3DXpert™ for SOLIDWORKS®

Slicing
Slice multiple parts

Tutorial S3: 14,0200,1599,1024(SP2)

3D SYSTEMS

In this exercise, we will learn how to perform **Slicing on multiple part**.

Slicing is the stage where the printing layers are set according to the printer definition or printing strategy.

To use this command we need to follow few steps (guided):

- Open **3D Printing Project** from the Initial screen.
- Create slicing to the parts with **Calculate Slicing** command.
- View and check slicing with **Slice Viewer** tool.
- Output file to printer and/or to file using **Send to Print** command.

! Notice/ Remember		Left mouse button name is **"pick"**
		Middle mouse button name is **"Exit"**
		Right mouse button name is **"Click"**

1. From the Initial screen *pick* **Open File**.
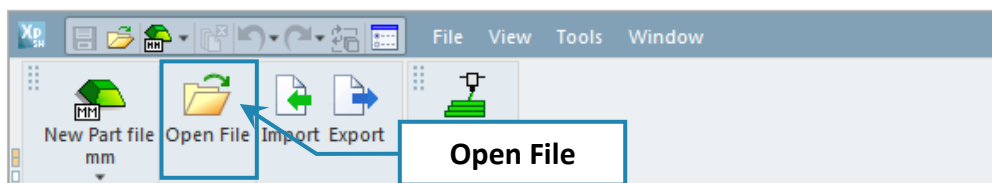
2. This command will open the **3DXpert for SOLIDWORKS Explorer**.
   Load file **3DP_Slicing_Project.elt** from the same folder where the downloaded files exist.
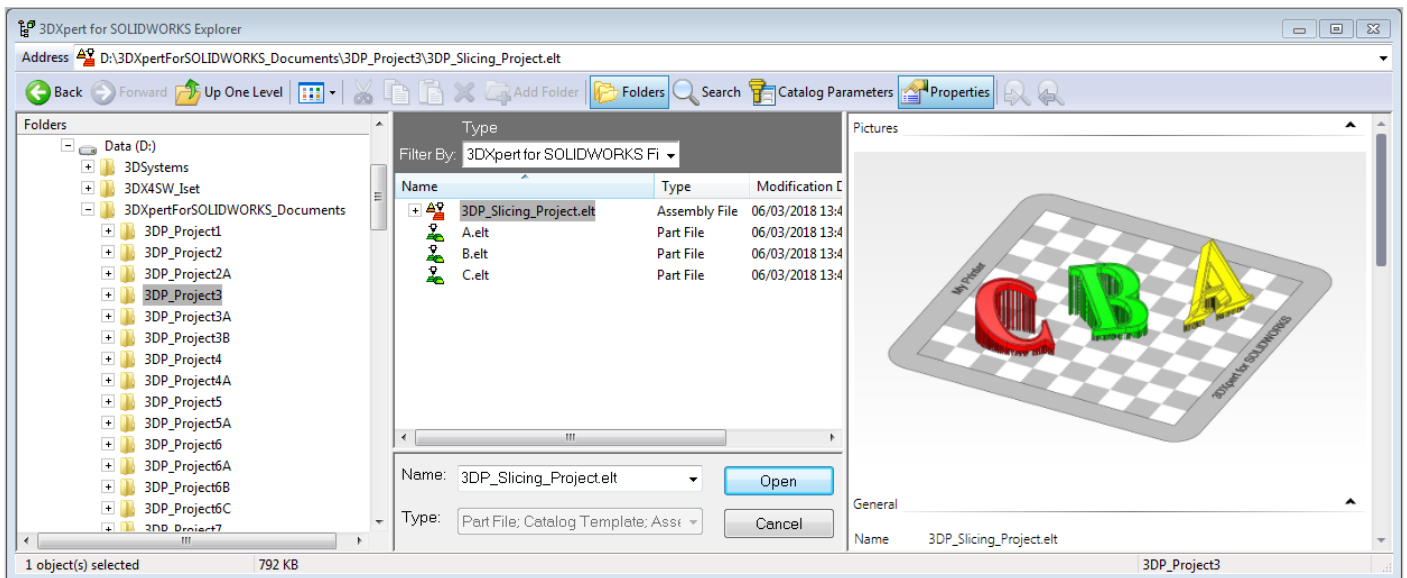
The project contains three parts, predefined with position and supports (the supports were done using the default Meta-Template. These supports are used here only for the purpose of this exercise, not as a printing guide).

Once the file is open, the screen looks like this:
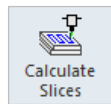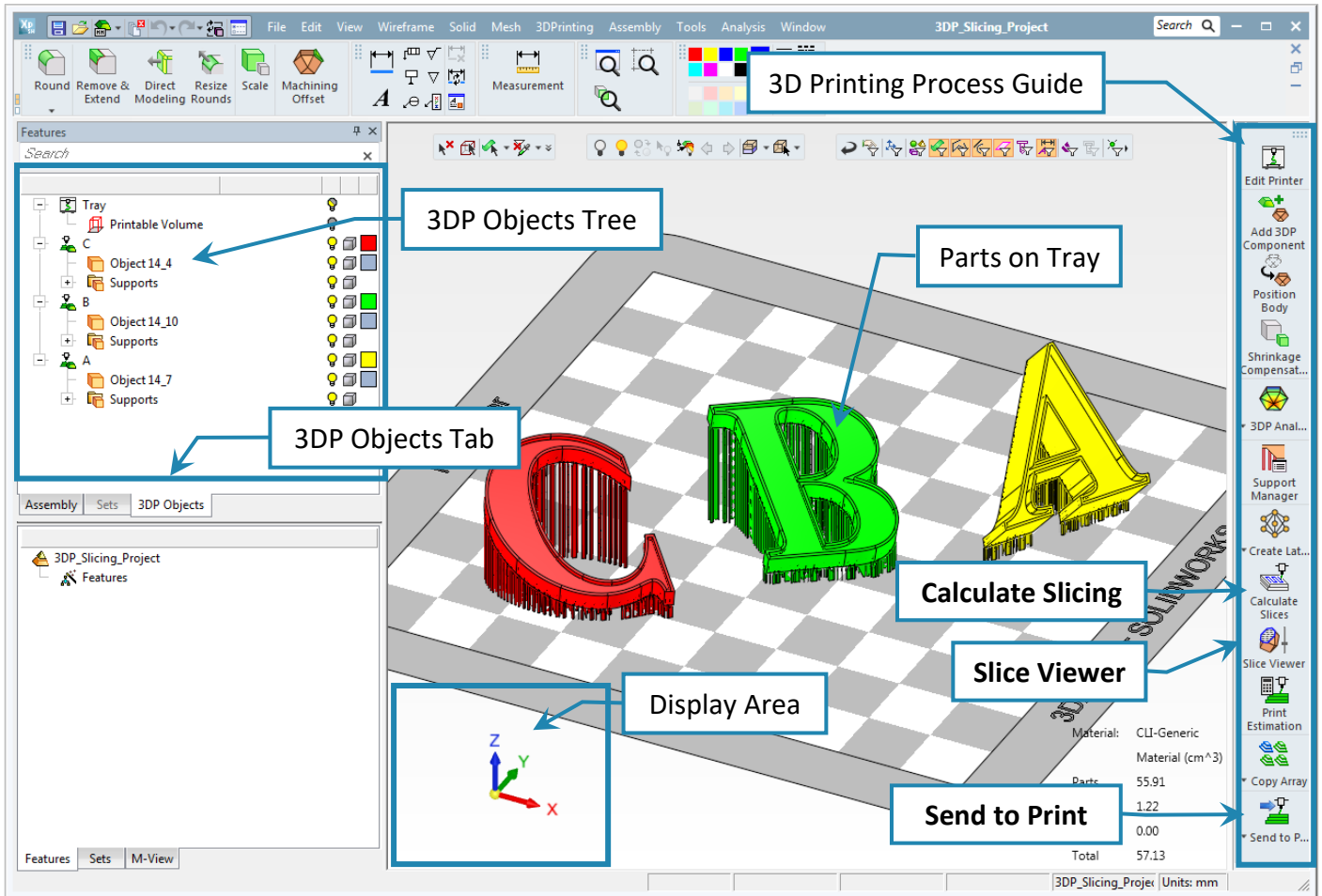
3. *Pick* the **Calculate Slicing** command from the **3D Printing Process Guide**.

Feature Guide		**Calculate Slicing**
Slicer Tool		1) Select Objects (the default is **Multi 3DP Components**).
		2) Set Parameters.
		The systems moves automatically from1) to 2) in case of **Multi 3DP Components**.
	✔	To approve and finish use the *"OK"*
	✔	To approve and continue use the *"Apply"*.
	✗	*"Cancel"* – exit the command without keep changes

On the screen **Object Slicing** parameters window shows up:
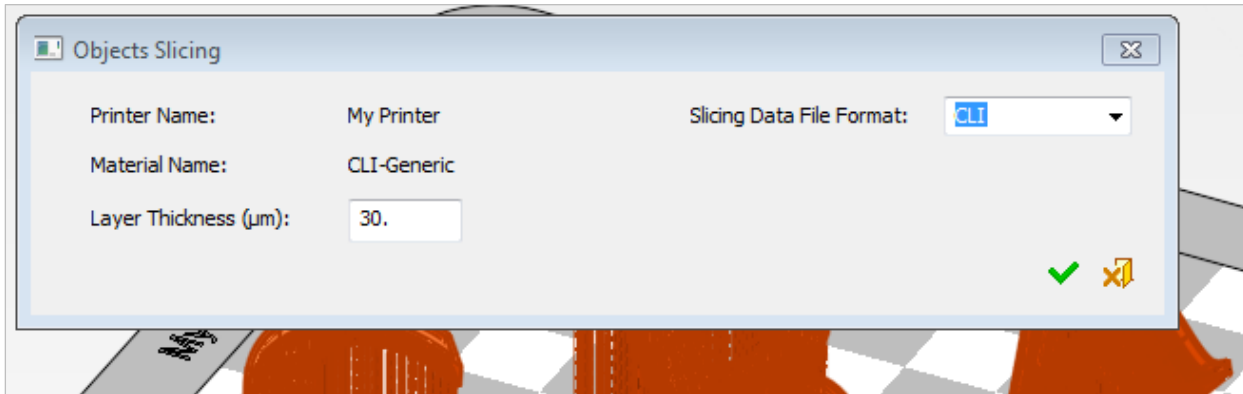
Slicing Data File Format – **CLI** - stands for **Common Layer Interface.**
This is an ASCII based universal format that describes layer by layer the **outer contour** of the sliced geometry and is often used to transfer slicing data to 3D printers.
CLI contours may represent the exact boundary of the model, or they can be at an offset from the boundary.
In each layer (or slice), the contours which lie exactly on the boundary of the model are referred to as C0 contours. The first offset inwards from that contour is referred to as C1. The next one is referred to as C2, and so on.
3DXpert for SOLIDWORKS creates C0 contours when slicing the parts and exports them to CLI using the **Send to Print** tool.
In 3DXpert for SOLIDWORKS Professional package, you can assign an offset to the C0 contours.

Layer Thickness (µm) – Define the layer thickness in microns – should be set according to printer capabilities or printing strategy.

*Pick* **OK** ☑ To approve and finish.

!	*Notice:*
	Slicing calculation may take some time, depending on the printed volume, number of layers and complexity of part(s) and supports.

A progress bar appear on the screen.
If you wish to interrupt the calculation, press the <Esc> key or click the 'X' button to close the calculation dialog.
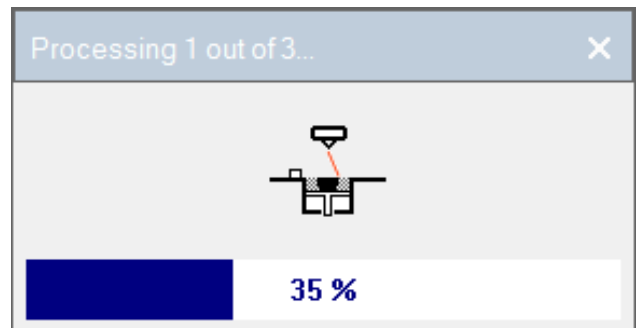
4. **Pick** the **Slice Viewer** [Slice Viewer] tool from the **3D Printing Process Guide** to view the slicing result.
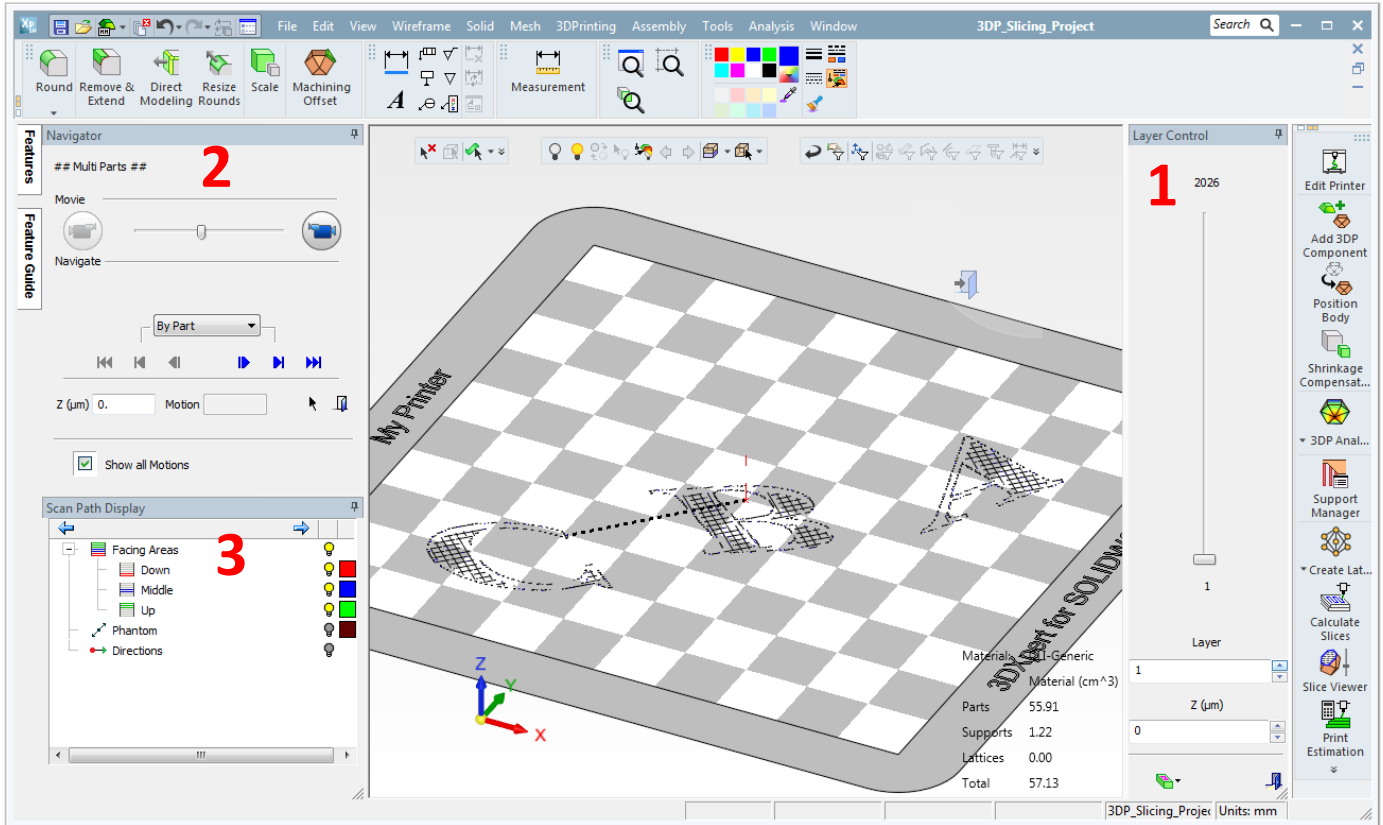
1 **Layer Control** - the **Layer Control** pane enables you to view the slices, slice by slice, by moving the slide bar on the dialog, by keying Layer number or by entering the Z height.

2 **Navigator** - the **Navigator** pane enables you to:

- **Navigate by Movie** along the motions in each layer. Control the movie speed using the Movie Speed Slider.
- See the scan-path and the order of the scan-path motions, per each layer.
- Jump back and forth according to the selected "By…" group of motion options, in the Navigator pane.

3 **Scan Path Display** - the **Scan Path Display** pane controls the visibility of the scan-path and scanning direction.

Look at layer 1, Z=0 µm. At this layer the support structures start.
(This type of supports follow a pattern shape).
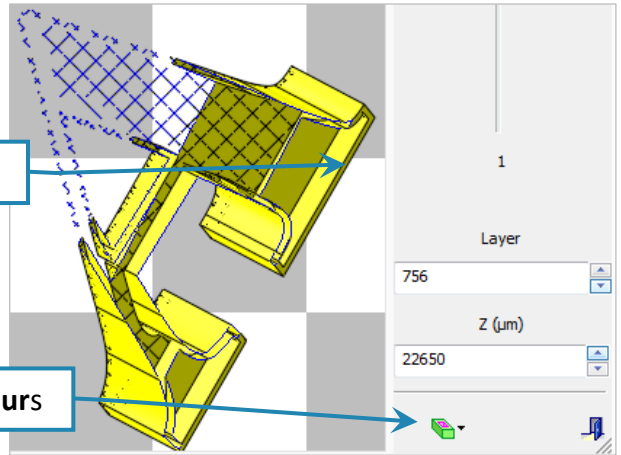In this case there are 2026 layers.

Supports Contours

Layer

1

Z (µm)

0
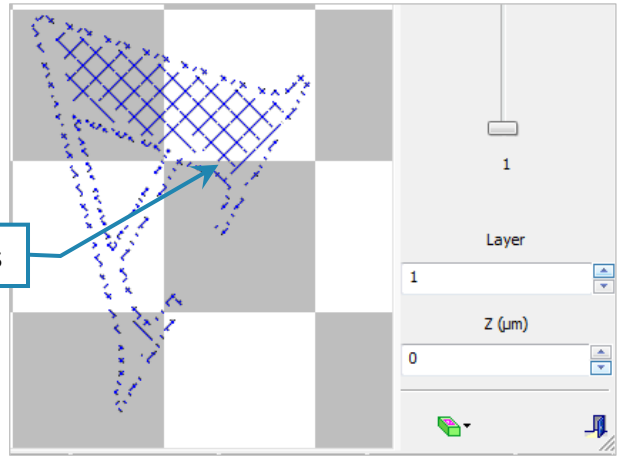
5. Move to layer 756, Z=22650 µm. It is possible to see the contours of the part at that layer and also the pattern of the supports.

Layer Contours

Layer Contours

Layer

756

Z (µm)

22650

6. From the bottom of the **Layer Control** change to different display modes .
    In Hidden mode it is possible to see only the layer, without the model itself.

Layer

1309

Original

Solid

Dimmed

Hidden

Z (µm)

9240
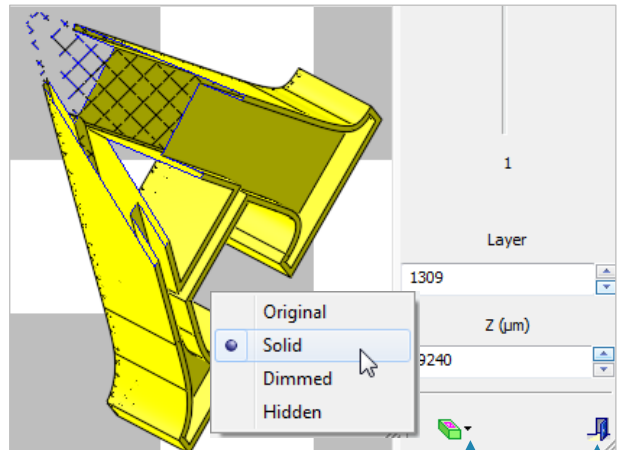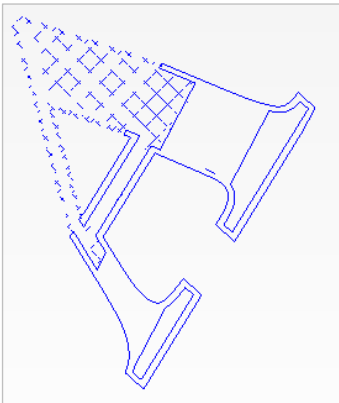
Display modes

Exit

To close use the exit icon .

After viewing and checking that the slicing is Ok we can now send it to print.

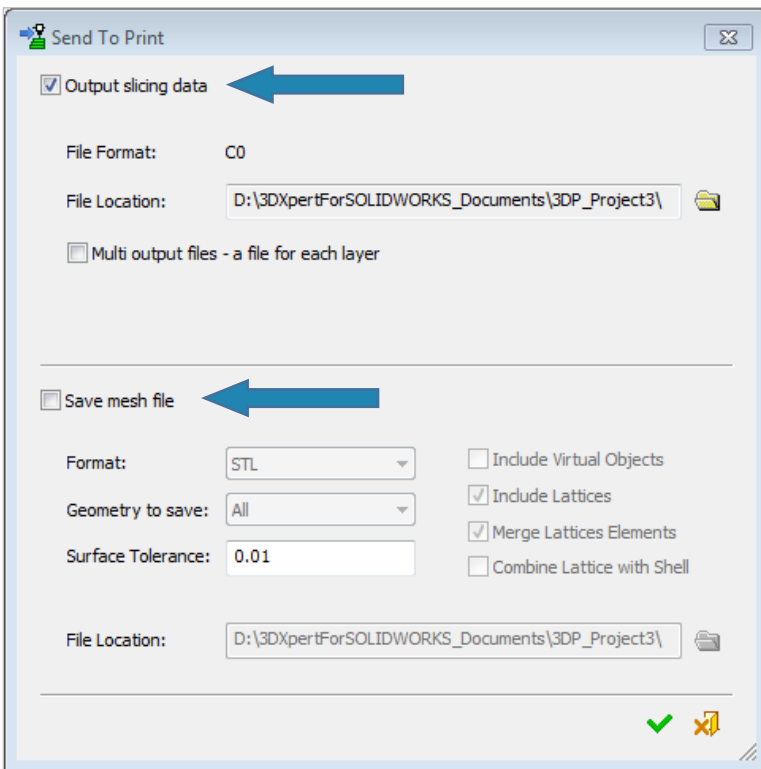7. **Pick Send to Print** command from the **3D Printing Process Guide.**

	Send to Print	
Feature Guide / Send To Print / Required / Optional	1) Select Objects (the default is **Multi 3DP Components**). 2) Set Parameters. The systems moves automatically from 1) to 2) in case of **Multi 3DP Components**.	
	✔	To approve and finish use the *"OK"*
	✔	To approve and continue use the *"Apply"*.
	✗	*"Cancel"* – exit the command without keep changes

On the screen appear **Object Slicing** Parameters Window:

Send To Print

☑ Output slicing data

File Format: C0

File Location: D:\3DXpertForSOLIDWORKS_Documents\3DP_Project3\

☐ Multi output files - a file for each layer

☐ Save mesh file

Format: STL

Geometry to save: All

Surface Tolerance: 0.01

☐ Include Virtual Objects
☑ Include Lattices
☑ Merge Lattices Elements
☐ Combine Lattice with Shell

File Location: D:\3DXpertForSOLIDWORKS_Documents\3DP_Project3\

There are two Send to Print options:

Output slicing data, insert destination folder path and if it is a **Single file** output file or a **Multi file** output file according to the printer.

Save Mesh file, four output formats are available: **STL, VRML, 3MF** and **OBJ**. Pick the required entities and destination folder path to save the file.

It is possible to do both at the same time.

   End of Exercise.

3D SYSTEMS