

# SMC 2D/3D CAD Library

## User's Guide



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# 1. Program overview and basic operation.

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In this chapter, we will give an overview of the program, including how to save 3D CAD files from the DVD-ROM.

Please, read carefully this few steps before using the program for the first time.

## Overview

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“SMC 2D/3D CAD Library” is based on PARTsolutions (PARTdataManager program) by CADENAS, a three-dimensional CAD data collection. In addition of neutral CAD data formats, it is possible to generate native CAD format data, as described below.

Direct generation of native files. Each one of these CAD format can be used if it is installed on your environment, however, if Direct generation is not available on your environment, please use the Macro export feature.	Inventor 2014~2018 SolidWorks 2012~2017 Pro/Enginner Wildfire (Creo Parametric 1.0-3.0, Creo Elements/Pro 5.0)
Can be converted to native files. (Macro file export)	CATIA V5
Neutral format	STEP 3D IGES 3D SAT 3D DXF 3D DXF 2D

## 1.1. Operating environment

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These are the computer requirements to run PARTsolutions.

### OS (Operating System)

Windows7, Windows8, Windows10

### CPU

Intel Pentium 800MHz or above.

### Memory

700MB or above free memory, under the conditions CAD system running.

### DVD-ROM drive

### Graphic card

64MB video memory; OpenGL compatible; 3D hardware acceleration recommended

## 1.2. Basic operation

### 1.2.1. 2 ways of using this DVD.

- (1) Directly from your DVD-ROM drive.
- (2) By copying the program to your PC.

#### (1) Directly from your DVD-ROM drive.

- 1 Insert the DVD into your computer's DVD-ROM drive.
- 2 Upon first launch, english language will be displayed. You can choose other languages in the upper-right corner of the main window.



- 3 Clicking the “PARTdataManager” item will then launch PARTdataManager program and three-dimensional CAD library.



Please proceed to “1.3.2 product selection and CAD data save”.

The main menu consists in the following items.

- PARTdataManager ⇒Launch main program.
- Legal informations ⇒Please read before use.
- User's guide (PDF) ⇒Display this manual.

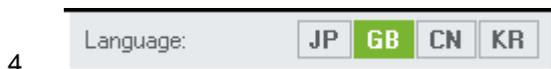
(2) By copying the program to your PC.

This is the use we recommend, as it will greatly reduce the program's response time.

- 1 Insert the DVD into your computer's DVD-ROM drive
- 2 Create a folder (by example [C:\¥SMC](#)) on your computer's hard drive and place the content of the DVD in this folder. Then, click on the start.bat file from the folder you created.



- 3 Upon first launch, english language will be displayed. You can choose other languages in the upper-right corner of the main window.

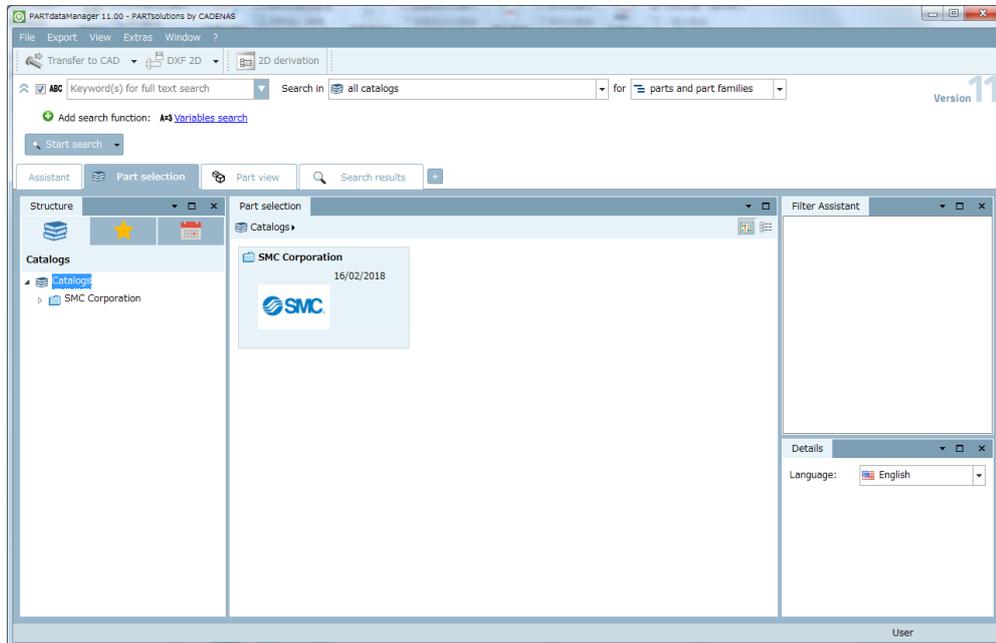


- 5 Clicking the “PARTdataManager” item will then launch PARTdataManager program and three-dimensional CAD library.

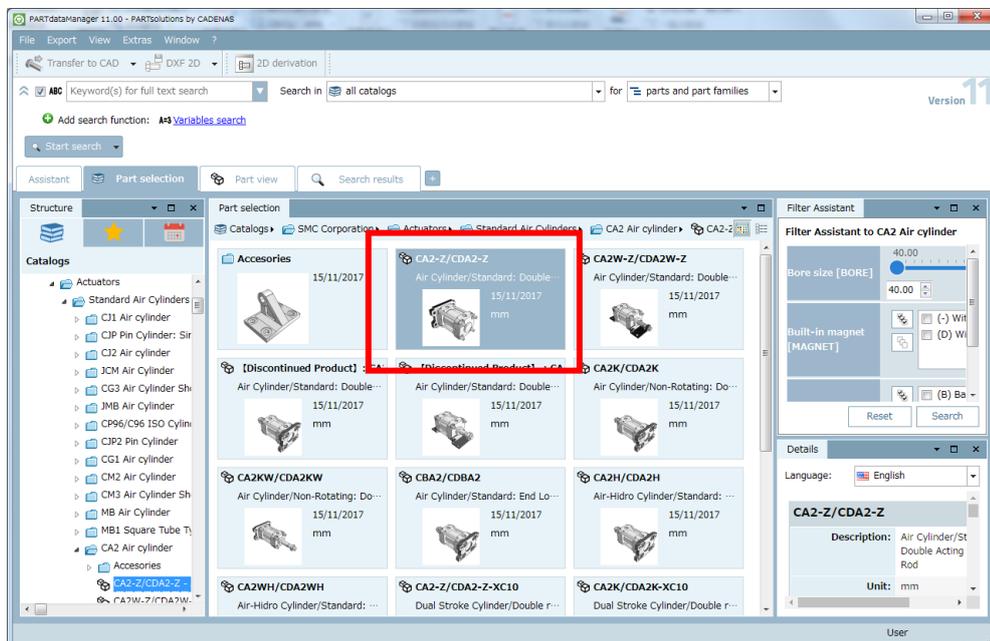


## 1.2.2. Product selection and CAD data save

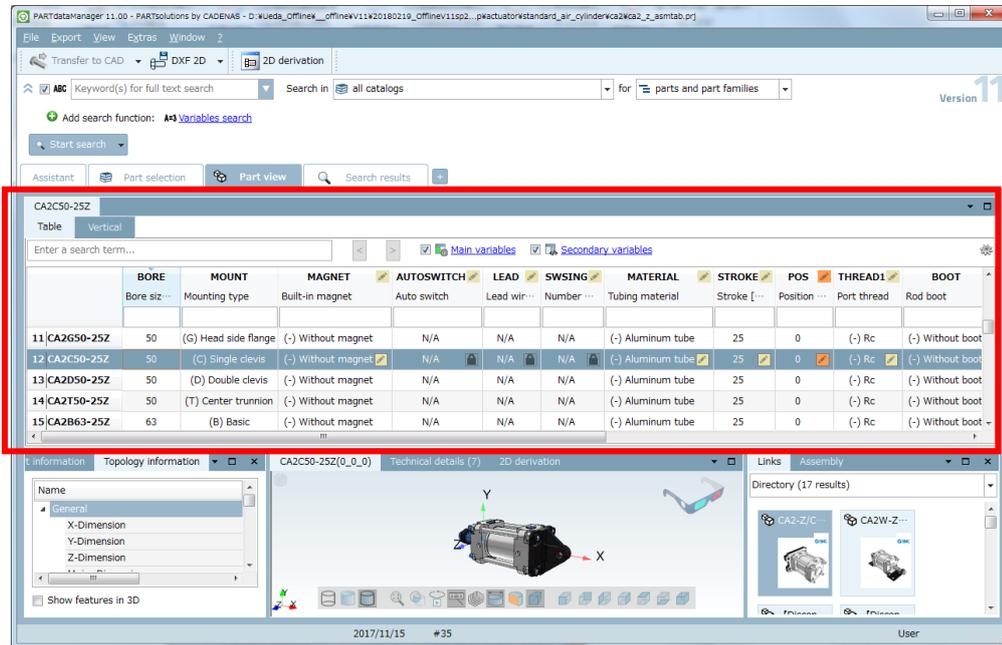
- 1 At startup, PARTdataManager will display the screen below.



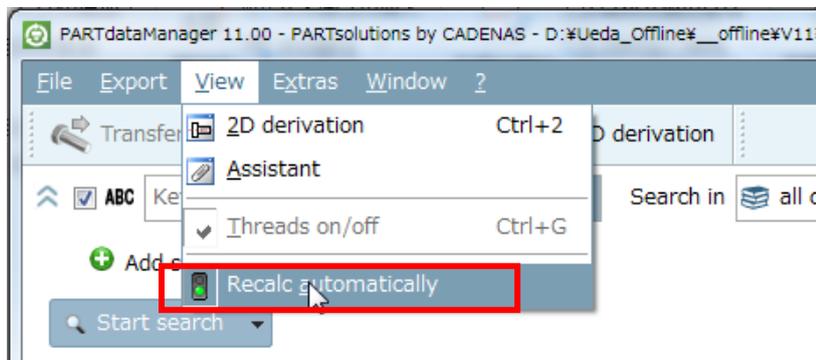
- 2 This is PARTdataManager's selection screen. From the tree view on the left, or the illustration menu on the right, you can select a product by double-clicking on it. In the following we will use "CA2 Air Cylinder" from "Actuator / Standard Air Cylinder" folder as an example. First we select CA2-Z Cylinder.



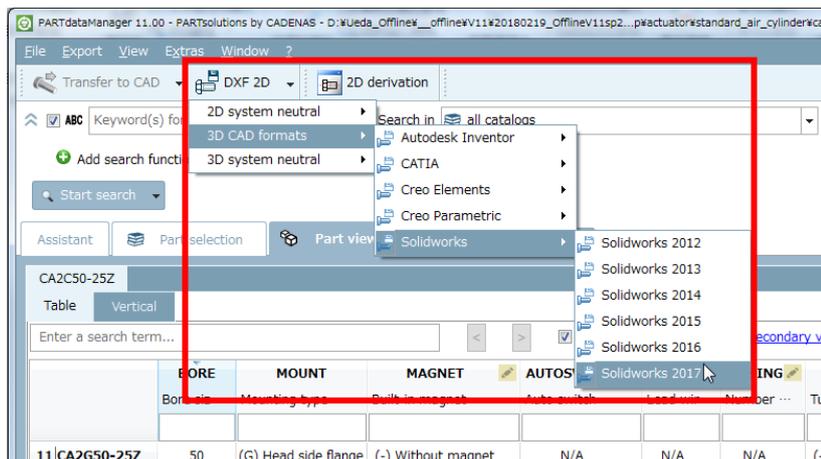
- The screen now changes to PARTdataManager's selection screen. In the red frame below, product parameters, such as Bore size, Mounting bracket and Stroke can be specified. When these options are changed, the 3D preview will be updated, and the corresponding three-dimensional CAD model will be displayed.



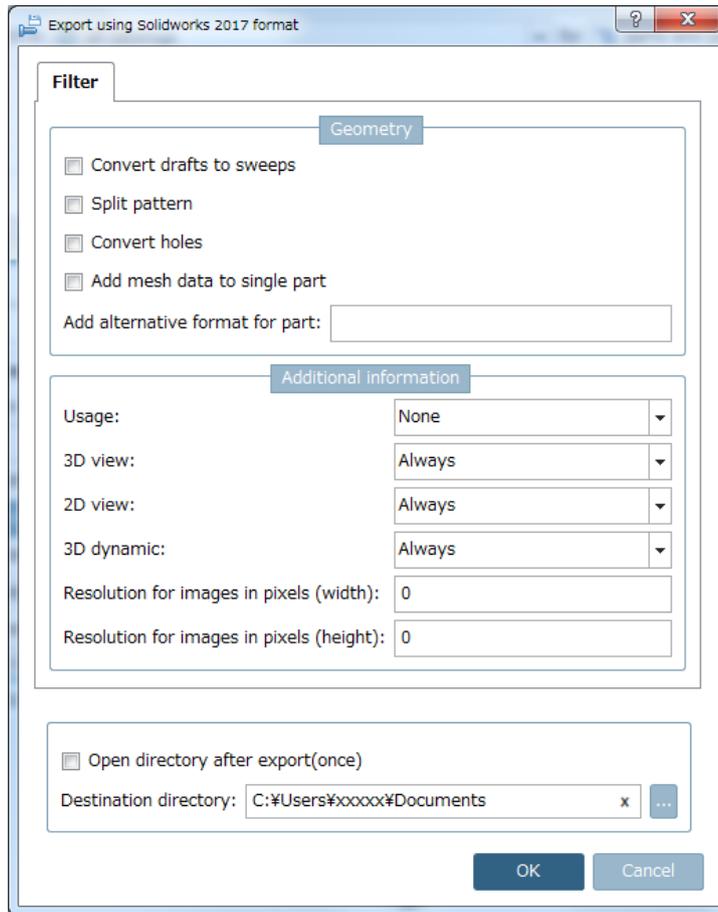
Automatic preview generation can take a long time depending on the data complexity. It is possible to turn it off by pressing the “Recalculate 3-D geometry on/off” button below. It is recommended to use the “off” setting for complex data.



- After displaying the specified product, we can finally export the corresponding 3D CAD data by selecting the desired format from the “Export in file” menu, as shown below.



- 5 Specify the file name in the next dialog box.
- 6 Then press OK to generate the file.



*(For more information, please refer to “2.2 CAD data export function”.)*

## 2. Individual features

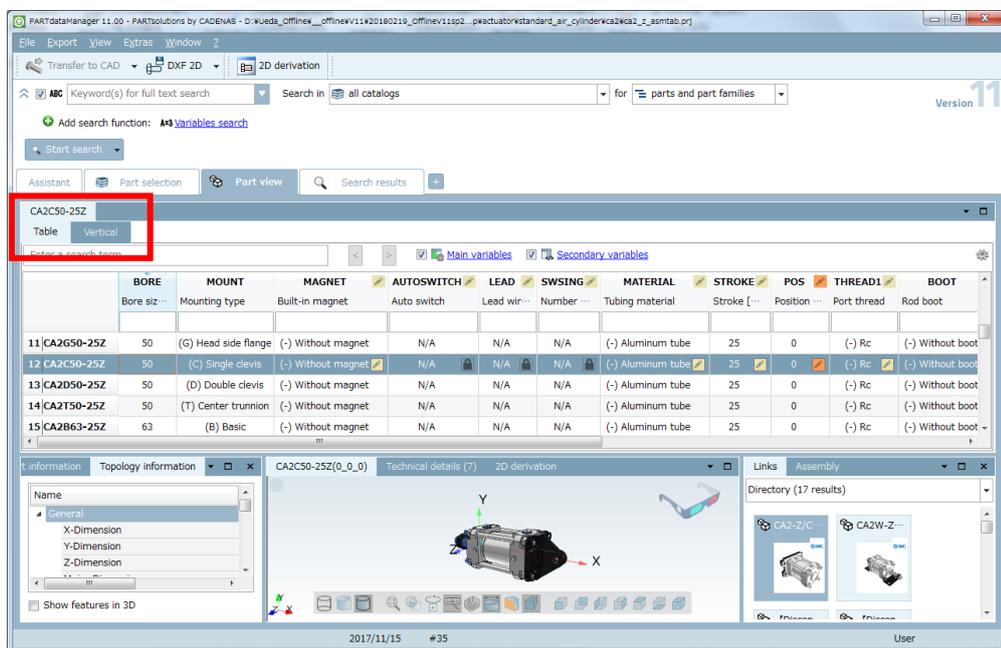
### 2.1. Component selection

#### 2.1.1. Switching part views

There are 2 views available to specify the details and options of the selected products.

It is possible to switch between these views from the icons below.

Standard view (Table view of parameters)



On the table, size etc. will be displayed. Select a row to activate it.

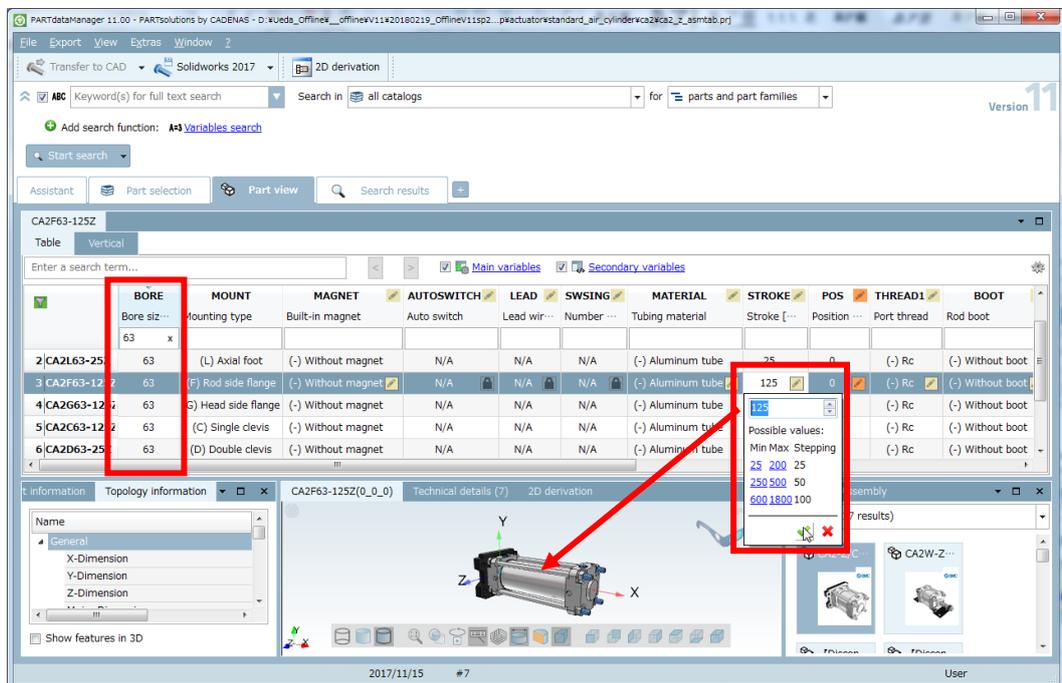
By selecting white cell information, like Bore, Mount, displayed for each product, the table can be easily narrowed.

	BORE	MOUNT	MAGNET	AUTOSWIT
	Bore siz...	Mounting type	Built-in magnet	Auto switch
1	CA2C40-25Z	(C) Single clevis	(-) Without magnet	N/A
2	CA2C50-25Z	(C) Single clevis	(-) Without magnet	N/A
3	CA2C63-25Z	(C) Single clevis	(-) Without magnet	N/A
4	CA2C80-25Z	(C) Single clevis	(-) Without magnet	N/A
5	CA2C100-25Z	(C) Single clevis	(-) Without magnet	N/A

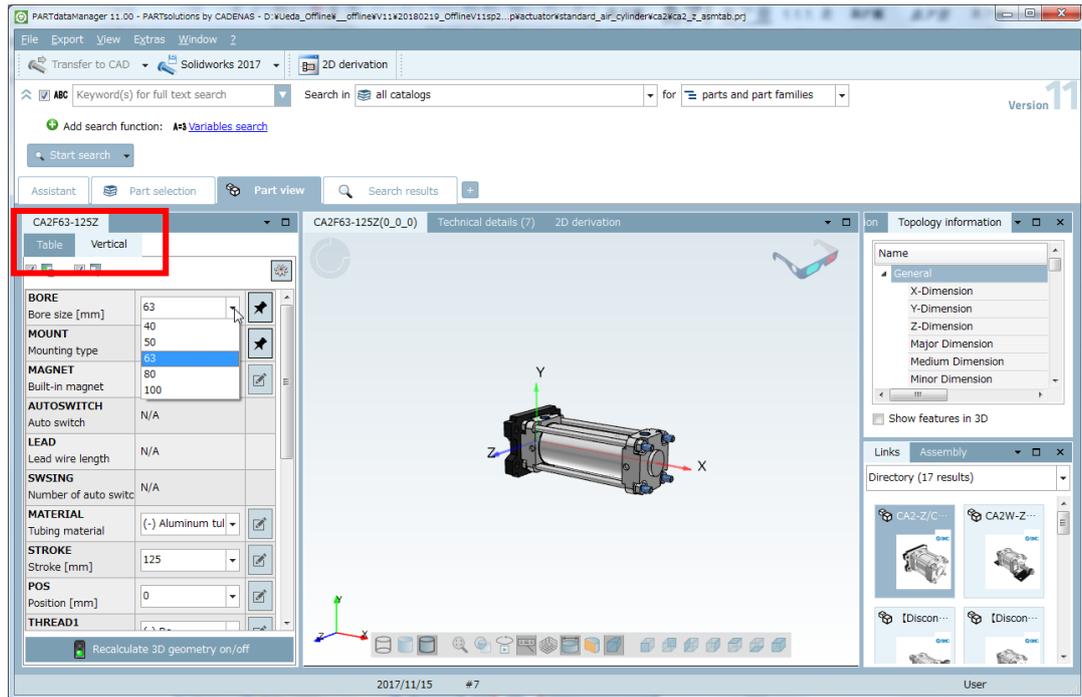
A yellow cell, like Stroke, Autoswitch, indicates a selection can be edited by clicking into it.

	BORE	MOUNT	MAGNET
	Bore siz...	Mounting type	Built-in magnet
1 CA2C40-25Z	40	(C) Single clevis	(-) Without magnet
2 CA2C50-25Z	50	(C) Single clevis	(-) Without magnet
3 CA2C63-25Z	63	(C) Single clevis	(-) Without magnet
4 CA2C80-25Z	80	(C) Single clevis	(D) With magnet
5 CA2C100-25Z	100	(C) Single clevis	

By selecting the value 63 as the Inner Diameter of the tube below, it is possible to narrow the choices that can be specified for the Stroke.



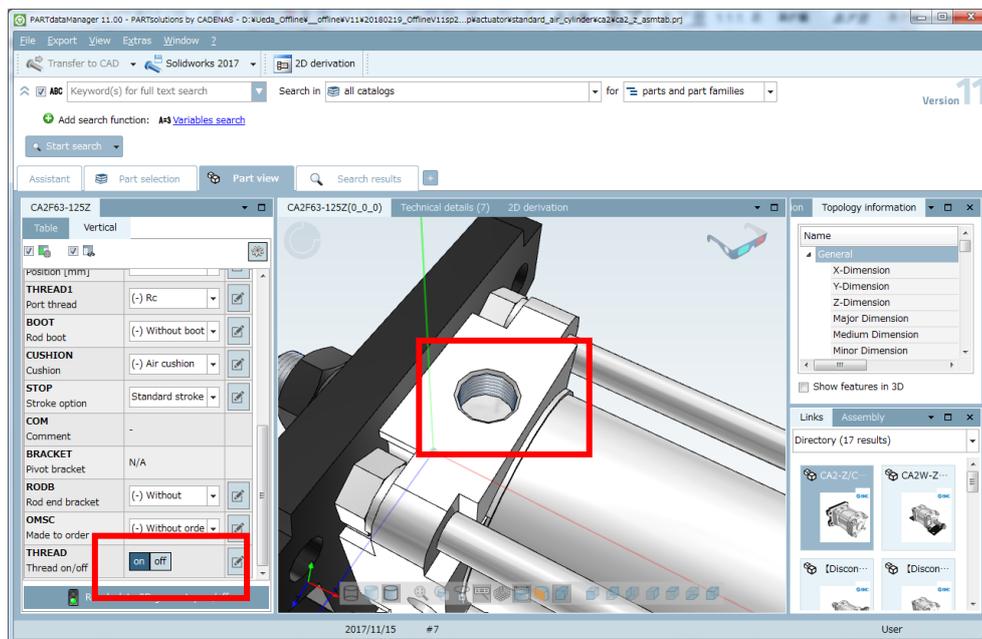
## Vertical view

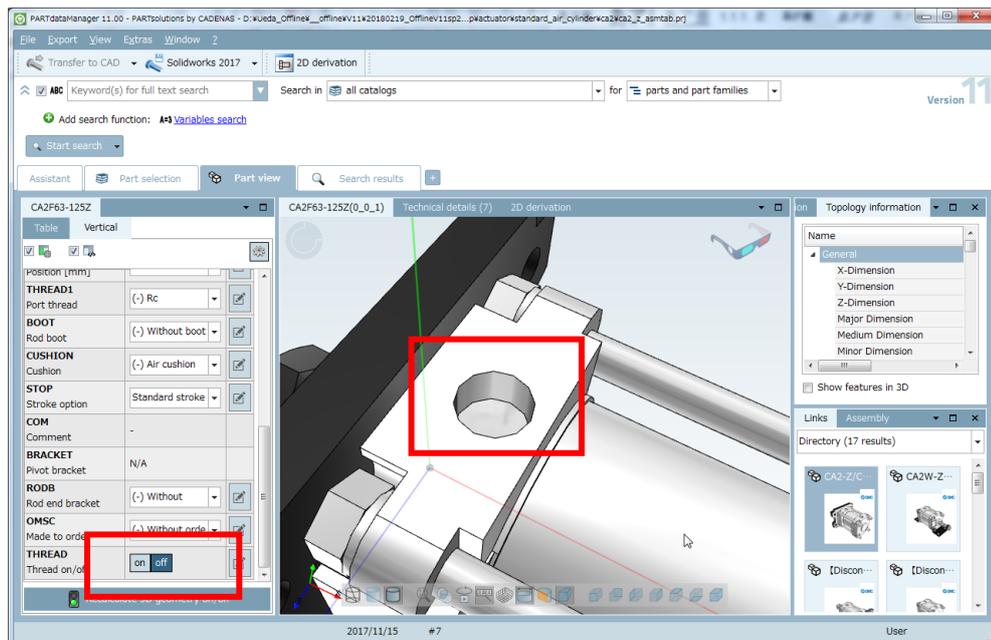


When Vertical view is selected, the optional selection (size or switches) can be configured one by one.

### 2.1.2. Screw thread

It is possible to turn on and off the display of the screw threads in the preview. As shown below, when the screw thread preview is set to off, the model will show empty screw holes.



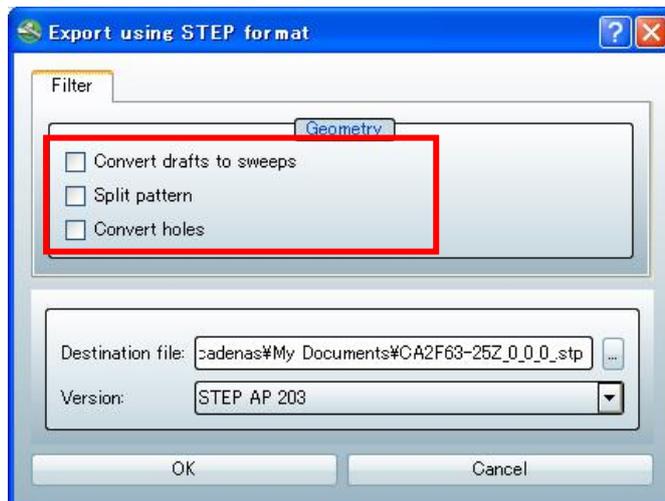


## 2.2. CAD data export function (2D - 3D)

Export conditions depend on the selected format.

For intermediate formats such as STEP and DXF, and CAD macro files, it is possible to export data even if the corresponding CAD system is not installed on your environment. However, in order to export native CAD formats, the corresponding CAD system must be installed.

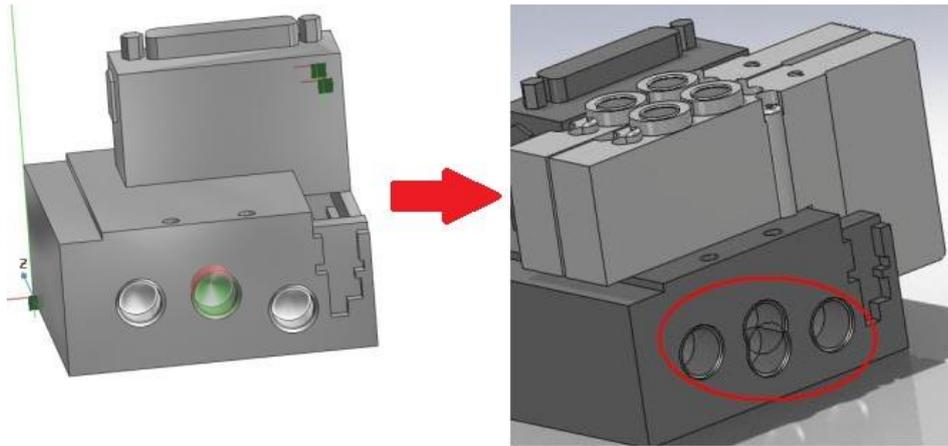
### 2.2.1. Caution



On the 3D export data dialog box, you will have choice between 3 options.

In case the exported data differs from the preview displayed in PARTSolutions, you can use one of these options to resolve this issue.

Typical problem example :



(Left : Original model / Right : data exported in Solidworks)

In the above example, the original model has been created using pattern array feature, but the CAD exported result hasn't been properly created.

Such problem can be avoided by choosing the “Split pattern” option. Although we fully understand this might be troublesome in some cases, we kindly ask you to use this geometry option when you encounter such a problem.

Note : if you don't check any geometry option, the CAD exported model will be created using pattern array feature. In most cases, this feature doesn't lead to any problem. However, there are some rare cases where the CAD data can't be exported properly using pattern array feature. By using this option, the CAD data will be exported without pattern array. In this case, each feature of the pattern will be generated separately, leading to a slightly longer generation time. The number of features will be greater, but the model will be accurately generated.

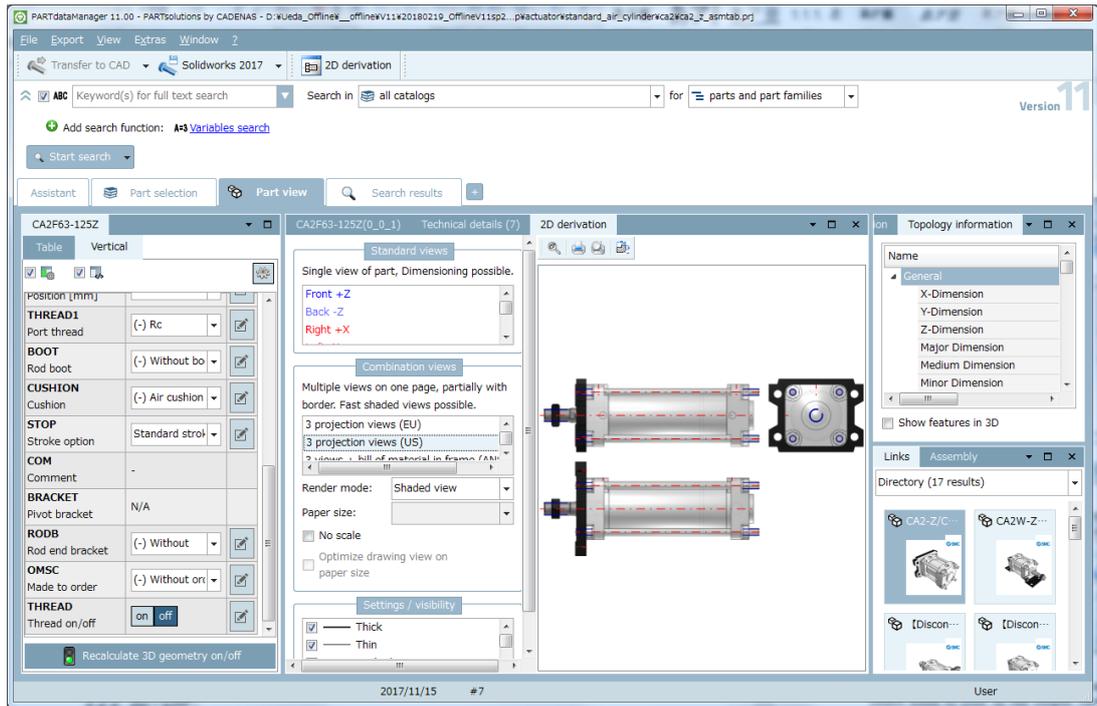
Similarly, if in some very rare occasions, the hole positions are not accurately modeled in the generated data, please use the “Convert holes” option. In this case, CAD model will be exported without holes feature.

### 2.2.2. 2D - DXF

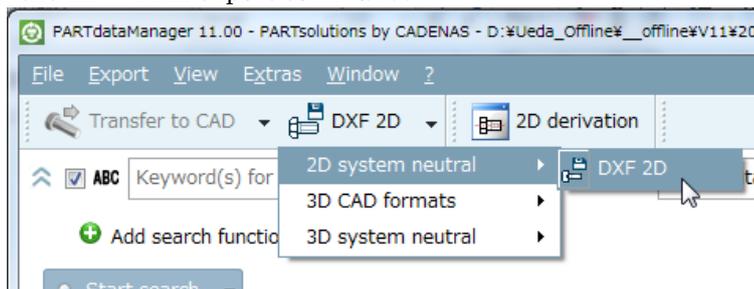


When “2-D derivation” is selected, a 2D projection of the current selection will be displayed alongside the 3D preview.

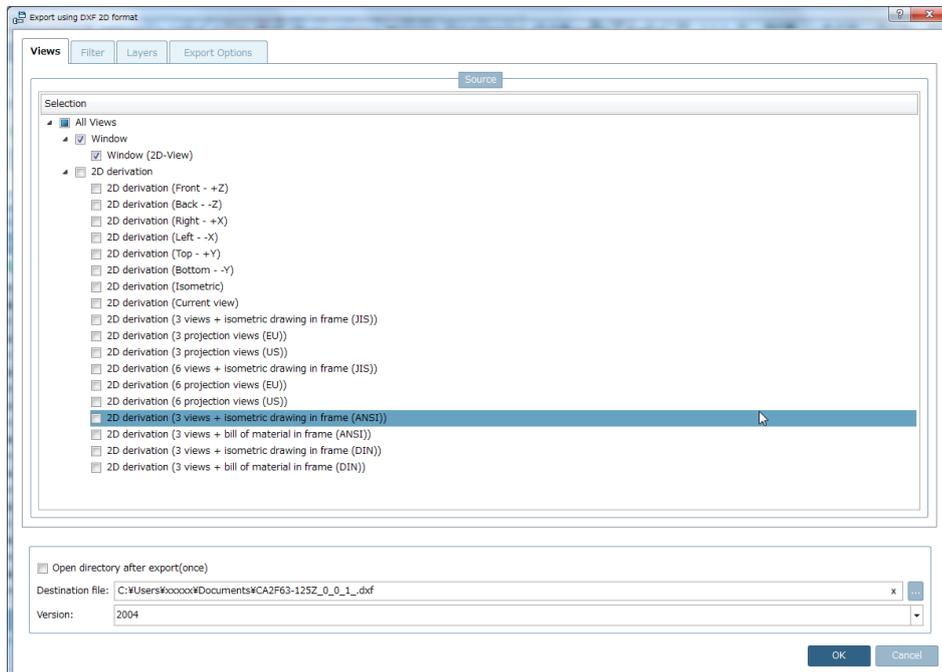
As shown below, the 3 surfaces of the model are now visible.



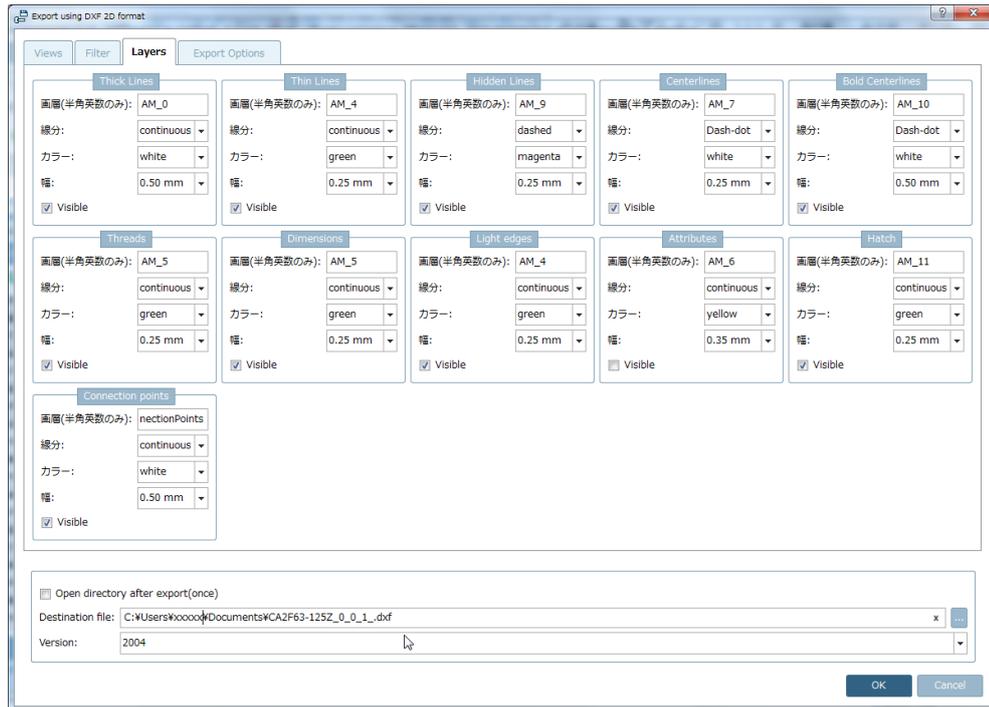
Now we run the 2D – DFX export command.



The actual export of the DXF file is done in the following dialog box. Select write to DXF in the output view source. (From the “source” drop-down list, choose “Selected views”, then select the view you want to export by clicking its corresponding checkbox.)



In the Export options tab, it is possible to set the origin, attribute and layer.



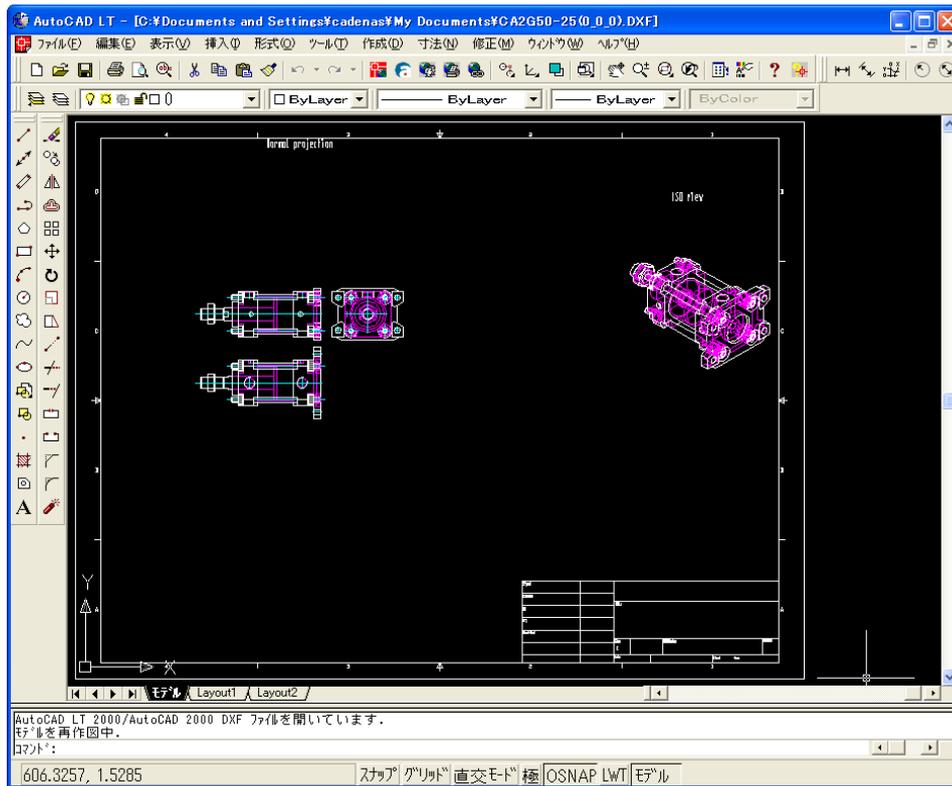
### 2.2.3. 2D – DXF for use in AutoCAD

Choose the appropriate view for 2D - DXF

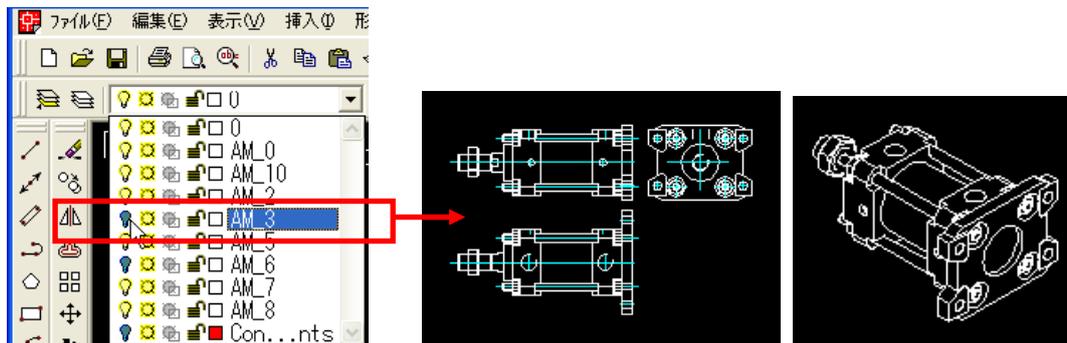
Example :



As shown below, when you open a DXF file in AutoCAD, the layer and color code for each type of line will be conform to the option selected in the export options dialog box.

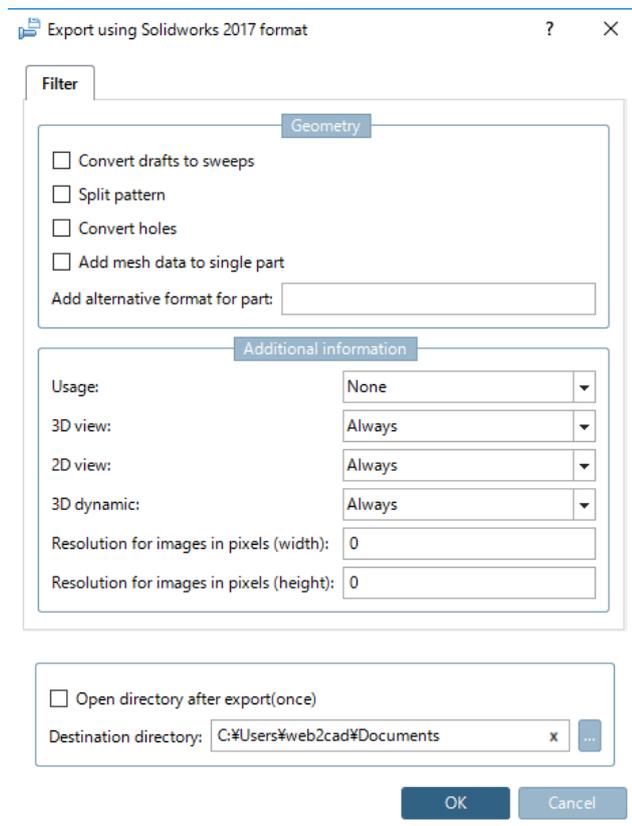
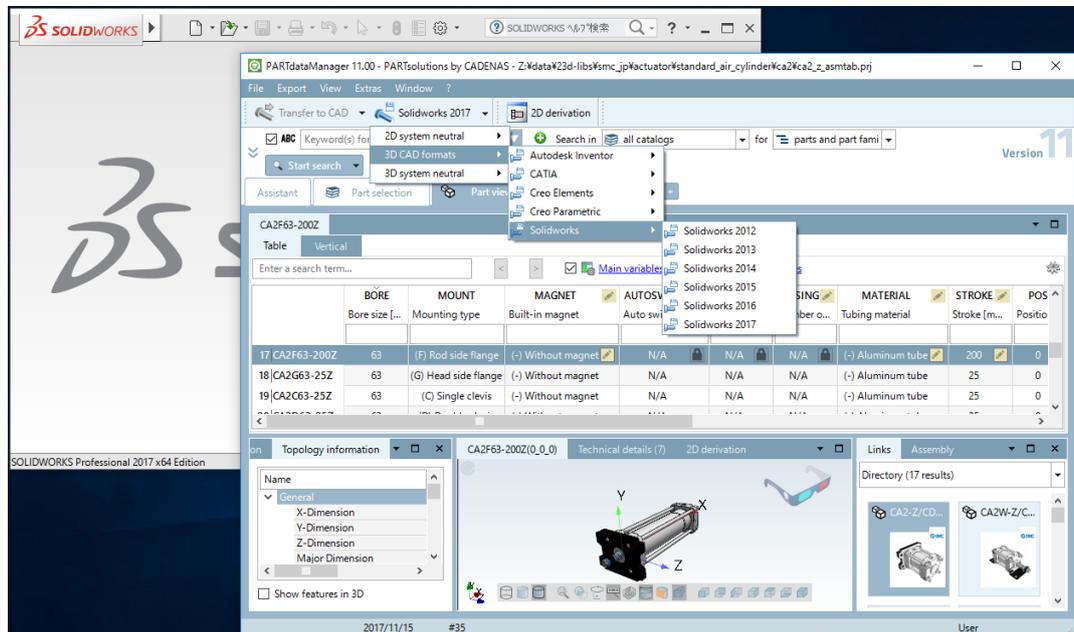


It is also possible to hide unwanted lines in the layer settings.



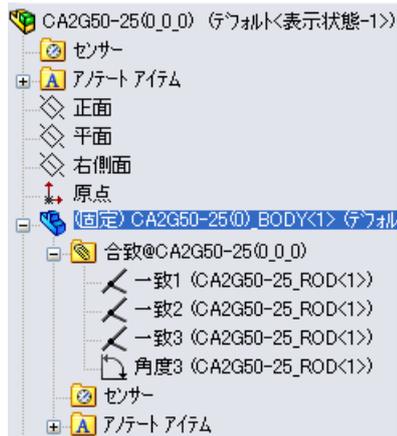
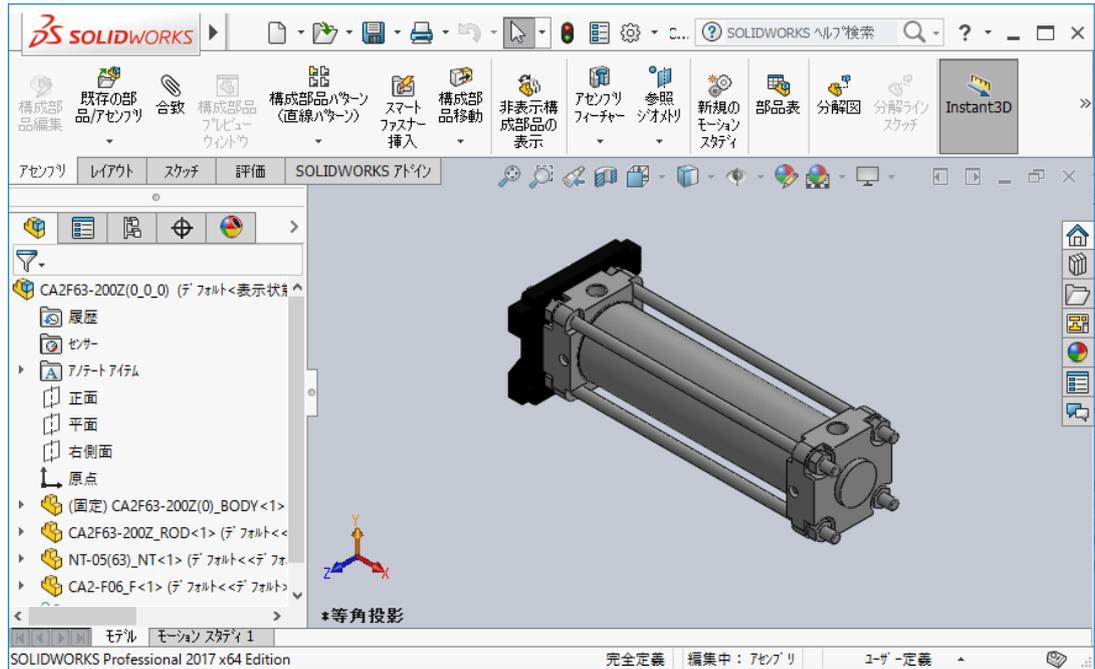
## 2.2.4. Solidworks 2017 – (Direct)

In order to use 3D data, the most efficient way is to export CAD data through direct driver.



As pictured in the illustration, after the product selection, start Solidworks 2017 in background, and select "Solidworks 2017" (or the version that matches your CAD system) from the export menu. After confirming the export directory, press the "OK" button, then the model will be drawn in the background Solidworks window. (In case of a multiple-parts data assembly, this process may take some time)

The illustration below shows the completed model in Solidworks.

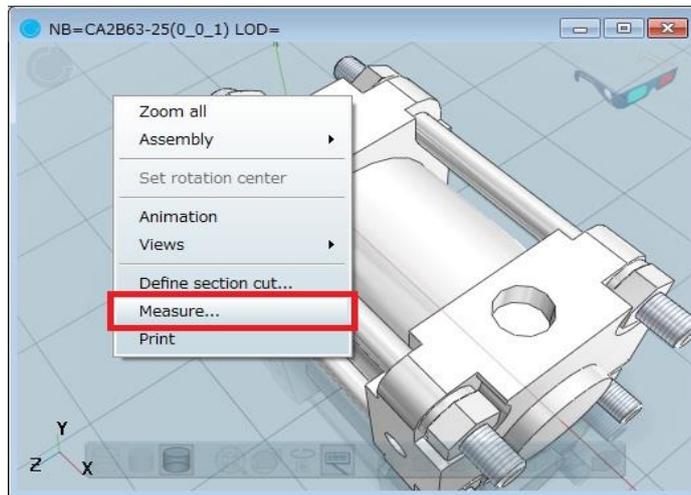


Since the data will be modeled completely on Solidworks, you can see on the illustration above that the model is created with all matching features (bound), and other features such as screw, therefore making it a native data.

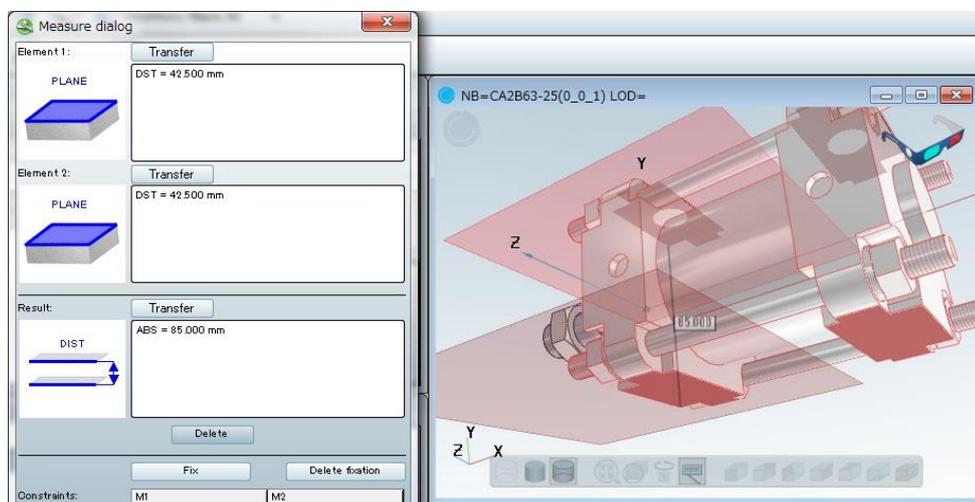
## 2.3. Dimensions measurement feature

### 2.3.1. Measuring on the 3D preview screen.

You can access the measuring command via the right-click menu on the 3D preview screen.

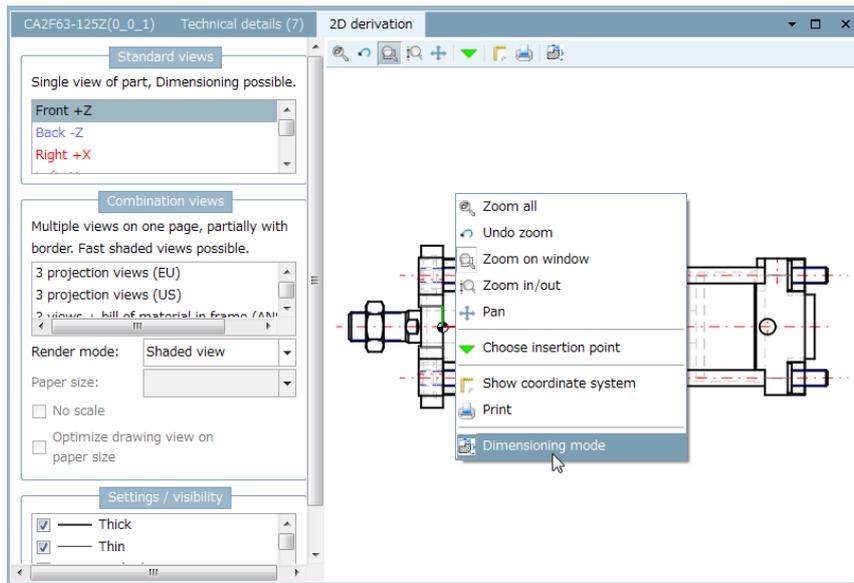


As shown below, in measurement mode, you will be able to see the model's size, like distance between two surfaces.



### 2.3.2. Measuring on 2D view

You can access the dimensioning command via the right-click menu on the 2D view screen.



As shown in the illustration, the dimensioning mode will allow you to check the model size via the dimensioning menu.

