

# Batch Flat Pattern Exporter Training

Batch Flat Pattern Exporter\_Training.zip



# Batch Flat Pattern Exporter - Overview

## Main Functions

- Batch export sheet metal part to DWG/DXF Flat Pattern
- Batch export Flat Pattern with bend line, layer and template
- Automatically generate SOLIDWORKS Drawing with Flat Pattern

## Benefits

- Automated development drawing work for processing, reducing work time
- Post processing of layer and bend line reducing work time.

**Course Objective:** Follow the simple step-by-step instructions and learn how to use Batch Flat Pattern Exporter.

The screenshot displays the Batch Flat Pattern Exporter software interface. The main window shows a list of files to be exported, including Model\_01.SLDPRJT, Model\_02.SLDPRJT, Model\_03.SLDPRJT, and Model\_04.SLDPRJT. Below the list are buttons for 'Delete All', 'Add Open File', 'Add File', and 'Add Folder'. A green arrow points from the 'Add Open File' button to a file icon labeled 'DXF'. The 'Export Option' window shows a 3D model of a sheet metal part and a 'Set Detailed Option' button. The 'Set detailed option' window shows the 'Drawing Templates' section with the 'Drawing template file path' set to 'C:\ProgramData\SolidIT\SpeedWorks\BatchFlatDraw\BlankDraw.DRWDO'. The 'Layer' section shows a table of custom layers with columns for Item, Color, Style, and Thickness. The 'Mapping' section shows a grid with a red cube labeled 'SW'.

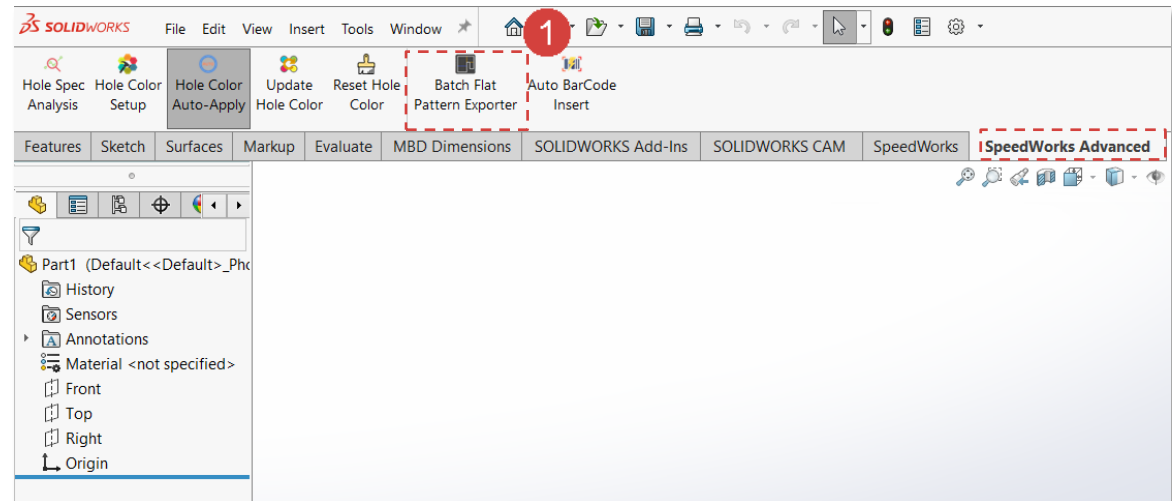
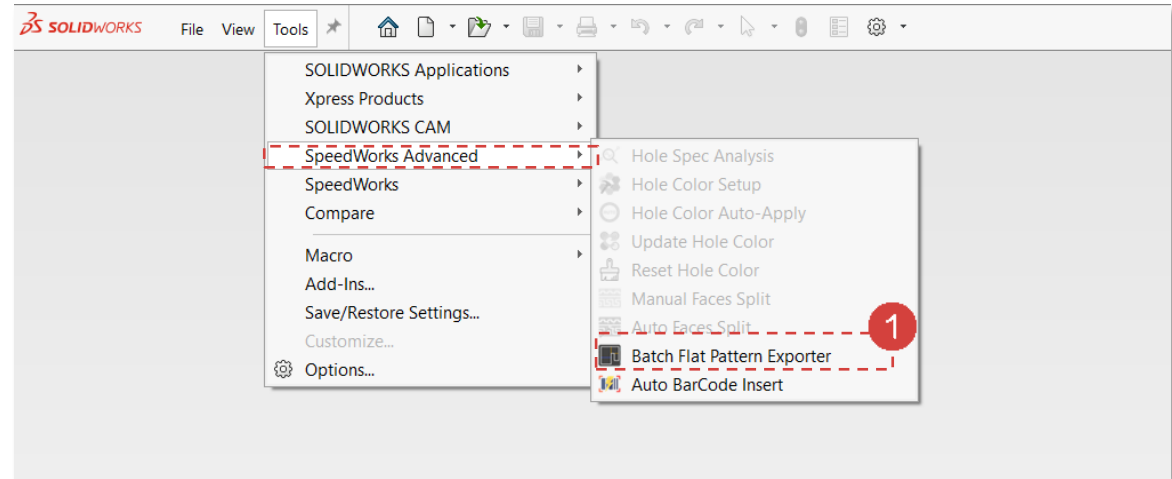
Item	Color	Style	Thickness
Outline	255, 255, 255	Continuous	0.25mm
Bend line-Up	124, 252, 0	Chain	0.18mm
Bend line-Down	79, 129, 189	Chain	0.18mm
Bend Note-Up	255, 255, 0	Continuous	0.18mm
Bend Note-Down	255, 255, 0	Continuous	0.18mm

## Batch Flat Pattern Exporter - Execute

Work on the part with the red circle (1) following instructions by yourself.  
The part with the black square (1) is an additional explanation or a reference.

1 Click **SOLIDWORKS** - [Tool] - [SpeedWorks Advanced] - [Batch Flat Pattern Exporter] to execute function.

or click [SpeedWorks Advanced]tab - [Batch Flat Pattern Exporter] to run the tool in part/ assembly/ drawing file.



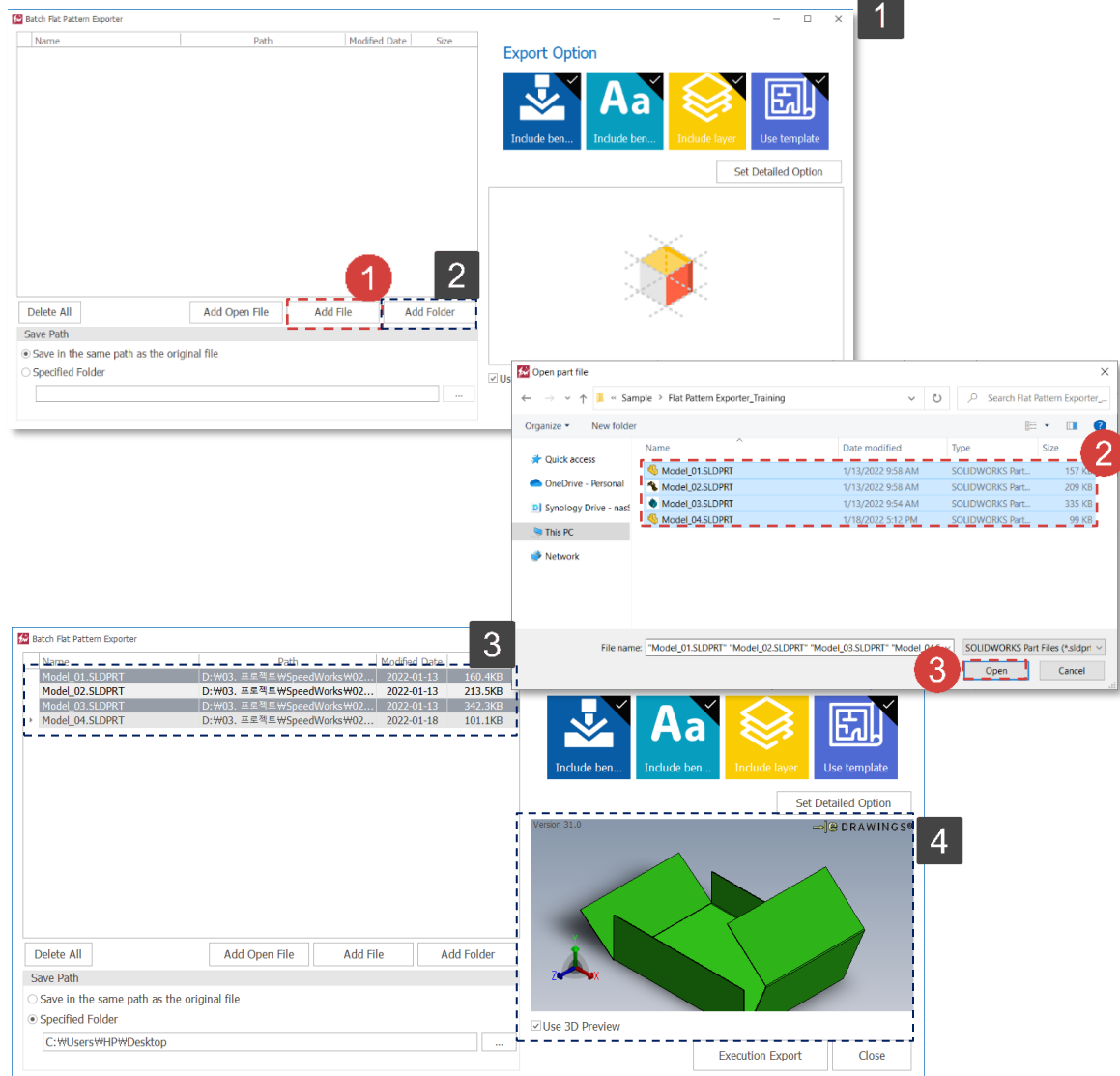
※ You can download example files at Support page in SpeedWorks Homepage(<https://speedworks.info>).

## Batch Flat Pattern Exporter - Adding Sheet Metal

Work on the part with the red circle (1) following instructions by yourself.  
The part with the black square (2) is an additional explanation or a reference.

- 1 Click **[Add file]** button.
- 2 Select **Model\_01.SLDPRT, Model\_02.SLDPRT, Model\_03.SLDPRT, Model\_04.SLDPRT** in sample file.
- 3 Click **[Open]** button to add file.

- 1 It is Batch Flat Pattern Exporter window.
- 2 Through **[Add folder]** button, batch add SOLIDWORKS part files.
- 3 This is part added target list.
- 4 Users can check 3D preview when select an item on list.

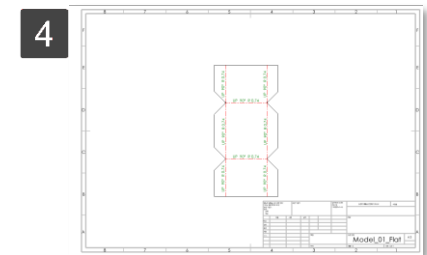
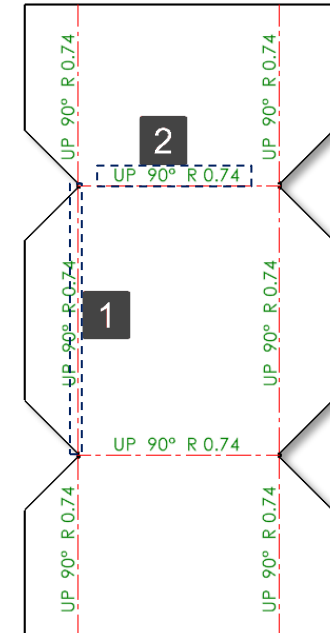
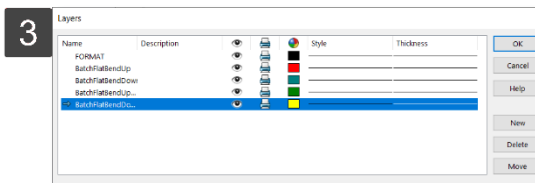
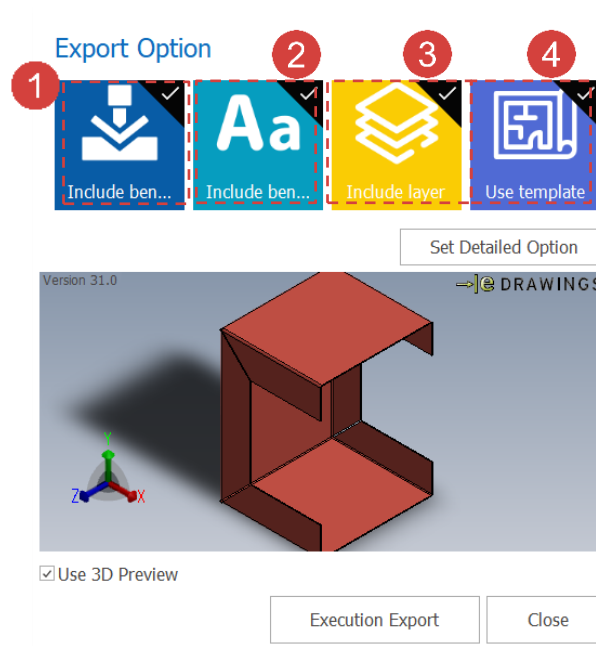


## Batch Flat Pattern Exporter - Export Option

Work on the part with the red circle (1) following instructions by yourself.  
The part with the black square (2) is an additional explanation or a reference.

- 1 Check [Export Option] - [Include bendline].
- 2 Check [Export Option] - [Include bendnote].
- 3 Check [Export Option] - [Include Layer].
- 4 Check [Export Option] - [Use Template].

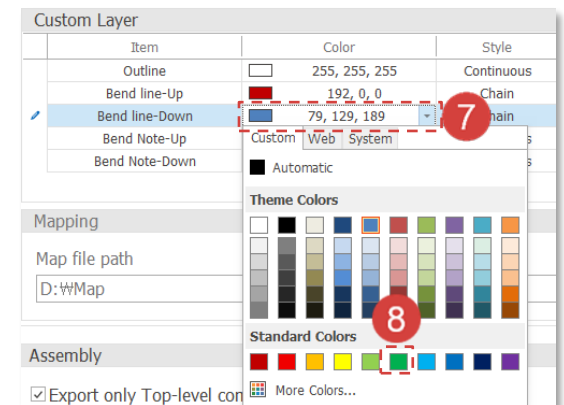
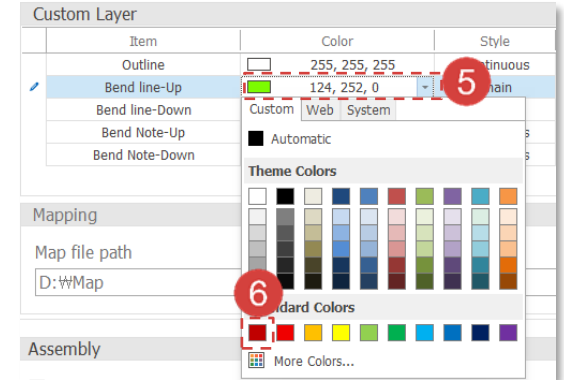
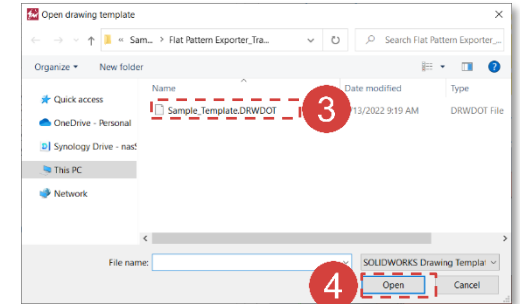
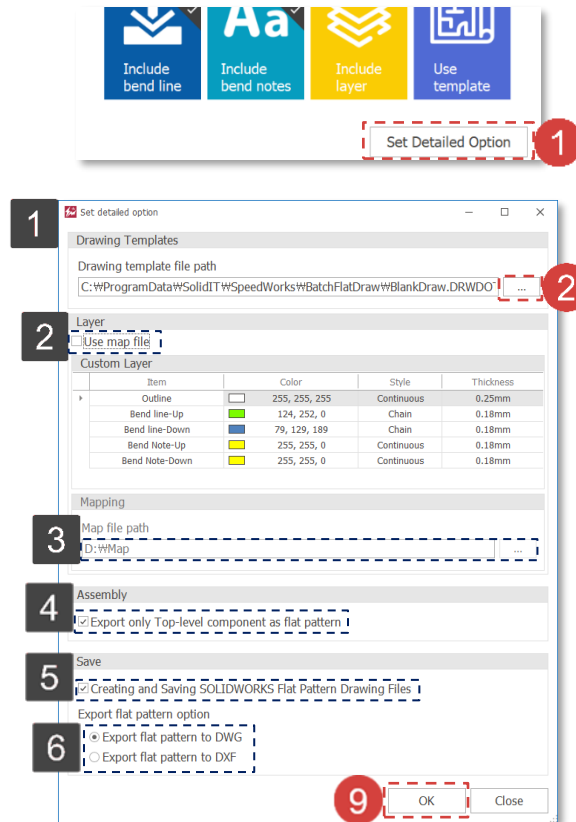
- 1 Output including bendline, when [Export Option] - [Include bendline] is checked.
- 2 Output including bendnote, when [Export Option] - [Include bendnote] is checked.
- 3 Exporting SOLIDWORKS drawing be applied to flat pattern, when [Export Option] - [Include Layer] is checked.
- 4 Exporting SOLIDWORKS drawing, specified drawing template is applied to Flat pattern view, when [Export Option] - [Use Template] is checked.
- 5 [Export Option] is checked status.
- 6 [Export Option] is not checked status.



## Batch Flat Pattern Exporter - Set Detailed Option

- 1 Click **[Set Detailed Option]** button.
- 2 Click **[Drawing Template Path]** - [...] button.
- 3 Select **Sample\_Template.DRWDOT** in sample file.
- 4 Click **[Open]** button.
- 5 Select **[Bend line-Up]** - [Color].
- 6 Select **Red** color.
- 7 Select **[Bend Note-Up]** - [Color].
- 8 Select **Green**.
- 9 Click **[OK]** button.

- 1 It is **[Set detailed option]** screen.
- 2 You can apply a mapping file.
- 3 Enter the map file storage path.
- 4 You can select flat pattern output for sheet metal parts in a subassembly.
- 5 This option allows users to create **SOLIDWORKS** drawings together when outputting a flat pattern.
- 6 This option allows users to output the flat pattern output file format by selecting either DWG or DXF.

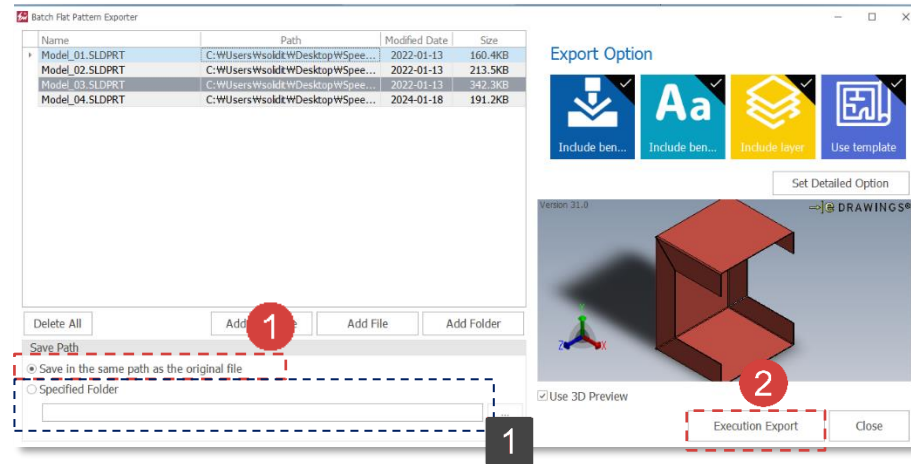


Work on the part with the red circle (1) following instructions by yourself.  
The part with the black square (1) is an additional explanation or a reference.

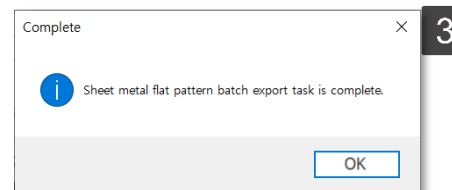
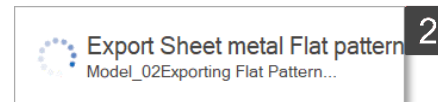
## Batch Flat Pattern Exporter - Export Flat Pattern

Work on the part with the red circle (1) following instructions by yourself.  
The part with the black square (1) is an additional explanation or a reference.

- 1 Select [Save Path] - [Save in the same path as the original file].
- 2 Select [Execution Export].



- 1 Output files can be exported to a specified location through [Specified Folder].
- 2 Users can check the status of the flat pattern output job in progress.
- 3 When the operation is complete, a completion message window appears.

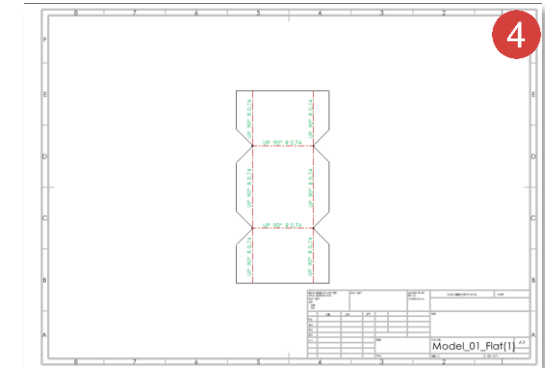
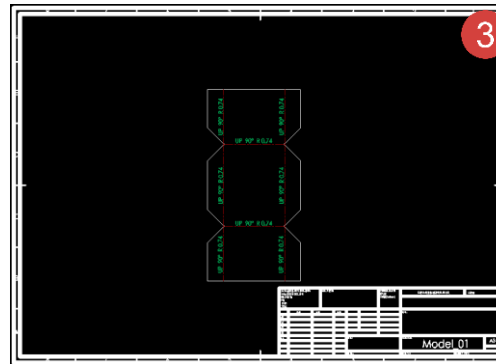


## Batch Flat Pattern Exporter - Result

Work on the part with the red circle (1) following instructions by yourself.  
The part with the black square (1) is an additional explanation or a reference.

- 1 A DWG file was created in the same path as the output destination part file.
- 2 A SOLIDWORKS drawing file was created in the same path as the output destination part file.
- 3 Bend lines, bend notes, layers, and drawing templates are all specified in the DWG drawing..
- 4 A SOLIDWORKS drawing has been assigned a **flat pattern view** insertion, **layers**, and **drawing templates**.

Name	Date modified	Type
Hole Color Editor_Training	3/7/2022 8:48 PM	File folder
Sample_Template.DRWDOT	1/13/2022 9:19 AM	DRWDOT File
Model_01.DWG	3/4/2022 2:57 PM	DWG File
Model_02.DWG	3/4/2022 2:57 PM	DWG File
Model_03.DWG	3/4/2022 2:57 PM	DWG File
Model_04.DWG	3/4/2022 2:57 PM	DWG File
Model_01.SLDPRT	1/13/2022 9:58 AM	SOLIDWORKS Part...
Model_02.SLDPRT	1/13/2022 9:58 AM	SOLIDWORKS Part...
Model_03.SLDPRT	1/13/2022 9:54 AM	SOLIDWORKS Part...
Model_04.SLDPRT	1/18/2022 5:12 PM	SOLIDWORKS Part...



- 1 If a name that is duplicated with the file to be output occurs, a number is appended to the file name as (1), (2), (3).

- 2 SOLIDWORKS flat pattern drawing file names are appended with the letters \_Flat.

