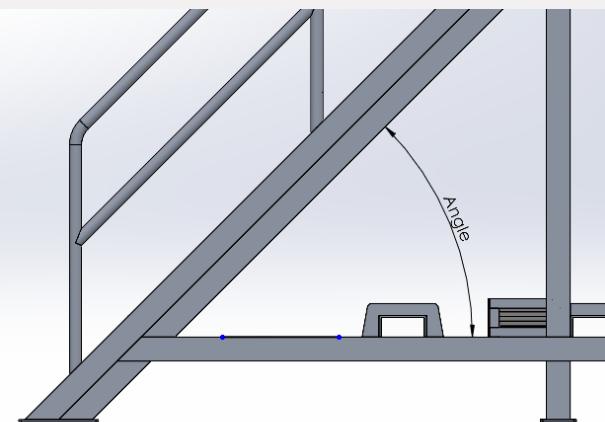
8.1.24. Forklift Slot checkbox: Check this for forklift slot. **Step-type platform does not support forklift slot.**



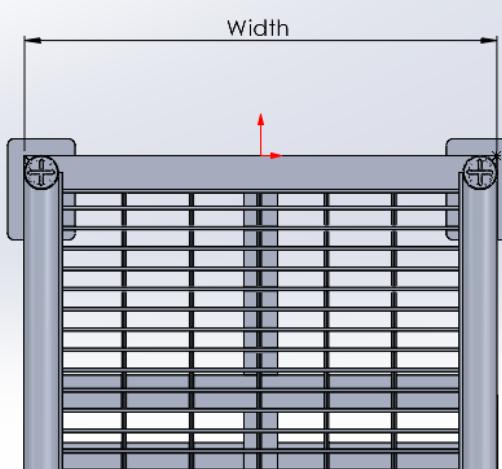
8.1.25. Height (mm): Height input for the platform in unit mm.



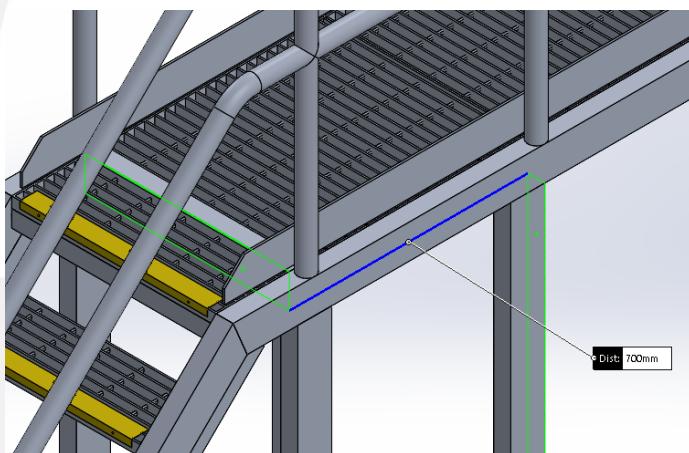
8.1.27. Angle (degree): Angle input for the platform in unit degree. Stairways platform only supports 20 to 45 degrees while Step-type platform only supports 60 to 70 degrees.



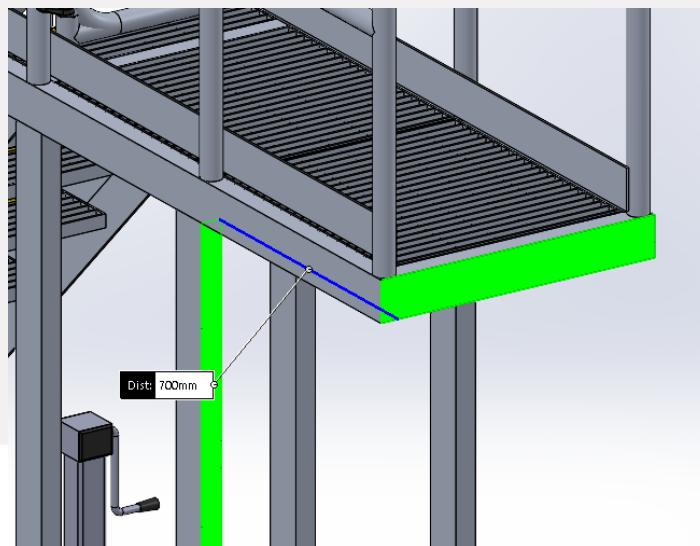
8.1.28. Width (mm): Stair width in unit mm.



8.1.29. Landing Length (mm): Landing length in unit mm. The distance measured from the back of top stair tread to the end of the frame.

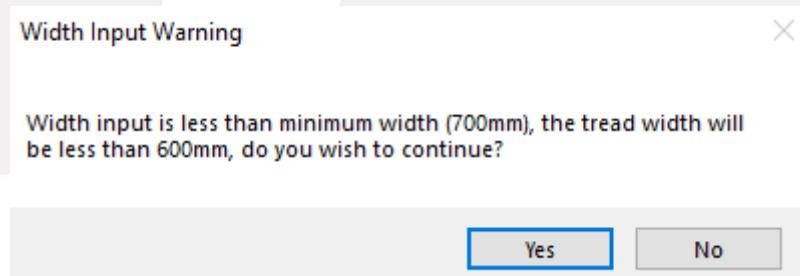


8.1.30. Overhang Length (mm): Overhand length in unit mm. The distance measured from the end of the frame to the end of top landing.



8.1.31. Once the user inputs all the parameters, click on Generate button. The Macro will validate the inputs first and display any error message if any input does not meet the criteria.

8.1.32. The width of stair tread for stairways platform should be more than 600mm while for step-type ladders should be within the range of 450mm ~ 750mm. Warning will be prompted if input does not meet the criteria. However, the user can choose to ignore the warning by selecting Yes.



8.1.33. For stairways platform, tread calculation will be carried out to calculate the tread risers (R) and goings (G) that fulfill the criteria below:

- 8.1.33.1. $130mm \leq R \leq 225mm$
- 8.1.33.2. $215mm \leq G \leq 335mm$
- 8.1.33.3. $540mm \leq (2 \cdot R + G) \leq 700mm$
- 8.1.33.4. $2 \leq \text{Number of tread} \leq 18$

8.1.34. For step-type platform, tread calculation will be carried out to calculate the tread risers (R) that fulfill the criteria below:

- 8.1.34.1. $200mm \leq R \leq 300mm$

8.1.36. The Models and Drawings will be generated and saved in the output location.

NO.	QTY	DESCRIPTION	PART NUMBER	MATERIAL
1	1	1100 X 1000 X 1000 MM	BT00000-PT-101	ALUMINIUM
2	1	A325 A3M 400 X 400 DRAIN	BT00000-PT-102	STEEL-TS
3	9	TS1A325A3M 215 X 600 SPAN	BT00000-PT-103	6053-TS
4	1	COUNTERWEIGHT	BT00000-PT-104	STEEL-ALUMINIUM
5	9	NAB EXCEEDED ANTI SLIP STAIR	6002113	ALUMINIUM

NO.	QTY	DESCRIPTION	LENGTH	MATERIAL
101	1	100X50X3 RHD	600	6050-TS
102	2	100X50X3 RHD	600	6050-TS
103	1	100X50X3 RHD	600	6050-TS
104	1	100X50X3 RHD	700	6050-TS
105	1	100X50X3 RHD	800	6050-TS
106	1	100X50X3 RHD	815	6050-TS
107	1	100X50X3 RHD	830	6050-TS
108	2	100X50X3 RHD	1445	6050-TS
109	2	100X50X3 RHD	1598	6050-TS
110	2	100X50X3 RHD	1600	6050-TS
111	1	100X50X3 RHD	2118	6050-TS
112	1	100X50X3 RHD	8143	6050-TS
113	2	50 X 50 HWD	25	6050-TS
114	2	50 X 50 HWD	605	6050-TS
115	2	50 X 50 HWD	606	6050-TS
116	2	50 X 50 HWD	607	6050-TS
117	2	TUBE 250 X 3	650	6050-TS
118	3	TUBE 250 X 3	1011	6050-TS
119	2	TUBE 250 X 3	1020	6050-TS
120	2	TUBE 250 X 3	1045	6050-TS
121	2	TUBE 250 X 3	1314	6050-TS
122	2	TUBE 250 X 3	4220	6050-TS
123	2	TUBE 250 X 3	4413	6050-TS
124	2	FB 100 X 6	448	6050-TS
125	2	FB 100 X 6	450	6050-TS
126	2	FB 100 X 6	313	6050-TS
127	4	PLATE 10 X 40 X 100	6050-TS	
128	2	ACORN NUTS & SPRINGS	6050-TS	
129	1	ACORN NUT	ALUMINIUM	
130	1	ACORN NUT	ALUMINIUM	

NO.	QTY	DESCRIPTION	APPROVED	DATE
REV.	DESCRIPTION	APPROVED	DATE	
A	INITIAL CONCEPT DESIGN			

□ Approved	□ Approved as corrected	□ Revise and resubmit
By: _____ Date: _____		

REVISIONS	APPROVED WEIGHT (KG)	DRAWN BY	DATE
REV. A	INITIAL CONCEPT DESIGN	_____	_____

WLL - 150 KG

REVISIONS	APPROVED WEIGHT (KG)	DRAWN BY	DATE
REV. A	INITIAL CONCEPT DESIGN	_____	_____

REVISIONS	APPROVED WEIGHT (KG)	DRAWN BY	DATE
REV. A	INITIAL CONCEPT DESIGN	_____	_____

REVISIONS	APPROVED WEIGHT (KG)	DRAWN BY	DATE
REV. A	INITIAL CONCEPT DESIGN	_____	_____

PERTH
 (08) 9459 8600

ADELAIDE
 (08) 8231 1513

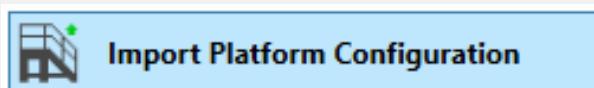
DARWIN
 (08) 8963 5621

sales@bendtechgroup.com.au

www.bendtechgroup.com.au

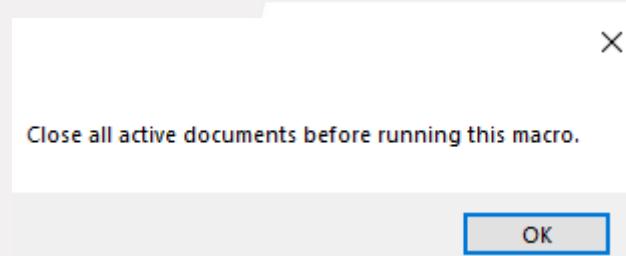
2-10 Kewdale Road, Welshpool WA 6106

8.2. Import PlatformConfiguration

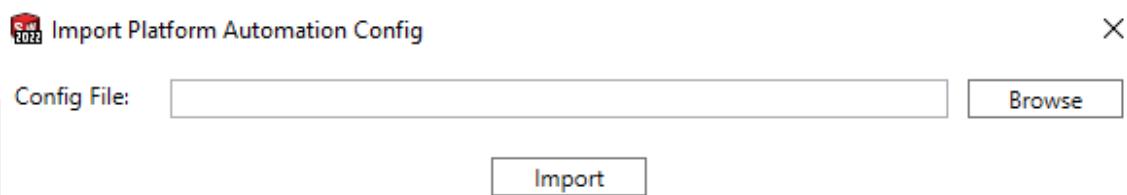


8.2.1. A macro that imports configuration file created by BTG Configurator website.

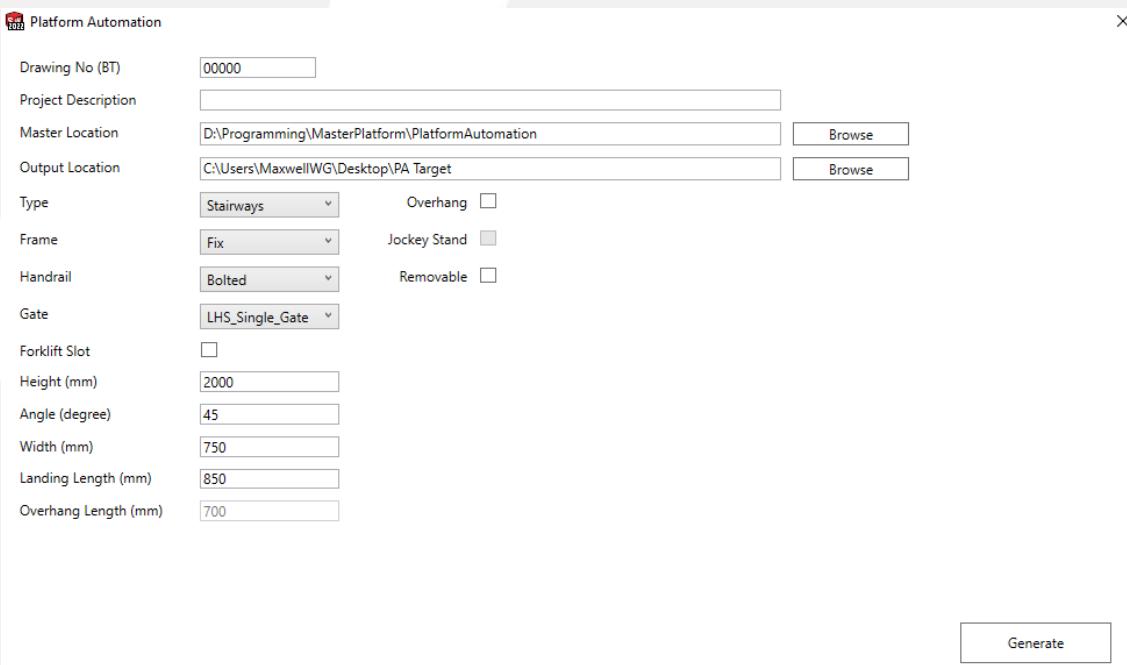
8.2.2. Make sure there is no document open when running this macro.



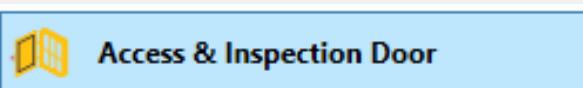
8.2.3. A dialog will appear to browse for configuration file.



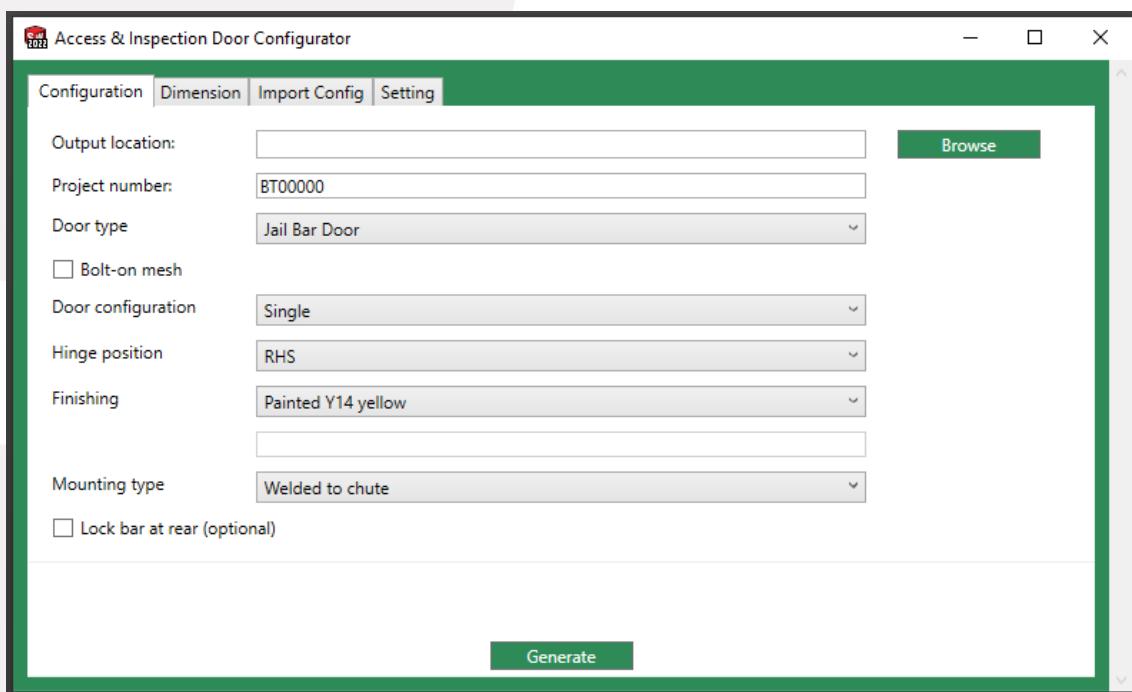
8.2.4. Once selected the configuration file, click Import. All the information from the configuration file will be imported into Platform Automation_macro.



8.3. Access & Inspection Door Configurator



8.3.1. This macro creates the standard Access & Inspection Door. There are 3 tabs in the interface of this macro when initiated.

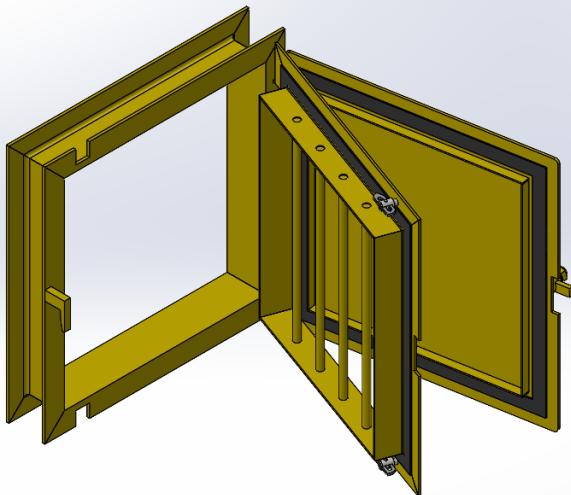
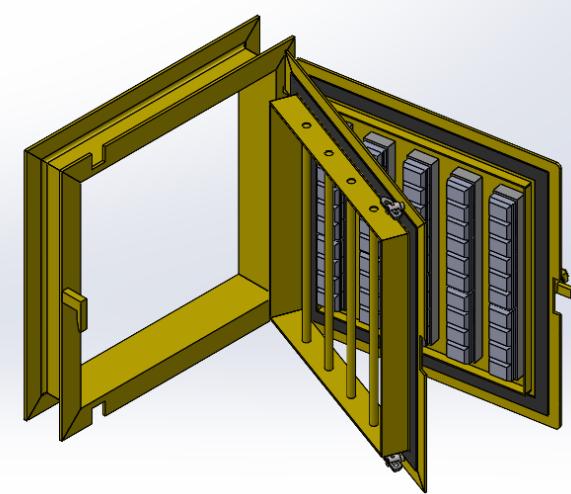
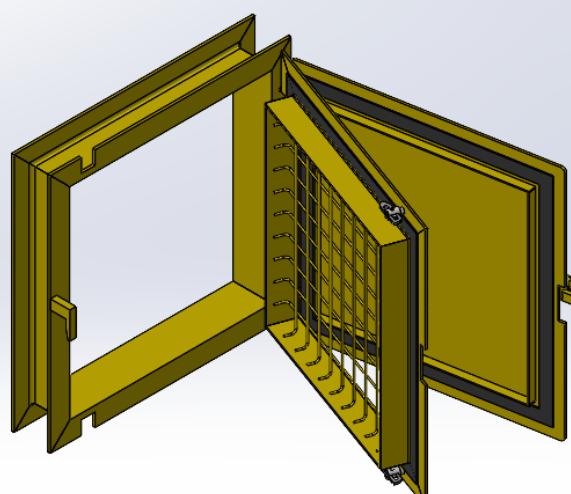


8.3.2. Tab 1 is for door configuration:

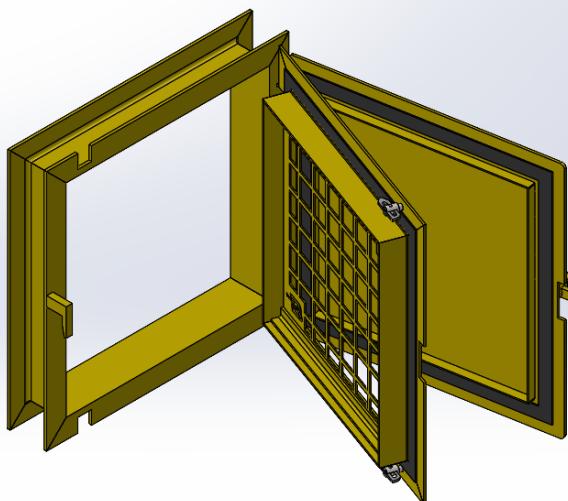
8.3.2.1. Output location: Define the output location for the door. Empty or invalid value is not accepted. You may use the browse button to choose the location.

8.3.2.2. Project number: The output file mostly will have the naming format of XXXXXXXX-AS-100.SLDASM, this setting will replace the XXXXXXXX. Empty value is not accepted

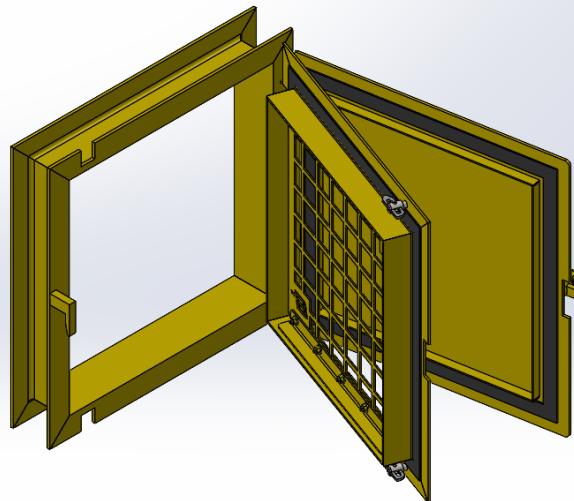
8.3.2.3. Door type:

Door Type	Reference Image
Jail Bar Door	
Armour Door	
Wire Guard Mesh Door	

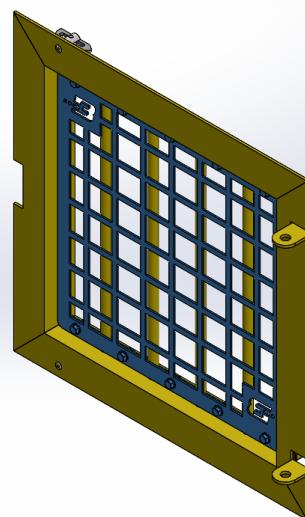
Welded Mesh Machine
Guarding Door



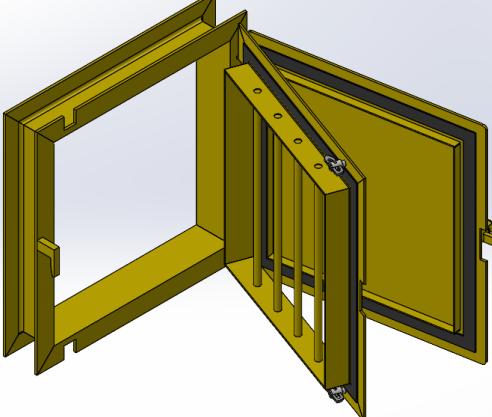
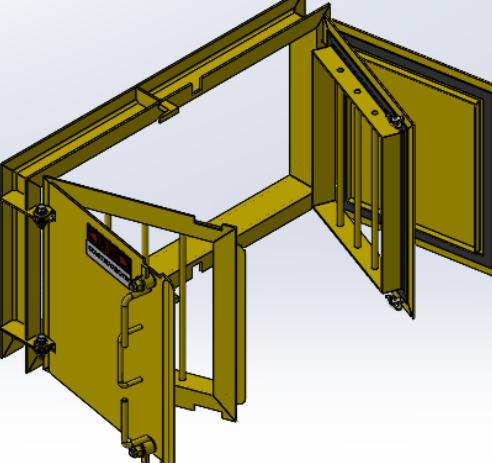
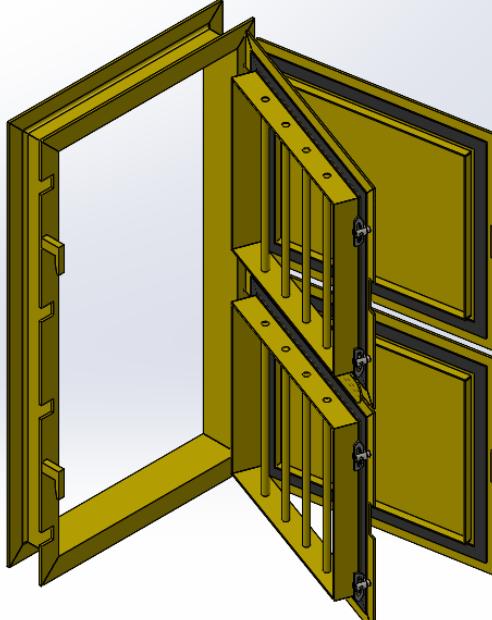
Bolted Mesh Machine
Guarding Door



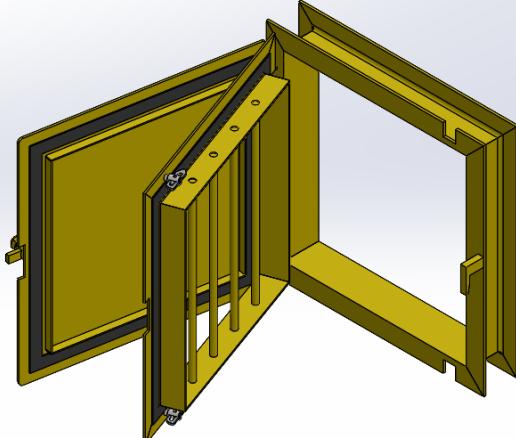
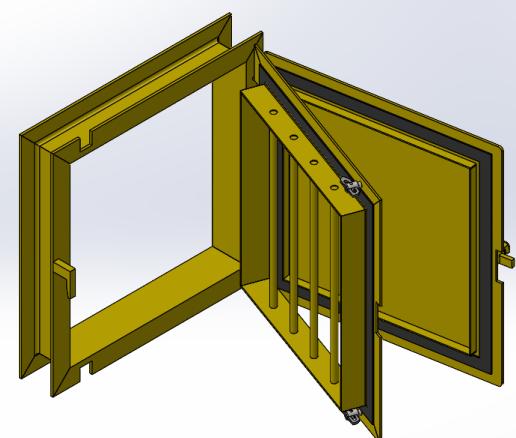
8.3.2.4. Bolt-on mesh: This option is only available for Jail Bar Door type. This option adds removable machine mesh on jail bar door.



8.3.2.5. Door Configuration:

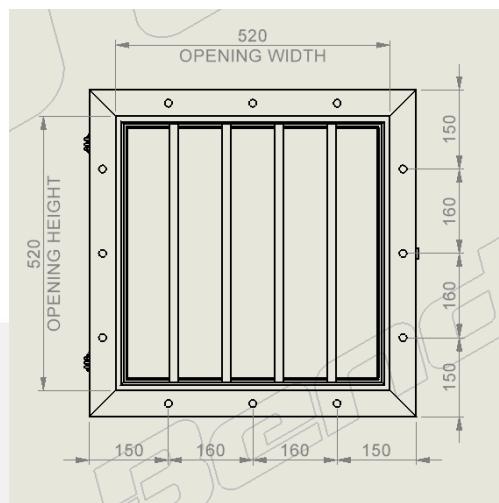
Door Configuration	Reference Image
Single	
Double Horizontal (Does not support Armour Door yet)	
Double Vertical (Does not support Armour Door yet)	

8.3.2.6. Hinge Position:

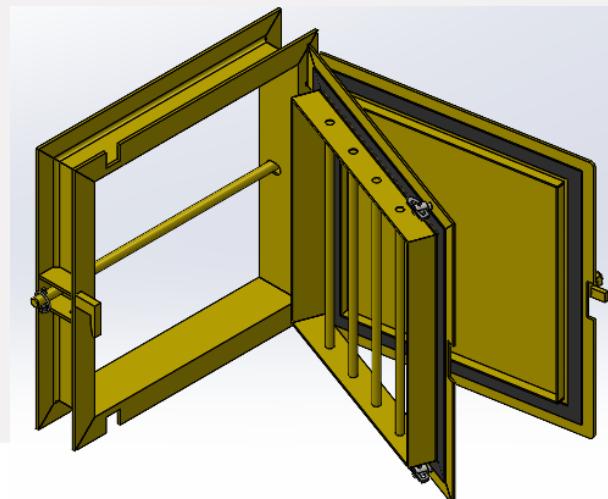
Hinge Position	Reference Image
LHS (Does not support Double Horizontal configuration)	
RHS (Does not support Double Horizontal configuration)	

8.3.2.7. Finishing: Painted Y14 yellow, Hot dip galvanized, Hot dip galvanized + painted Y14 yellow or Other. If choose Other, the field below Finishing field must be filled in.

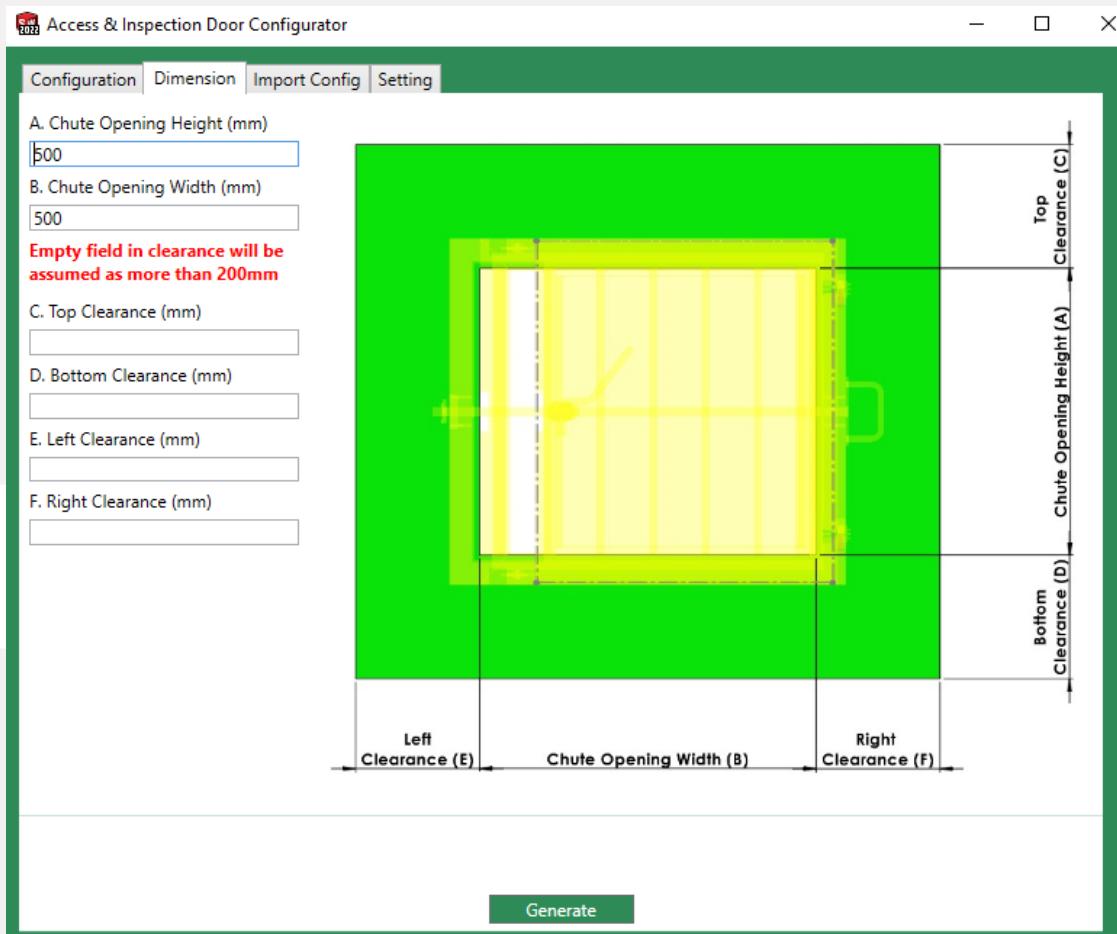
8.3.2.8. Mounting type: Welded to chute or Bolted to chute, if choose Bolted to chute, bolt holes will be added to the model.



8.3.2.9. Lock bar at rear (optional): Check on this option will add Safety rail to the model.



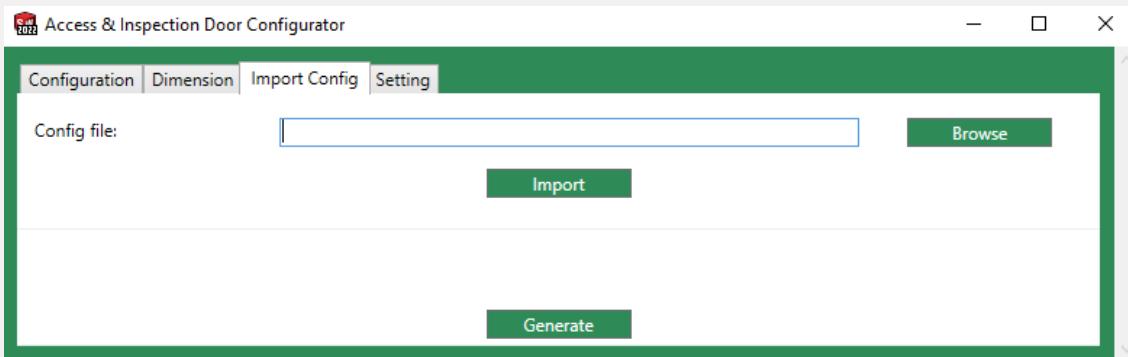
8.3.3. Tab 2 is for dimension:



8.3.3.1. The model will be generated with door opening = chute opening + 20mm.

8.3.3.2. If the clearance is empty, it will be assumed that the area to weld or bolt the door onto the chute is sufficient. Else, if the filled in clearance is less than or equal to 70mm, door opening = chute opening. The clearance cannot be less than 50mm.

8.3.4. Tab 3 is for Configuration Import.



8.3.4.1. The configuration can be imported from a text file with the following format:

```
aidoor.btgconfig - Notepad
File Edit Format View Help
JailBarDoor;Single;LHS;true;true;false;;false;true;Bolted;700;700;200;200;200;200
```

8.3.4.2. There should be 16 inputs separated by semi-colon. This file is usually generated from BTG Configurator Web, you may create the file manually if you understand the format.

8.3.4.3. There is a Browse button for users to browse for the config file path.

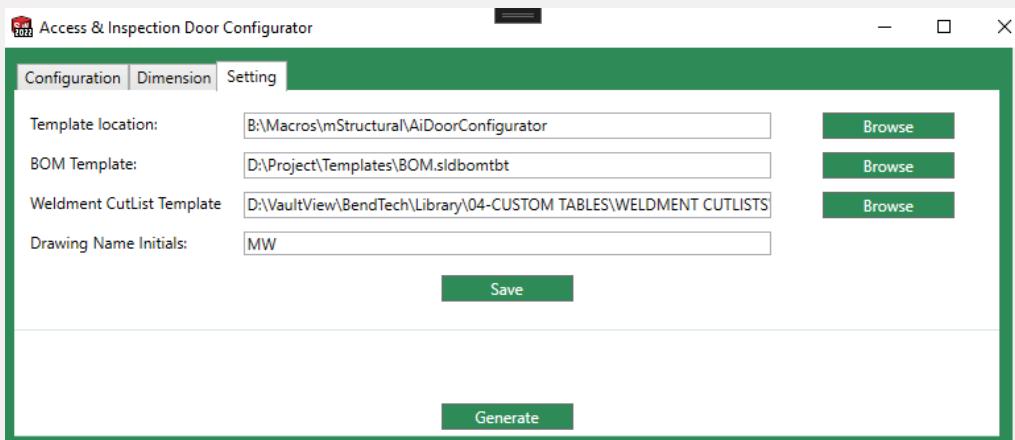
Alternatively, you can type or copy the file path manually.

8.3.4.4. Finally, you can press the Import button to import configuration.

Imported messages will be shown once completed. Change back to the first tab and you can see that the configuration inputs are updated.

Door type	Wire Guard Mesh Door
<input type="checkbox"/> Bolt-on mesh	
Door configuration	Single
Hinge position	LHS
Finishing	Hot dip galvanised
Mounting type	Bolted to chute
<input checked="" type="checkbox"/> Lock bar at rear (optional)	
Configuration imported! Check configuration tab.	

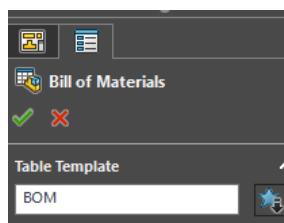
8.3.5. Tab 4 is for setting:



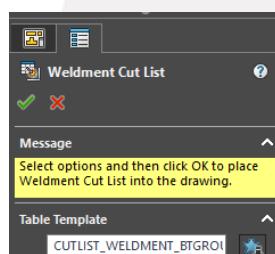
8.3.5.1. Template location: Location for all template model for this macro.

Default will be set at B:\Macros\mStructural\AiDoorConfigurator.

8.3.5.2. BOM Template: Location for your BOM template for cut list drawing.



8.3.5.3. Weldment Cut List Template: Location for your weldment cut list template for cut list drawing.



8.3.5.4. Drawing Name Initials: Your name initial for quotation and cut list drawing.



8.3.5.5. Save: Save all the settings. You must press this button if made any changes.

8.3.6. Generate: After all fields have been correctly filled in, press this button to generate the model. Cut List drawing and Quotation drawing will be generated together as well. For detailed logic behind the model and drawing generation, please refer to Access & Inspection Door Configurator rulebook by the Author of this manual.

NO.	QTY	DESCRIPTION	PART NUMBER	MATERIAL
101	1	DOOR FRAME	BT00001-PT-101	STEEL
102	1	JAIL BAR DOOR ASSEMBLY	BT00001-AS-150	STEEL
103	1	OUTER DOOR ASSEMBLY	BT00001-AS-200	STEEL
104	1	DANGER SIGN	BT00001-SIG-100	GRADE 304 STEEL AS 3678
105	16	NYLON WASHER 16 x 30 x 2	CUSTOM NO.1 NYLON WASHER ID16 x OD30 x 2	Nylon
105	2	M6 M12 x 80	M6x80	
105	4	M6 M12	M6x12	
105	4	M6 M12	M6x12	
108	2	NN M12	NN M12	
109	2	NYLON BARREL OD16 x ID12.5 x 31	NYLON BARREL OD16 x ID12.5 x 31	Nylon

BT00001-PT-101
DOOR FRAME

NO.	QTY	DESCRIPTION	LENGTH	MATERIAL
201	1	100 PFC	620	GRADE 304 STEEL AS 3678
202	1	100 PFC	620	GRADE 304 STEEL AS 3678
203	1	100 PFC	620	GRADE 304 STEEL AS 3678
204	1	100 PFC	620	GRADE 304 STEEL AS 3678
205	2	PLATE 5 x 60 x 150	620	GRADE 304 STEEL AS 3678
206	1	PLATE 16 x 50 x 70	620	GRADE 304 STEEL AS 3678

BT00001-PT-101
DOOR FRAME

REVISIONS

REV:	DESCRIPTION	MODIFIED BY	DATE
A	INITIAL CONCEPT	SMM	24/04/2014

By _____

REVISIONS

REV:	DESCRIPTION	MODIFIED BY	DATE
A	INITIAL CONCEPT	SMM	24/04/2014

By _____ Date: _____