

TopSolid 2013 What's New

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TopSolid'Design 2013: What's New



This document describes the improvements made to the **design** application of the **TopSolid'Design** software: **2013** version.

Installation

Prerequisites

The installation procedure proposes to install or upgrade the components listed below with the following versions:

Component	Version
Microsoft .NET Framework	2.0
Microsoft .NET Framework	4.0
Visual Studio C++ Redistributable	2005
Visual Studio C++ Redistributable	2008 SP1
Visual Studio C++ Redistributable	2010
Sentinel driver	7.5.7
Sentinel RMS Licence Manager (floating license manager)	8.5.1

Assistance

In order to provide you a better quality of service and help you to quickly solve your problems, the new **Help** | **Remote assistance** function allows your favorite support team to take control of your computer using the **TeamViewer** software.

For those who are using a specific remote control system, a configuration keyword allows you to define the path to another application:

Syntax:

U_APPLI_REMOTE_ASSISTANCE_NAME <tab> application name

Configuration

The **Help | Configuration** function has been modified in order to display the configuration files of each application in separate tabs.

For each file, the path is displayed and if a file is not used because the application does not manage it in the *GROUP* folder for example, this information is displayed between comas at the end of the file name.

(Configuration			
	TopSolid'Design TopSolid'Draft TopSolid'Wood			
	TOPHOME : C:\Missler\V614 TOPLANG : EnglishUs TOPCONFIG : C:\Missler\Config614 TOPGROUP : C:\Missler\Group			
	GROUP CONFIGURATION FILES			
	top.cfg : file not found top.mat : C:\Missler\Group\top.mat			
	PERSONAL CONFIGURATION FILES			
	top.cfg : C:\Missler\Config614\top.cfg top.dgi : C:\Missler\Config614\top.dgi top.mat : C:\Missler\Config614\top.mat top.tex : C:\Missler\Config614\top.tex keyboard.cfg : C:\Missler\Config614\V614\keyboard.cfg			
	ОК			

The new configuration dialog box

Document

Print settings in the drawing

When a document contains several drawings, the print settings can now be defined in each drawing. First of all, the different settings (color, thickness, etc.) must be performed using the **File | Page setup** function, and then saved into a configuration file using the **Save cfg file** button.

Printer configuration	
Name: B&W.cfg	Browse Open cfg file Save cfg file
Device > Windows	•
Colors Line Paper Others	
I want the second se	and a second

Therefore, several configuration files can be saved according to how the drawings will be printed.

Then, when modifying the drawing, the new **Printer** tab allows you to choose the printer, the configuration file, the paper size and the document orientation.

1	Modification	
	DRAWING Printer Drawing	
	Printer > PDFCreator •	
	Configuration: B&W.cfg Browse	
	Paper> A3	
	Orientation	
	Portrait Eandscape	
	OK Cancel	

These settings can be performed in the template document; they will be used first during a simple or a multiple printing.

User interface

Dialog

In the dialog bar, when several options are available in a mode, a small symbol made up of 2 arrows is displayed at the end of the button.

🚯 😂 🔊 🛛 🕼 😂 🗳 🕼	😂 🍣 🕼 🐜 🗳 🖕	🗳 🔩 🔌 🤗 🕸 🕼 🔾 🖸 🔤 🛤	2
COORDINATE SYSTEM OR SKETCH DEFINE MODEL Co	oordinate system= CONSTRAINT 👻	Mode= DYNAMIC * Hook= NEAREST EDGES * Face to dril	II:

Example of dialog bar of the **Shape | Drilling** function.

Menus

The display of the alpha bar, the quick layer bar, the quick line style bar and the main menu is now managed as modes in the **Window** menu.

Image	Win	dow Help	8
Redraw Save configuration now Save configuration when leaving Reset configuration Close toolbars Select menu context		Redraw Save configuration now Save configuration when leaving Reset configuration Close toolbars Select menu context	
 ✓ Alpha bar Detached main menu ✓ Quick layers ✓ Quick line styles 		Alpha bar Detached main menu Quick layers Quick line styles	
	 Image: A start of the start of	Tile vertically Tile horizontally Cascade Design : Document1 * < <current>> (Associative mode)</current>	

Presentation

The new **Tools | Presentation** function allows you to save all the graphic data (orientation, rendering mode, zoom, multi-windows, graphical cross section, active layers, visualization mode of hidden elements) of a document in order to recall them later.

These presentations are automatically included in the new **Presentations** tab of the tree in which a contextual menu allows you to:

- add a presentation;
- rename a presentation;
- display a presentation (a presentation can be displayed by dragging and dropping it into the graphical area or by double-clicking on it);
- delete a presentation;
- redefine a presentation;
- start a slideshow from the selection of several presentations or all the presentations;
- calculate an image for each representation from the selection of several presentations or all the presentations (**TopSolid'Image** required).



Presentation in rendering mode with one view and one layer.



Presentation in rendering and wireframe mode with 3 views and all the layers.

Moreover, the presentations (without the graphical cross sections) are also available in **TopSolid'Viewer** if the **D_DOC_SAVE_ALL_VISUALIZATION** configuration keyword is set to 1.

Tree

The new Import indexes contextual command allows you to get the tabs created in another document.

If the imported tab contains a set which also exists in the current document, such as the set of lights, it is automatically reconnected.

If the imported tab contains a set which does not exist in the current document, but is created automatically by **TopSolid**, then this set is created and the tab is reconnected (e.g. drivers set, key points set, auxiliary elements set, publishings set, alternative sets are also covered).



Parameter

When creating or modifying a parameter, the new **PARAMETER** option now allows you to enter an expression in the minimum and/or maximum value of a parameter.

Bringing up the contextual menu on the **PARAMETERS** folder now allows you to:

- create a parameter;
- import parameters from another document;
- sort the parameters.



From a selection in the **PARAMETERS** folder of the Entities tree, you can now create a table and define parameters as drivers.



In the Edit list dialog box, the sorting order is now kept in the current session of TopSolid.

Edition

Repeat

Points, coordinate systems and texts can be excluded from a repetition.

Name

When naming an element from the tree, it is now possible to add a designation.

Curve

Contour with machining circle

For example, when it is required to manually draw the tool path inside a pocket, the **WITH MACHINING CIRCLE** option, available in the advanced options, allows you to display the tool bulk under the cursor.



Example of contour with machining circle.

Thickened curve

The **TRIM BY LINES** and **MITRE CUT** options have been added to the drop-down list for choosing the type of end. They enable you to trim a thickened curve either by lines, or by mitre cut.



In magenta, the curve to thicken and in red, the trimming curves.





In black, the thickened curve.



On the left, the curve to thicken and on the right, the selection of the 1st segment allows you to get 2 thickened curves (in black and red).

Smoothing

Now, the number of given point is associative; it can be modified using the **Modify element** function or from the tree.

Shape

Free constrained block

The new **Allow non-parallel faces** option allows you to create a constrained block on 4 planes whatever their orientations.



Example of constrained block creation (in blue) on a trapezoidal structure made up of profiles.

Deactivation of a group of operations

Now, a group of operations can be deactivated with a condition.

By this way, the deactivation condition is set to the group, the operations inherit this condition and all the new operations included in the group too.

When an operation is extracted from the group, its deactivation condition is kept.

Trimming

The dialog of the trimming by an imprint curve has been modified; now, the extension length is only asked when the trimming curve is open.

Coating

Now a coating operation can be modified from the tree using the **Modify** contextual function.

Cleaning

New options have been added to the **Shape | Manage | Clean geometry** function.

Clean shapes	23				
Complete clean	Complete clean				
🔽 Complete clean geome	try				
Linear tolerance	: 0.010mm				
Angular tolerance	: 1.000*				
Dptimize geometry					
✓ Simplify faces					
Linear tolerance : 0.	010mm				
✓ Optimize blends					
Advanced options					
Optimize tolerance edges					
Mepair edges	.0.010				
	: ju.u i umm				
Minimum length	0.010mm				
Remove small entities					
Fast clean					
Fast clean edges or faces					
I Faces					
[√] Edges					
ОК	Cancel				

The new options of the **Clean geometry** function.

The **Repair edges** option also allows you to make a repairing on edges of faces. You can specify the edge tolerance to follow and it is recommended to enter a tolerance greater or equal to the global cleaning tolerance.

The **Optimize blend** option allows you to find the "fillet" information on surface according to the Parasolid modeler definition. In order to get a lighter model, the optimization of the shape after simplification is recommended by Parasolid.

The Simplify faces option allows you to make a simplification in addition to the repairing.

The **Remove small edges** option allows you to delete edges whose length is less than the specified value. Then, the surface is rebuilt on the remaining edges.

The **Remove small entities** option allows you to delete the micro-surfaces which could be ignored by considering only the neighboring surfaces to design the shape. The deleted topologies are:

- small edges and small faces;
- peaks;
- thin faces;
- notches.

Simplification

New options have been added to the Shape | Manage | Simplify geometry command.

Simplify shapes		
Simplify geometry		
Simplify geometry		
Tolerance : 0.01000mm		
Optimize geometry		
Repair edges		
unear toierance : 0.010mm ▼ Optimize blends		
Advanced options		
Show simplified topologies		
Merge		
Simplify B-splines		
Approximation tolerance : 0.01000mm		
Angular tolerance : 5.000*		
V Keep degree of faces		
OK Cancel		

The new options of the Simplify geometry function.

The Keep tangencies option allows you to keep a continuity of tangency between simplified faces.

The **Repair edges** option is the same than the one available in the **Clean geometry** function.

The **Merge** option allows you to merge redundant geometries such as the hole face and the threading face of a tapping hole. In the previous versions, redundant geometries were automatically merged. This means that in a blind tapping hole, the threading face was merged with the hole face and was extended up to the bottom of the hole.



Shape to simplify



Result of the simplification with merge: the threading face is merged with the hole face.



Result of the simplification without merge: the threading faces are identical to the original shape.

Superposed surfaces

Now, the superposed surfaces can be deleted from the tree.

Checking in multi-core mode

The check geometry functions such as the **Shape | Manage | Check geometry** function have been optimized in order to automatically use the available processors. This allows you to save up to 30% of time.

Machining process checking

The new **Shape | Manage | Check machining processes** function allows you to display and modify all the machining processes defined in the operations of a part.

Two buttons are also available; they allow you to export the list into a text or an **Excel** document and create an associative table in the document.

Machining processes list				
	Table		Export	
Name	Designation	Machining process	Data	
s pocket		standard process		
🗧 🔤 pocket		standard process		
📗 🐉 drilling operatior	1	screw	6 x Tapping holes M 4x10	
🛛 🖏 drilling operation	1	screw	2 x Holes Ø9.8 Borings Ø10 [H7]x15	
🛛 🖏 drilling operation	ı	screw	Ø30 Counterbore Ø35x5	
🛛 🖉 drilling operatior	1	screw	4 x Holes Ø8	
Chamfer				
		ОК	Cancel	

Example of a machining process list for a part.

Moreover, the machining process list of a part can be displayed in a BOM column using the **MACHINING PROCESSES** function.

1	Modification		
	Title: PROCEDE		
	- Function		
	Defined functions >	MACHINING PROCESSES	-
		MACHINING PROCESSES	
	6	ELEMENT ID ELEMENT CLASS THICKNESS	

Example of a BOM template displaying machining processes.

Assembly

Deactivation

The deactivation of a part or a component with or without condition now can be done from a selection in the tree.

Main	Favorite	Main set Entities Layers	
🔰 💓 AS	SEMBLY	: (DETAILED) (3)	
⊡ .₩	💾 🧐	Delete	
	P1 8	Extract	
	🥩 🐝	Visible = YES	
	ŧ	Geometry independent of drivers = NO	
		Optimize tappings	
	ŧ	Move to another set	
		Activation / Deactivation	
	\rightarrow		

Alternative set

Now, in-place assemblies can be included into alternative sets as well as the main set.

In place assembly

The multi-selection icon 🗐 is now available when including a part in an in-place assembly.

Management of slave part modifications

In a document containing a slave part, when modifying the part's characteristics, only the **Cutting-up**, **Stock** and **Machining** tabs are available and in the **Cutting-up** tab, only the over dimensions can be modified. For example, when the over dimensions of a slave part are modified, the properties are forced. From the tree, the **Initial characteristics** contextual function allows you to get the characteristics of the master part.

Components

Profiles on curves

When including a profile component, the new **ON CURVES** option allows you to create the profiles on each linear segment of a contour or a sketch with or without mitre cut (mitre cut, covering/covered mode,...). The profiles thus created are grouped into an entity named **Multiple component process**, and you can then:

- modify one or all the profiles;
- add or remove contours;
- modify the cut type;
- modify the key point and the rotation angle.



Example of a wireframe structure on the left and the structure with profiles on the right; all the profiles are included at the same time thanks to the **ON CURVE** option.



The structure is completed with new curves (on the left) and new profiles are automatically added (on the right).

Distribution

The distribution allows you to distribute components between two stops by modifying the length of one instance automatically such as a rail guard between two walls.

It is implementing by the new **Assembly | Define component | Define distribution positioning** function which allows you to define the positioning coordinate system and the driver parameter to modify.

Then, during the inclusion, the new **DISTRIBUTION** option allows you to include the component between two planes either by automatically modifying the driver's value of the last instance, or by leaving an empty space.

The component whose dimensions have been modified can be positioned either at the end of the distribution (**Sequential** mode), or in the middle (**Alternated** mode).

The distribution can be modified globally (modification of parameters, replacement of component,...) or locally on one component only (sections).



Example of distribution positioning.

In red, the positioning coordinate system whose Z+ axis is oriented towards the distribution direction and the driver parameter named L which manages the rail guard length.



Example of distribution between two stakes; the latest instance is automatically modified in order to fill the empty space.



Example of distribution between two stakes; the midst instance is automatically modified in order to fill the empty space.

Driver propagation

A linear, linear constraint and circular propagation can now be defined as a driver, which allows you for example to shift one or several occurrences.

Once the propagation has been defined, an offset parameter is set on each occurrence and you can shift an occurrence along the propagation direction with a double click or with the **Shift instance** contextual function. An offset in the opposite direction can be done by entering a negative value.

When a propagation is defined as a driver in the template document of a component, the offset of the instances can be done in the assembly document using the **OFFSET INSTANCE** of the **Modify element** function.



Example of component, the linear propagation of the blue sub-component has been defined as driver.



In the assembly, on instance is not properly positioned.



Then, it is possible to shift one instance of the driver propagation

Driver block

The dialog for the inclusion of a component containing a driver block has been modified. The different modes are now displayed in a drop-down list and the **NEW CONTOUR** mode allows you to create the housing on the fly.

When the housing has been selected, the arrows allow you to choose a repositioning plane and define an offset according to the plane and/or a length.



Multi driver block

Moreover, when the component contains drivers, the multiple inclusion mode allows you to include several components in several housings with the same driver parameter values.

Excel catalog

When creating a catalog using the **Assembly | Components | Edit catalog header** function, the parameter designations are now exported as comments in the Excel cells.

	А	В	С	D 🛔
1	\$code	L	La Lo (Total L	ength)
2	1000x500	1000		
3	1200x500	1200		
4	1200x600	1200		
5	1500x600	1500	600	12
6	1500x800	1500	800	12
1.141	And the second second	and the second second		

When the document contains a big amount of parameters, the display of the designation will be helpful.

Boolean drivers

When a component contains Boolean drivers defined one after another in the driver set, the new **D_COMPO_ASKPARS_BOOLEANS_SIMULT** configuration keyword allows you to group them by 2 or 3 during the inclusion of the component.

```
Syntax:D_COMPO_ASKPARS_BOOLEANS_SIMULT <tab> nb (nb can be equal to 2 or 3)Example:D_COMPO_ASKPARS_BOOLEANS_SIMULT3
```

Result:

OK With extrusion profiles= YES 🗲 With drillings= YES 🗲 Centered positionning= YES 🗲

Modification of parameters

When modifying the parameters of a component, the **PARAMETERS** option offers two new modes which allow you to display all the drivers in a window similar to the **Parameter | Edit list** window and to modify them. The **CONFIGURE SINGLE** mode allows you to modify the selected component only, whereas the **CONFIGURE MULTIPLE** mode allows you to modify the parameters of several components at the same time. When different components are selected, only the drivers with the same name are displayed.

Tools

Axis-curve/Plane-face intersection point

The **Trimmed face= NO/YES** option has been added to the **AXIS-FACE** mode.

With basic geometry such as imported geometry, it allows you to quickly recreate the intersection point between the cylindrical face of a hole and its reference face.



In magenta: the drilling axis, in blue: the intersected face, in red; the intersection point (reference of the drilling).

Coordinate system on axis and point

In addition, the new **Coordinate system on axis and point** allows you to quickly recreate the coordinate system between the axis of the cylindrical face and its reference face.



With imported geometry (basic shape), the reference coordinate of a hole can be quickly created.

Coordinate systems

The relative and bisector coordinate systems are now available in the coordinate system icon bar. By this way, they can be created on the fly without ending the current function.

Attributes

Updating materials from textures

When the materials have been created from textures and then new textures are added (in a new or an existing family), these new textures are now taken into account when the function for creating materials from textures is run again.

Analysis

Collisions

When the collision analysis is performed using a multi-body shape, the message now displays the number of colliding bodies instead of the number of colliding shapes.

TopSolid'Draft 2013: What's New



This document describes the improvements made to the **draft** application of the **TopSolid'Draft** software: **2013** version.

Dimension/Detailing

Dimension

When modifying a dimension, the first dialog now allows you to modify the number of digits and display the trailing zero.

LOOK EXPORT SHIFT CONSTRAINT	Precision= 2 🔹 Trailing zero= NO 👻	New position:

Drill table

In the **Drill table** section of the **File | Properties** function, the new **Search hole if through boring** option allows you to distinguish the hole and the boring operations of a hole+boring drill in two separate lines.

Document properties	X
Document properties	Drill table
TopSolidWood properties	Drill indexes height : 6.000mm
General ⊕ ⊕ ⊕ General information - ♀ ♀ User information	✓ From top to bottom
Coordinate system Tolerance reference	Search depth faces for depth calculation
Geometrical tolerance ⊕ ↑ ↑ Dimension	Search hole if through tapping
- Pb Drafting view - Pb Projection parameters	Search hole if through boring
Table/bill of material	Allow operation transformation
Geometry/Miscellaneous	V Automatic update
Cartesian points	Entry points config Displayed text on view : Entry point

The setting of the option in the document properties.

KEY	INDEX	SYMBOL	TYPE	COORDINATES	ZDP	DIAMETER	TOL_DIAMETER
RK	1	Ų	Hole	-55.000 * -30.000	0.00	8.00	8.00
RK	2	Ų	Hole	-55.000 * 30.000	0.00	8.00	8.00
RK	3	Ų	Tapping	-22.000 * 0.000	0.00	4.00	4.00
-	-	-	Hole	-	0.00	3.30	3.30
RK	4	Å	Boring	-20.000 * 35.000	-5.00	10.00	10.00 H7
-	-	-	Hole	-	-5.00	9.80	9.80
RK	5	Ų	Tapping	-11.000 * -19.053	0.00	4.00	4.00

Result shown in a drill table: the hole+boring drill is displayed in two separate lines.

This setting is also available in the **Drill table** section of the **Tools | Options** function.

Moreover, the two new options **THROUGH DRILLS ONLY** and **NOT THROUGH DRILLS ONLY** allow you to create a drill table containing through holes or blind holes only.

Title block

Two new properties are now available:

- The **DAY DATE** allows you to include the printing date in a cell of a title block. This date uses the system date and is automatically updated when loading the document.
- The **LAST INDEX DATE** allows you to include the date of the last revision index and is automatically updated when a new revision is added.

Translators

SolidWorks Spatial

SolidWorks configurations are now supported during the import.

The translators have been updated in order to support the following formats:

Translator	Supported version
ΑμτοCAD	2012
Acis	R23
Inventor	2013
Catia V5	R6 – R22
	V5-6 R2012
Parasolid	V25.1
Pro/Engineer (Datakit)	2000i
	Creo 1.0
Pro/Engineer (Spatial)	16 – WildFire5
	Creo 2.0
Google Sketchup	8
SolidWorks (Spatial)	98 - 2013
Unigraphics (Datakit)	NX8
Unigraphics (Spatial)	NX8
-	
Export	2012
AutoCAD	2012
	R23
Catia VS	
Devecelid	
Parasolia	V25.1

TopSolid'Mold 2013: What's New



This document describes the improvements made to the **TopSolid'Mold** software: **2013** version.

Parting elements

Parting lines from surfaces

The new **Parting lines from surfaces** function allows you to assign some surfaces defined in another document (TopSolid'Design creation and/or surface import from an external CAD system) in the current document. Once done, TopSolid'Mold will automatically find the parting lines related to these geometries. Of course, it is possible to reuse the shrinkage factor applied to the part to be injected, defined during its insertion. The parting set management is also done automatically.



Internal extruded surface

2

It is now possible to define an **internal extruded parting surface** by selecting an edge path between a start edge and an end edge, as it is already possible for the external extruded parting surface.



Extruded direction

Now you can **extrude** a parting surface according a **TANGENT** direction, as it is possible in TopSolid'Design. This mode is available directly when creating the surface, in addition of the existing directions related to the current coordinate system.



Sewing tolerance

The **sewing tolerance**, used by TopSolid'Mold during the sewing process between parting shells and parting surfaces, is now available to the end user. You can set the default value in **Tools | Options | Blocks | Blocks sewing tolerance**. This setting can be edited and modified when creating the blocks, in the **advanced options** >>.

Movement

Guiding rail processes

After inserting guiding rails in TopSolid'Mold, it is now possible to define directly their process length. You can enter a value, or indicate on screen the expected length, as you do for the slides.

Stroke test

It is now possible to test the global stroke of a block assigned to a slide. This test is also available for a slide created on frame. In this way, the checking is more efficient and reliable.

TopSolid'Mold 2013: What's New

Cooling system

3D drilling colorization

You can now color the cooling drillings in the 3D design. When renaming the circuit, a new option allows you to color or not the created circuit. The default color can be set in **Tools | Options | Cooling and Runner | Regulation Property**. The color is applied directly on the operations; in this way, you can export your colored part, for example.

This color can be modified in the **Circuits Management** window. An option allows you to color drillings, during the computation of their visualization.

🖃 Cooling22 @430 : coolina @439 : cooling @446 : cooling @453 : cooling @460 : cooling 🛓 - Cooling23 @310 : cooling @319 : cooling @326 : cooling @333 : cooling @340 : cooling @347 : cooling @360 : cooling Check collision Check efficiency Visualisation Modification Color : • ٦ Color drills Representation : CYLINDER Transparency n 10 Show components 📃 Keep shape ΟK Cancel

Circuits management

Cooling circuit management in 2D

A management tool is now available when creating a 3D cooling system view. It is easier to select the cooling circuit(s) to project, with their associated parts, if you need them. You can find this improvement in TopSolid'Draft, from the Mold | Draw regulation | 3D Circuit function.



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Management

Drilling modification

It was possible to modify all the drillings of an element, if those drillings got the same type. Now, a new option has been added: you can filter the drillings according to the dimension of a reference drill. You can set this new way to modify in **Management | Modify Drills | Equivalent Diameter**.

TopSolid'Electrode 2013: What's New



This document describes the improvements made to the **TopSolid'Electrode** software: **2013** version.

Creation

Computation of the applied spark gap

Sometimes, the electrode shell offset failed due to an internal kernel error. It was possible to create the electrode, but without the applied spark gap. In the 2013 version, we have developed a new way to compute the offset in such a case. The spark gap, in this situation, will be computed on a solid shape that will be used to create your "real" electrode shape.

"3 in 1": Creation of 3 electrodes in 1 step

Until now, TopSolid'Electrode allowed two ways to create your electrode:

- Use for the file: It was the mode you used to create a "theoretical" electrode. The software managed three spark gap values used when dimensioning the electrode points of control. They were also useful in TopSolid'Cam and/or TopSolid'Draft.
- Apply to the part: It was the mode you used to create the "real" electrode, with one spark gap value applied.

When you needed to create the three "real" electrode shapes, with three different spark gaps values, you had to start three times the electrode creation function.

It is now still possible to manage the two previous modes, but they have been renamed for more clarity:

- Theoretical;
- Unitary.

A third mode has also been added: **3 values**. This mode allows, as a step by step wizard, to create the three electrodes, with three spark gap values, in a single operation:

 Spark gap parameters Theoretical Unitary 	🔘 3 values	
Default		■
Rough	: 0.25mm	-
Semi-finish	: 0.30mm	Color >
Finish	: 0.10mm	Layer: 0
	04	

Electrode creation on colorized faces

It is now possible to create electrodes on shapes which include colorized faces. When starting the function, TopSolid'Electrode analyzes the different colored areas (they can come from an import process or they can be colorized in TopSolid'Design using the function **Shape | Mechanical/Other operations | Color**) and allows you to create an electrode per area.



TopSolid'Electrode creates each electrode, as a step by step wizard. Of course, you can manage the parameters for each shape, according to the dimensions of the electrode shell.

Modification

Electrode shell update

It is now possible, by right-clicking on the shell in the design tree, to manage the update of this element. It allows you to add/remove faces from the initial shell, for example if the reference part has been updated, and if TopSolid'Electrode were not able to automatically update this shell.

TopSolid'Progress 2013: What's New



This document describes the improvements made to the **TopSolid'Progress** software: **2013** version.

General

Tool set checking

When a Progress document is loaded, a warning is now reported when some of the tool sets are missing ("punch set", "stripper set", "die set"...).

Most of the time, it happens when loading a TopSolid'Design document (*.top) as a Progress document (*.pgs). If this type of warning occurs, the tool sets must be rebuilt using the **Progress die | Tools | Rebuild tool sets** command.

Part preparation

Isolate bend: Isolate bend with forming

This new option is used to isolate the bends containing center or lateral formings, before using the **Unbend** for intermediate stage command.

It displays the preview of the planes that will be used to create the edges that will isolate the bend. The lateral extension length of the planes, as well as the extension radius of the imprint tools, can be adjusted with the **Overlength** and **Radius** parameters.



Preview of imprint tools and imprint results.



Result after an unbend for intermediate stage operation.
The **Divide** sub option allows you to divide the bend into several subparts in order to prepare a partial unbend.



The **Divide** option.

Unstamping: Analyze unstamping results

The Analyze unstamping results command (FTI) now allows you to view:

- The forming zones;
- The safety zones.

Displaying the **FORMING ZONES** enables you to view different areas on the part, according to the type of recorded strains depending on the combination and intensities of stretching and compression analyzed at each point.



Forming zone analysis sample

Displaying the **SAFETY ZONES** enables you to associate the analysis of the formings to the elastic limit criterions in order to view areas according to explicit feasibility criterions.



Safety zone analysis sample

However, this analysis has some limitations. It uses a constant thickness value for the whole part. So, it modelizes quite approximately the forming processes of parts made with several stages. The limit conditions of the forming process are ignored (influence of stripper plate, of holding stiffeners...).

The results are only available for **unstamping simulation** made with the module of our **FTI** partner.

Flange unfold



This new command is dedicated to unstamping/straightening of formings standing on the border of parts. It replaces user part geometries of an area by its unfolded geometry.

Two different modes are available:

EXTENSION: this mode extends the replaced area by a linear extension of a given length.
 The holes standing inside the replaced area or the geometries at the border of the part are lost. The user must recreate it if needed.

The option is made for simple cases.



Sample of result given with the Extension option

UNSTAMPING: this mode makes a "real" local unstamping of the area to replace. Unstamping is made on a leading surface that may be computed automatically by the function, or given by the user (then, it must be created before).



With automatic leading surface mode, the surface that will be used is previewed before the execution of the operation.

The automatic leading surface mode is efficient when the boundaries of the area to replace are simple, continuous and without too big concavities (the created surface must have no self-intersections).

It automates tasks that were previously done with the following commands:

- Cut part;
- Composite surface;
- Leading surface;
- Unstamping;
- Trim and thicken.

Of course, these commands are still available to treat difficult configurations.

This option is only available if you have the Forming Simulation FTI optional module.

Modify bend: Advanced management of overbending

In this command, several new options have been added.

- In the MODIFY FACE ONLY mode, it is now possible to compute the new bending angle according to the modified radius value, or to compute the new bending radius according to the modified angle value.
 These computation modes are similar to the computed modes of the Overbending on profile command.
- In the ROLL ON NEIGHBOUR FACES mode, the new option Manage continuity allows you to control how the additional geometries will be added, between the modified area and its connected areas.
 It the option is set to NO, the additional geometries will be linear.



Reduction of the bend length with a linear additional area.

If the option is set to **YES**, additional geometries will be cylindrical if the neighbor area is, and it will take the same axis and radius.



Reduction of the bend length with a cylindrical additional area, as the neighbor face.

Body management

This new command is dedicated to the management of parts for which we want to move some sub parts relatively to the others. This is typically the case of some connector parts that contain unbendable sub parts linked by some forming areas.

In **Invert strip** methodology, this command will be used to isolate the unbendable areas, to reposition it in the part, and then to unbend it step by step. The forming areas will be removed to be replaced with some transition shapes designed with the help of the connected sub parts.

Five different subfunctions are available:

- Divide;
- Move bodies;
- Remove bodies;
- Add body;
- Merge bodies.

The **DIVIDE** function allows you to divide the part into several subparts.

Several division methods are provided:

- By giving some points and a direction. The part will be divided along the planes perpendicular to the given direction and passing by the points.



- By giving some planar faces parallel to each other. The part will be divided by the planes defined by the faces.



- By giving a trimming profile.



The **REMOVE BODIES** function allows you to remove some of the bodies previously isolated, most of the time, in order to replace it by their equivalent unfolded shapes.

The bodies are selected by one of their faces. It is possible to remove several bodies in a raw.



The **MOVE BODIES** function allows you to change the position of some bodies to apply the movement produced by the unfolding of the whole part.



Example of body displacement that reaches the unfold position.

The **ADD BODY** function is used to add new bodies to the part.

In general, these bodies are some flat transition parts, designed by the user with the help of subparts of the shape.



Finally, the **MERGE BODY** function allows you to unite the bodies together before importing the flatten part in order to create the final strip.

Strip

Import part: Templates of theoretical unbend allowances

In the **Import part** command, in the unfolding parameters, when selecting the theoretical bend allowances computation method, it is now possible to choose a template of method among several predefined templates (see **Tools | Options | Sheet Metal – Unbending |Compute bend allowance as**).

Flattening parameters	×
Prevent update of element	
Bend allowances Formings treatm	nent
Use only global computation	
neutral complementary lengths theorical	Neutral bend line coefficient: @322=0.333 Complementary allowance: @324=1mm
din6335 neutral+complementary lengths+complementary theorical+complementary shifted internal draft	Neutral bend line coefficient: @325=0.5
	Templates: Default
Use tables	Press type> Neutral 🔹
Ve tool to use >	Choose v-tools by bend radius
OK	Cancel

This option is also available in the Sheet Metal | Generate flat pattern command.

Strip edition: Follow existing/subsequent operations: Redefinition of the template operation

In the Operations tree, when you edited a strip, it was already possible to manage the **Follow** existing/subsequent operations option on the initial definition of a station.

When using the **Follow existing operations** mode, it is now possible to redefine the operation used as template on the previous station using the **Update existing operation to follow** contextual command.

For instance, in the screenshot below, the station 3 is in **Follow existing operations** mode, based on element @231 of station 2.



The **Update existing operation to follow** command will redirect the template of the followed operation on the last operation targeted by the insertion operation cursor.

In the screenshot below, the result of the isolate bend operation @269 will be then applied to the station 3.



Free bending

In the **Free bending** command, it is now possible to apply the bending movement to another body of the part.

This configuration of use may occur when a strip is created with the **Multi bodies strip** option in a general process of invert strip design, and when wanting to change the orientation of a part in the strip (in general, when wanting to prepare a forming process).

Tools and dies

Cutting die: Dimensions of the second part by diameter

In the **Cutting die** command, when a circular profile is selected, it is possible to define the dimensions of the second part of the die with an explicit diameter value. It allows you to define rounded values, more easy to treat for machining.



Corner relief: Selection of edges to treat

In Symmetrical wire, Fillet and Chamfer modes, when the edges to operate are selected in loop mode, now it is possible to exclude some of the selected edges.

Tools

Duplicate for machining

The export format of the generated file can now be configured: ***.pgs** or ***.top**.

The new available format ***.top** shall be preferred to send a part for machining on a TopSolid'Cam station that does not have TopSolid'Progress, or to import the generated file in TopSolid 7 (that only read V6 file with the extension ***.top**).

TopSolid'Wood 2013: What's New



This document describes the improvements made to the **TopSolid'Wood** software: **2013** version.

Constrained block version 2

The new **Allow non-parallel face** option is now available in the **Constrained Block** function to help you create the part between non-parallel planes.

This option makes possible the creation of triangle and trapezoidal parts.

Example: Non-rectangle part



A red arrow will appear: it allows you to change the direction if the constrained block refers to the first face selected.



• Enter the third shift and select the third face.



Mode= LENGTH * Dimension= 200 Second plane or point

After entering the length, the red arrow will appear; you can click on it to invert the direction of this constrained block.

It is possible to select the points C and D to create the third and the fourth faces. The third and fourth faces created will be perpendicular to the bisectrix (E); this bisectrix is created between the faces A and B.



 Finally, you can place this part using the distribute mode or linear constraint, or just select one face and enter the offset distance.

OK Positioning shift= @217=50mm Click on arrow to invert direction





Example: Triangle part

The configuration principle is the same as the << Non rectangle >> example, except for the fourth face, you select a point.

• Select the face of the triangle part.

- Select the second face of the triangle using the Allow non-parallel faces = YES mode.
- Select the third face of the triangle.



• Select the intersection point of the face 1 and face 2.

• Select the face to give the position.





Driver block version 2

The **Driver Block** function has been enhanced with new operations in the 2013 version.

Adjust position

You can adjust one face of the driver block just by clicking the red arrow:

- You can input the shift distance to adjust the position: the faces can adjust from original face with the input value.
- You can input the length: the face you adjust will have a distance from the opposite face.



It is possible to input two different adjust distances on the opposite side, but you cannot input two lengths to control on part.



Reminder: Give preference to publishings

When inserting the component with driver block or when modifying it, it will always take the publishing face at first.

When inputting the component with driver block if you have the publication on the face, the driver Block will put automatically on the publication, if not, it will position the component with the first face fund.

<u>Warning</u>: In order to avoid cyclic references, you should create the automatic Publishing on an alternative set and not on main set.



Modifying the name and the designation of the Driver Block

Now we can give and modify the Driver Block with a different name and the designation. It allows having a name to explain the utilization of the parameter and the have a designation refer to the position method.

Name: V	Designation: Volume	
---------	---------------------	--

Multiple inclusion

DRIVERS SET : (1)
 V: Volume
 @31 : housing element
 V.x : X length of V = 600
 V.y : Y length of V = 500
 V.z : Z length of V = 860

Now it is possible to select several houses to insert the same Driver Block component just in one time. The other additional parameter inputted will be the same for all the insert components.

• To use this operation, insert your component, and select the first houses of Driver Block If you want to insert several Driver Blocks, select **Yes** to **Multiple inclusion**.

ОК) М	fultiple inclusion=	YES	Housing mode=	HOUSING	•
-------	---------------------	-----	---------------	---------	---

- Then, you can change the Housing mode for the next Driver Block in the rolling list **Housing Mode**, and then click on the arrow of the previous driver block to change the position face or give them the adjust distance and Length, or select the Box to position the next Driver Block.
- Once all the Driver Block are inserted, validate with **OK**, then you can input the common parameters for all the inserted Driver Blocks.



Double wrap

The double wrap is now available in the Driver Block component to manage the adjust distance of the thickness of the part to the destination Cabinet.

This type of the new Driver Block can be created by clicking on the double arrow >>: when you create the driver block

CURRENT COORDINATE S'	YSTEM 🕟 🛛 leng	th= 600	Y length= 500	Z length=	60
OK Double wrap= YES * Margin on X-= 19	Margin on X+= 19	Margin on Y-= 5	Margin on Y+= 0	Margin on Z-= 19	Margin on Z+= 19

Just like the Driver Block component, the input value for margin is just used to create the model; these margin values will be recalculated with the thickness of the part of the destination Cabinet where we insert the component.

Name	Nominal value	Designation
Housing.x	600mm	X length of Housing
Housing.y	500mm	Y length of Housing
Housing.z	860mm	Z length of Housing
Housing.x1	22mm	X- thickness of Housing
Housing.x2	22mm	X+ thickness of Housing
Housing.y1	0mm	Y- thickness of Housing
Housing.y2	0mm	Y+ thickness of Housing
Housing.z1	22mm	Z- thickness of Housing
Housing.z2	30mm	Z+ thickness of Housing

When inserting the Driver Block with the double wrap, the inside face of the double envelope can be repositioned by clicking on the yellow arrow inside. (Those red arrows are the outside face of the driver block).



Defining the default insert mode of the Driver Block

It is possible to define the default insert mode in the creation step of the Driver Block.



It is always possible to use another insertion mode than the default one when you insert your component.

Creating a new Block when inserting the Driver Block

When inserting the Driver block component, one new option allows you to create directly a block, where you can insert your component, once you click on the option new contour, the function **Contour** is started.

HOUSING INSIDE A BLOCK LIKE A COMPONENT NEW CONTOUR

Housing mode= HOUSING

It is possible to create a new contour. It is also possible to create the contour by passage Mode:

RECTANGULAR Pass mode= PASS ON SEGMENT 4 >> Start curve or point:



Once the contour is created, the function extrude will start automatically, you just need to input the value to give the height of block where you need to input the component.

Alignment= NORMAL ** Mode= HEIGHT	▼ Type= SOLID * DIRECTION	>> AUTO DIMENSION Height:
-----------------------------------	---------------------------	---------------------------

Hiding the insert box

From today, you can directly hide the insert box just by modifying the **Hide block** mode to **Yes**.

Housing mode= INSIDE A BLOCK	✓ Hide block= YES	Volume:	
------------------------------	-------------------	---------	--

Driver block and articulation

The new Driver block of the 2013 version is now compatible with articulation configuration. This improvement allows easily getting the Door and the Drawer with the articulation system.



You just need to right-click on the element don't move to make it fixed because the driver block cannot add automatically the constraint. For example as this door, you should right-click on the Hinge pivot and **make it fixed**.



@75 : 3 x Etageres @72 : Instance 1

± 11

🗄 🧊 @54

@73 : Instance 2
@74 : Instance 3

6

2

3

Analyze

Delete

Modify Name

Controls

Define driver

Propagation driver

Now it is possible to define the propagation as a driver.

This option allows you to modify locally an instance of the repetition, for example in order to avoid the collision of separation and hinge pivot of doors.

This new option is available for linear and circular and Wood propagation.

Creating the propagation driver in the component

- Edit the repetition in the construction tree, then in the repetition's propagation, you can define it as driver using a right-click.
- Define the propagation driver name.

Name of driving element:

Define the propagation driver designation

OK Designation of the driving element: Driver propagation

Using the propagation driver in the assembly



Example: A cabinet has a collision between the Hinge pivot and the separation.

- Modify the instance of the separation (**Modify** function, and then click on the separation).
 - In the dialog bar, select **OFFSET instance.**

OFFSET INSTANCE

Then give the offset value.

Offset= @1095=-50mm

<u>Note</u>: The positive value makes the offset in the propagation direction and the negative value makes the offset in the opposite direction of the propagation.

Of course, you can modify this propagation from the construction tree.





Distribution of the component

The new function of the distribution allows distributing the component between 2 elements. In this example, the separation between 2 walls will change automatically the length of one of the instance to adapt the total length.

In this example, the separation is inserted with the width 1000mm and the dimension of the last instance is changed automatically to adapt the total length.



To define this type of component, you need to use the **Assembly | Define component | Define distribution position** command.

The component distribution will be defined by two elements:

- One positioning coordinate system: The coordinate system must be linked to the variable dimension of the component. And the direction of the Z+ must be the variable direction of the component.
- You must have one parameter to change the dimension.

Then, when you include the component, the new **DISTRIBUTION** option allows you to adjust the dimension of the component between 2 planes.

OTHER POSITIONING DISTRIBUTION End point:



Once you have selected the **DISTRIBUTION** option, you have to select the coordinate system to start it. The selected coordinate system must have the same direction as the component. It means the direction Z+ should be in the direction Z+.

Reference coordinate system



Then, you need to select the arrived coordinate system; it is possible to select a coordinate system, a face or a point.





Once the second plan is selected, the distribution parameters will appear:



- **Distribution Mode**: **Mark out/Distribute.** In the mode **Mark out**, only the last instance of the component will change the dimension to adapt the distance. In the mode **Distribute**, the entire component will change the dimension to fill the total distance.
- **Numbering type: Sequential/ Alternative.** In the Mode **Sequential,** The modified dimension will be positioned at the end of the repetition. The Mode **Alternative** will modify the middle component.
- Fill out Mode: Last/Void. Allows deciding if the component modified is included (mode Last) or not (mode Voided).





OK Distribution mode= Mark out ** Numbering type= Alternated ** Fill out mode= Void **



When you modify the distribution:

OK COMPONENT DISTRIBUTION PARAMETERS RESET DISTRIBUTION TEMPLATE >>

- **OK:** Allows you to validate the modification.
- **Component:** Makes changes to the selected component. For example, you can interchange the component by another one.

Interchange: Once you have selected the **Interchange** function, you need to select the component to be inserted. After inserting the component, it is possible to modify the length value. By clicking **OK**, the length will be calculated by the distribution in the assembly. Using the **Measure** function, you can measure the distance in the assembly, or you can use the **Default value** button. The interchanged component will take the length value of the component.

```
OK MEASURE Longueur= @2767=1000mm DEFAULT VALUE=1500.000mm
```

Note: To interchange the distribution by another, the reference coordinate system and the parameter used

for distribution does not need to be defined in the same way. You just need to have the distribution in your assembly.

- **Distribution parameter**: Allows you to modify the parameter of the distribution.
- **Reset**: Resets the default value of the distribution.
- **Component:** Allows you to modify the component model. In the advanced options (>>), you can change the model of the entire component or only the selected component, and the component not yet changed.



Extruded component on profile

When inserting the extruded component, this new option allows you to directly create the profile, and especially the profile made up of several lines.

When inserting the extruded component from the **Wood | Other processes | Extruded component** command, this option is available using the **On curve** button after selecting the component to be placed.

X+	X-	Y+	Υ.	Z+	Z·	THROUGH POINT	CONSTRAINT POSITIONING	ON CURVES	First point or axis:	Γ
----	----	----	----	----	----	---------------	------------------------	-----------	----------------------	---

• Select the contour (or the contour with the multi-selection) on which you want to position the profile. It is possible to create the new contour with the **Profile** or **Sketch** modes by clicking the **On curves** button.

				_
NEW CONTOUR	=	PROFILE +	Curves	

• Once you have selected the curve/sketch, set the profiles options :

STOP Keypoint=P2 R	otation angle: @75=0*	Cutext= Cutext mitre	
--------------------	-----------------------	----------------------	--

- **Key Point**: Allows you to choose the key point of the profile, where you want to position the profile.
- Rotation angle: Allows you to define the rotation angle for the profile.
- **Cutext:** Allows you to select the automatic cut mode for the entire component.

Cutext=	None	-	
	None		ī
	Cutext mitre		l
	Cutext key covering/covered		l
	Cutext key first extruded covering		l
	Cutext key first extruded covered		L

- **None:** No cut will be applied to the extruded component.
- **Miter cut:** MITRE cut for all the profile.
- Cutext key covering/covered: Each component will have one side covered and one side no covered.
- **Cutext Key first extruded covering/covered:** The first profile component will have 2 covered or 2 covering components.





First profile covering

First profile covered

• Once all the parameters are set, validate by clicking the **STOP** button to include the profile.



Here all the plinths are inserted just on the new profile.

When modifying the profile :

ОК	COMPONENT	RESET	TEMPLATE	ADD/REMOVE CURVES	Key point= P1	Rotation angle: @88=0*	Cut= Mitre cut	▼ >>
Consector		CONTRACTOR DURING STORY	Concerning of a part of the second of		Contraction of Contraction	A REPORT OF A R		Concept

- **OK**: Allows you to validate the modifications.
- **Component**: Allows you to modify only the selected profile, and not all the profiles.
- **Reset**: Allows you to reset all the modifications to 0 (template, key point, rotation).
- **Template**: Allows you to change the template of the inserted component.
- Add/Remove curves: Allows you to add and remove the profiles in multiple mode.
- Key Points/Rotation angle/Cut: Allows you to modify the parameters on all the included profiles.
- Advanced Options >>: Allows you to choose whether the modifications of parameters, key points, rotation angles will be applied to the entire instance, or only on the instance which has not been changed yet (from the **Component** button).

Restrictive rule of automatic processes

This new option is available when creating/modifying the tools of your component and allows you to choose which part you want to cut when activating the processes.

Before, when activating the automatic processes, all parts that collided with the tools were cut.

Example: The tool creates two drillings in this component.





If this component is inserted in the assembly where you have two parts joint them together. The drilling of the case find the collision of two parts, it will automatically cut two parts will the drilling processes:



From now, in the assembly template, if a volume (Cylinder or extruded shape) is created in the place where we want create the drilling of the case, only the part that has the collision with the shape will be cut.

Creating the extruded shape:

The shape (shown in green) limits the drilling of the case on one part only.



TopSolid'Wood 2013: What's New

Link the limitation of drilling with the shape

- In the construction tree, right-click | Edit set, and then right-click on the tools | Define tool.
- In the dialogue bar, select **Define Clash shape**, and then select the extruded shape.

The extruded shape is link to the drilling operation now. The drilling operation is only works on the part that have the collision with the extruded shape.





User machining as component process

From now is possible to define a user machining as a component process. The user machining allows defining the tool path to use to create machining operation.

In this example of the hardware, the process is not groove or pocket.

So this user machining is defined in the hardware component with a special tool path.



Once the user machining is created you can define it as tools.

<u>Note</u>: The user machining is not a physical operation. It is necessary to input it with its identification. To do this, you should write the identification from **Tools | Options |Others| Display elements identifiers**

Symbolic tree Ø Display elements identifiers	
🔲 Display elements levels	
🔲 Display elements bill of material	informations
Elements echo from layer tree	
Precise	
🔘 Fast	

Define user machining with **Assemblage | Define component | Define tools.**

Like define normal tools, just give the name, when you need to select the local operation to insert, just write the identification of the user machining, you can get it from construction tree.

) @42 : part	
→2/2 2 /2	Local operation to insert in tool: @61
📲 @73 : hole = *	Educat operation to insert in tool, jean
🕀 @40 : extruded shape on curves	

The user machining now defined as the component tools when you use this component and use automatically process.

Wood unbent function

The new **Ben part** option allows unfolding the bend parts. The result of unfold part provides the unwound shape with all the characteristics of this part. These characteristics get the additional information (Operations, Dimension, Designation, Matter...).

This option is available when you define your parts. After you select the part for definition, just choose **Yes** for **Bend Part**.



<u>Note</u>: This option is already available in the last version, but it allows only calculating the rectangle dimension refer to the selected face.

After selecting the **OK** button, the unwound options will be shown.

Thickness=	@47=10mm	Neutral fiber coefficient= @48=0.5	Level= 0	•
22222222222222222			S135535435263	

- Thickness: Here you have to give the thickness.
- Neutral fiber coefficient: Allows managing the distance between the neutral fiber and the unfold face with the input value. The neutral fiber is the curves who have the same length on unfold flat part. With the coefficient 0.5, the neutral fiber will be placed on the middle of the parts, 1 will be the other side of the part, and 0 will be the selected face.

<u>ckness of the part</u>	<u>Selected face</u>	Neutral fiber with the coefficient 0.5
- Level: give the lev	vel where the unfold par	rt will be placed.

- **Follow the tangent faces Yes/No:** Allows selecting automatically the tangent faces of the unwind face when you use the option Bend.
- >> : allow opening the advanced configuration parameter:



- Isoperimetric number: Define the line quantity that will be placed on the Bend part to calculate the Unfolding. Higher this value is, better the quality will be but the calculation time will be long. The value 20 allows getting the good quality and a reasonable calculation time.
- **Tolerance:** The tolerance allows defining the calculation of the geometries precision of you part. Smaller of the tolerance is, more better the calculation you can get, but you need long time for calculation.
- **Drilling for reference face Yes/No:** Allows getting the drilling on the reference face or not.

Warning: The angle of the unfolding drilling will be the same as the angle between selected face and the drilling. So to get the drilling vertical to the unfolded face, you should have a drilling vertical to the reference face.

- **Select the faces to unwind:** Allows selecting the reference face to do the unfolding.
- Validate by clicking on **OK**.



- Then select the original coordinate system to do the unfolding. It is possible to select the axe that appeared on the point fund on the unfolding face. These axes are tangent to the selected face. It allows place the unwound part on the destination coordinate system.
- Choose the destination coordinate system for unfolding part.

TopSolid'Wood creates automatically the unwound part and opens the window of the definition of the part.



Once the part is unfolded; it is possible, by editing the construction tree to modify several parameters.

It is also possible to right-click on the **Unwind processes** | **Report coordinate system and points** to select some additional elements (Frame or points) on the original part to add them to the unfolded result. It allows adding the machining operation point or frame to the final unfolded result to do the machining.







Unfolded part has the drilling and the start/finish points allow doing the machining on the unfolded part.





The **Bent part** option should always be enabled in place at the last level of assembly. For components with bent parts, it is necessary to define there with the **Bent part** option on **No**. It is in the final assembly, we have to redefine the bent parts selecting by the detection and choosing the **Add bend sawing-up** option.

Roughing operation

Now it is possible to define the wood operation (Mounding, Groove and Rabbet) as the roughing operation.

This one allows defining the machining with two paths with two different tools, and be able to do two machining steps in TopSolid'WoodCam.

• Create the first Moulding machining with the Roughing tool and select the option **Roughing operation** in the configuration windows.



• Create the second Moulding for finishing operation.



MITRE CUT

Limitation of the ends of thicken profile

The two new options are available to the operation **Thicken** to limit his extremities refer to other profile, by the mode miter cut or planar cut. This option is very useful to manage the cutting between two parts created by trace thicken.

These	new operation are available in the rolling list End type when you thick	End type=	LINES	-	ľ
the pr	ofile.		LINES	1	i
-	Trim by line: Allows limiting two ends by selected lines.		CIRCLES OUTSIDE		
-	Mitre cut: The ends of the profile will be the middle angle line		CIRCLES INSIDE		l
	between reference line and the two profiles selected.		TRIM BY LINES		l

Once the reference profile selected, 3 options allow doing following modification:

- The option **AUTO** detected automatically the extremity line and limited Thicken line.
- The option **ALL SEGMENTS** detected automatically all the components of the contour and create all the thicken component on all the segments.
- The options **nothing** allows not limiting the profile on one side or on the other side.



This Extrude Block is a reception desk comes from the thicken line by miter cutting mode.

Improvements of part definition

When you define the part, the cutting-up frame is now linked to the selected axes. Use the associative frame allows getting the automatic debit update by the dimension and the position of the parts.

OK	Select axis automatically=	YES	Bent part=	YES
	A new way to use the	oart de	finition	

To continue to use the frame of cutting-up non-associative of the selected axe, you should use the configuration mode *D_PROP_ZWOO_SAWINGUP_ASK_ASSOC_FRAME* with the value 1, in the file *topzwoo.cfg* placed in the folder *Missler\Config\V614*. The function of part definition will be changed, when you define the part, it will ask you the frame is associative or not.

In the Cutting-up tab, when the part is defined with the mode Select the axes automatically :

- The mode **Respect the axes** is used by default with a frame associative link to part edge. In this mode, It is possible to inverse the axes by clicking on the arrow that placed on the part. When we click on the yellow arrow, we can turn over the frame with 90° (It is the same result when we click **Invert axes**).



 If we cannot find the cutting-up axes automatically, an error massage will be shown on the Alpha bar (Axes not found) and you must define axes manually (if the Add to cutting-up button is selected, otherwise, the axes will be selected when the part is added to cutting-up).

Define Define part.					
@55 : Axes not found.	X+ X-	Y+ Y) Z+ Z·	THROUGH POINT	Length axis:

- If the mode **Length always highest dimension** is used, the associative frame created will be the position with the X dimension in the longest dimension of your part.

In this mode, the button Invert axes is not usable and it is not possible to select the yellow arrow.

TopSolid'Wood 2013: What's New

Orientation Ørientation Ørientation
Part grain orientation Pat has grain orientation
Part has no grain orientation INVERT AXIS
Dimensions Respect axis
 Length always highest dimension

.....

- If we use the mode **Respect axes** and the axes of the part is not fund automatically, you will have an error massage in the alpha bar and the definition window is hidden to allow selecting the axes for definition.
- If all the modification of axe of debit, and the over dimension don't change between with and the length, it means over dimension on length is always on length, whatever about the axe direction.

<u>Note</u>: In the mode **Edge**, if the axe is invert, the over dimension will changed automatically to follow the width and length of the panel.

Г

Sizes	Values	Modes	Over dime
Length	400.0mm	edge shape	Omm
Width	196.0mm	edge shape	4mm
Thickness	19.0mm	additional	Omm
Sizes	Values	Modes	Over dime
Sizes Length	Values	Modes edge shape	Over dime 4mm
Sizes Length Width	Values 196.0mm 400.0mm	Modes edge shape edge shape	Over dime 4mm Omm

Improvement of the Add to cutting-up button:

- The situation of Cutting-up is link to each part: one part didn't add to cutting-up is always no cutting-up, until you select cutting-up.
- When you define the part not yet define, the Adding-up option will keep the situation that you used last time.
- When you define the part already defined in the old version with TopSolid'Wood 2013 :
 - If the part does not have information about debit, **Add cutting-up** will like a new part not yet defined.
 - o If the part has a frame of Cutting-up: the **Add Cutting-up** will be selected automatically.
 - If the part have the machining and draft property, but not the Cutting up information, the **Add Cutting-up** will not be selected.
- If the part definition is not added to cutting-up, the window of definition will open automatically, without the dialogue bar.
- When you define the part, if you select several parts and click on stop, all the parts will be defined automatically with the associative Frame in mode **Respect axes**, and they will be added to cutting-up.

Improvements of Define multiple parts

From today, if the selected part of multiple definitions is not yet added to cutting-up, they will be added to cutting-up automatically, in the mode **Respect axes**.

If the axe is not fund automatically, the message will be shown in the Alpha bar and part will be grey in the multiple definition windows, and it is not possible to modify the part no added cutting-up with multiple definition.

For avoid the error axe of cutting-up in multiple definition, it is not possible to invert the axe direction and use the mode **Respect axes** or **Length always highest dimension** in the **Define multiple parts**.

Define the part on the repeated parts:

- If all the assembly is selected, the BOM will show all the parts de repetition: the modification is local for each part.
- If the repetition is selected by clicking on the parts. The BOM will consider the template of the repetition: the modification is global for all the parts of repetition. The modification will not rewrite the information of each part, if the part is modified locally.

Configuration of machining workstation

Now it is possible to predefine the machining working station in **Tools | Options.** You can define one list of working-station for multi machining. This rolling listed is available when you define the part in the machining Tap.

<u>Note</u>: The 2 default working station in old version is proposed in the rolling list to have the compatibility with old version files.

Name	Designation	Work station 1	Work station 2
Bottom-Left	Bottom-Left	×	
Bottom-Right	Bottom-Right	-	×
Top-Right	Top-Right	-	-
Top-Left	Top-Left		-


Other improvements of panel entity

When creating the panel

When you create or copy the panel, all the edge will be positioned refer to the coordinate system and the first axe selected in **Tools | Options | Configuration TopSolid'Wood | Edge/Laminate**. All the edges are oriented on the part like the codification in the BOM.

Coordinate system		
Sawingup coordinate system	Machining coordinate system	Draft coordinate system
Naming axis order		
Axis 1 : ∑· ▼ Axis 2 : ∑+ ▼	· Axis 3 : X- • Axis 4 : X+ •) Axis 5 : Z+

The first edge will be always on the axe 1 refer to the selected coordinate system on the part. (Cutting-up coordinate, machining or Draft).

dges	s and	laminates cre	ation wizard for panel			23
Edge	es					
	N*	Codification	Edge type - code	Length	Beginning cut type	End cut type
	1	6644	Flat edge chamfer - ep 2	432.5mm	Mitre cut	Mitre cut
	2	6644	Flat edge chamfer - ep 2	190.0mm	Mitre cut	Mitre cut
	3	6644	Flat edge chamfer - ep 2	432.5mm	Mitre cut	Mitre cut
	4	6644	Flat edge chamfer - ep 2	190.0mm	Mitre cut	Mitre cut
Ec	dges	are similar				
Va	iuts an	e similar				
	>>					
Lami	ninate	shapes	- Concentration	18580-0258	1.22	1
	N°	Codification	n Material	Coating	, Th	ickness
	1		Laminate		@	69=1mm
	2		Laminate		@	70=1mm
🔽 La	amina	ites are similar			ADD C	DIFICATIONS
				C 1		
				Lancei		
				N.	and the second s	583A

Here the edges are positioned refer to cutting-up coordinate system, 1st edge on Y-.

Then, the other edges are positioned in the rotation order of Z+ from the selected frame, like the yellow arrow in this example.



In the edges creation windows, we can select or unselect the laminate for add or delete them.

	N°	Codification	Material	Coating	Thickness
~	1	Codif-X	Balsa	TMaterial%02_Blue	@200=2mm
	2	Codif-X	Balsa	TMaterial%02 Blue	@201=2mm

When copying the panel

Attribute and property of the panels

In the document TopSolid'Wood, if the option copy panel is used to recreate the panel entity from some panel already existed, the edges, the laminate, and the options of the panel already existed will be copied. No property will be copied (designation, material, type...).

If the panel is copied from the **panel template**, all the properties and attributes will be copied on the new panel.

<u>Note</u>: The material and the coating will not be copied from the **panel Template.** To copy the material and the coating of the panel template, you have to copy the following configuration words ZWOO_D_PROCESS_PANEL_COPY_MATTER_COATING with the value 1.

Defining the part on the panel not yet defined

When you copy the panel on some part not yet defined, TopSolid'Wood will link some default cutting-up property to part (on associative frame) without open the part definition window. We can change this mode by put some configuration value ZWOO_D_PROCESS_PANEL_COPY_OPEN_DEFPART with value 1:

- If the config value is 1, **Define part** window will be opened.
- If the config value is 0, **Define part** window will not be opened.

Then, the cutting-up frame will be shown on the part and one red arrow on the first edge.

Here is possible to modify the frame by clicking on the arrow. If the edges are positioned refer to the frame, I will be changed automatically after changing the frame.



The red arrow allows changing the direction of the edge.

Slave panel management

With the version 6.14, you can manage slave panel entity. The method to do this is the same as the slave part from just one part.

• In the construction tree, **right-click on the entity panel | Slave part**.





- Normal update of the original part:
 - Normal update: The slave part will be updated automatically.
 - Warning when modified: The slave part will be updated with warning to user.
 - Forbid any modification: The original part cannot be modified.
- Forbid any modification for the slave part: Forbid or not the modification of slave part.
- **Name of the file**: Allows you to change the name of the file of the part.

Improvements of the end cut types of edges

For the panel function, the limitation of the end cut types of edges will be better in some special case:

Result in version 2012	Result in version 2013

Edge codification improvements

Several improvements in this version are be done for edges and laminates management.

A new option All edges and laminates must have one codification is available in Tools | Options | TopSolid'Wood Configuration | Edge/Laminate | Codification.

If this one is selected, the temporary edge and laminate cannot be valid; they must be linked to some codification.



So, if this option is selected, when you create the panel if there is no codification linked to the Edge and the melamine surface you cannot validate it.

For example, in this Panel configuration window, the laminate is not linked to some codification. And the OK button is grey, you cannot use it.

The **Add Codification** button is useful to directly create the codification of laminate.

oating : TM hickness : 1	aterial%01_Whit
IN KDESS	DO DO
LM-MEL-BL	1
	ADD

N°	Codification	Material	Coating	Thickness
☑ 1		Laminate	TMaterial%01_White	@74=1mm
2	2 Laminate		TMaterial%01_White @75=1m	
ADD COE	DIFICATIONS tes are similar			

	N*	Codification	Material	Coating	Thickness
~	1	LM-MEL-BL1	Laminate	TMaterial%01_White	@74=1mm
	2	LM-MEL-BL1	Laminate	TMaterial%01_White	@75=1mm
٩DD	CODI	FICATIONS			
ADD	CODI minate	FICATIONS			

Once you created the laminate codification. It is possible to click on OK to validate the panel configuration window. The codification that you created is added to the TopSolid'Wood library.

This operation now is available in the function: Edge, Laminate, Panel, Edge configuration, Laminate configuration.

Now, in order to avoid use some material no prepared for the edge and Laminate, the edge and the laminate is no more available in the material configuration function.

Edge configuration: In the **Edge to apply**, the codification of the edge is shown.

Codification > [CH-0.8mm-P	VC U-blanc	 EDGE
Standard : TOPW00D Type : Thin edge Variant : Flat edge Varsion : 01 Code : 0.8 Material : Pvc u Coating : Mat white paint		

Improvement of the Edge codification automatic generation

In **Tools | Options | TopSolid'Wood Configuration | Edge / Laminate | Automatic codification of Edge,** it is possible to generate automatically the codification of the edge by some rules. This codification can be added to the Edge codification list by clicking on the Button **Copy codifications in the Edge configuration.**

With this option, if this codification is already created, an error massage will be shown and it is not possible to copy the Edge. From now, the button **Replace codification in the Edge configuration** allows you to automatically replace the existing codifications.

This function is very useful in case you have new materials to use and/or the codification rules have changed.

ted codific	ations				
	Codification	Edge type - code	Edge matter and coa	Over dimens	Calibration overvalua
E	CHANTSV2-0.8mm-pvc u-mat wh	flat edge - 0.8	pvc u - mat white paint	Omm	0.8mm
5	CHANTSV2-0.5mm-pvc u-mat wh	flat edge - ep 0.5	pvc u - mat white paint	Omm	0.5mm
E	CHANTSV2-1mm-pvc u-mat whit	flat edge - ep 1	pvc u - mat white paint	Omm	1mm
	CHANTSV2-2mm-pvc u-mat whit	flat edge - ep 2	pvc u - mat white paint	Omm	2mm

Improvements of multi draft

In the version 2013, several points of Multi Draft are modified in TopSolid'Wood.

Free scale factor

Now it is possible to select if the scale factor of the multi Draft is free or link to the factor in the list. You can make this one in the **document property or the application property**.

Draft	
Free scale factor	
Scale factor from list	

When you use the multi Draft function, the function will automatically calculate the dimension of the 2D view to adapt the dimension of the Draft document:

- If the scale factor is free, it will find the correct dimension for draft.
- If the scale factor is from list, TopSolid'Wood will automatically select automatically the nearest smaller value than the automatic one in the list.

Machining View in the template

Now is possible to put the machining view in the template. You need to use **Define part | Machining | First position/Second position.** To position the view you need to use the function **WOOD | Multi Draft |Create template | Create view | First Machining/Second Machining.**





Scale factor of 2D view: When positioning the 2D view, we propose you to use the factor 1. In fact, when you use the multiple drafts, the software wills calculate automatically the best scale factor to position the view. If the factor is different than 1, it will multiple them with the best factor.

Example: If the factor 0.1 is used here, and the function Multi-Drat find the best factor is 0.5, so the result of factor of 2 D view will be 0.05.

Various Improvements of use

- The dimensions of the part are created by composite dimension.



- The drilling dimension is now supported on the draft.



- The dimension of wood operation: the note created on the wood operation is now composed by 2 lines; the first line has the name of operation and second line has the machining tools information.



- The edge dimensions are shown in the direction of the Edges and will be on the center of them.



To manager better the view and avoid overcharge of the draft, the automatic dimension will be only shown on the principal view of the draft document.

If you want to use the old way show all the edge dimension, you can use the configuration *ZWOO_DFT_NEW_EDGE_NOTE* with the value 0.

Improvement of the 3D index

Options setting

In **Tools | Option | Predefine index**, the type list is now shown either by **Predefined part types** or **Types of document elements**, or both with **All types**. The types of parts you don't want to number must be selected.

Available types	Element type	Prefix	Start value	*
 Predefined part types Types of document elements All types 	 edge shape Achetée Assembly facade Cabinet Caisse Caisson case minifix case pin smooth pin steel pin wood pin minifix Fabriquée 			F
	<			1

This list allows you to select the types of parts for which we do not want to predefine the index. We can prefix the number for each type of part and give them the start value for several types.

Start value for automatic index

For the automatic index, it is now possible to define the start value of the first level.

Numbering mode= AUTOMATIC * Multi-level mode= YES * Start value= D

- If the automatic index starts with a letter, a letter is required.

Start value= D	🗩 ASSEMBLY : (MIXED) (4)
	📴 💼 < > G - Glasse shelf kit
	👜 💼 < > F - Simple cabinet Cabinet A
	👜 💼 < > E - Simple cabinet Cabinet B
	🗄 💼 < > <mark>D</mark> - Simple cabinet Cabinet C

- If the automatic index starts with a number, a number is required.

Start value=	4

🗩 ASSEMBLY : (MIXED) (4)
📴 💼 < > 7 - Glasse shelf kit
🗄 💼 < > 6 - Simple cabinet Cabinet A
🗄 💼 < > 5 - Simple cabinet Cabinet B
🗄 🛅 < > <mark>4</mark> - Simple cabinet Cabinet C

Second numbering for parts

A second type of numbering, the second numbering, is available with the automatic numbering of elements.

OK Numbering type= Second numbering 🖘

Possibility of using the principal numbering or second numbering for the Automatic index. The two types of automatic numbering work the same way as in previous versions.

Example to use the two type of numbering

- **Principle numbering**: Allows you for example to provide the same index for identical geometries to manage only one program for several parts.



- **Second numbering**: Allows you for example to create the index in the multi-level mode in order to identify the parts in the different subassemblies of the project (see examples shown in yellow below).



Ardis Interface

The Ardis interface has been improved to allow you to export the machining information to Ardis. The interface of previous versions with the format R41 is always available, but you can get a new format XML (you can manage it in **Tools | Option | TopSolid'Wood Configuration | Cutting off |Ardis Configuration**) that have the machining information for each part. The following machining operations are supported:

- Calibration;
- Drilling/multi-drilling;
- Groove/Rabbet;
- Pocket;
- Moulding;
- User machining

All setting options are created in **Tools | Option**. For more information about the Ardis XML interface, refer to the *Ardis XML Interface* training document.

<u>Warning</u>: In order to use this export, some special additional tools are required in Ardis. For more details, refer to the *Ardis XML Interface* training document.

Example: TopSolid'Wood exports one part with the Groove, drilling and pocket information.







↑ TopSolid'Wood Project

Part in TopSolid'Wood

Part in Ardis

Function	×	Y	Length	Width	Radius	Z2	Side	Operation
DRILL	490.50000	50.00000			2.50000	8.50000	Back face	
DRILL	490.50000	80.00000			4.00000	10.00000	Back face	
DRILL	490.50000	320.00000	1		4.00000	10.00000	Back face	
LINE	0.00000	388.00000	500.00000	388.00000		11.00000	Back face	Groove
ARC	450.00000	200.00000	374.69796	317.27472	139.17864	19.00000	Back face	Pocket internal
ARC	374.69796	317.27472	103.52833	302.13806	250.46515	19.00000	Back face	Pocket internal
ARC	103.52833	302.13806	54.39299	168.74070	124.76142	19.00000	Back face	Pocket internal

↑ Ardis information of part exported from TopSolid'Wood.

Document presentation

The new function in **Tools | Option | Presentation** allows you to save the presentation of a Design, Draft or Machining document.

These presentation save all the graphic information (orientation, rendering mode, zoom, multi-windows, graphic cut, act if level, hidden element) of your document.



The presentations thus created are automatically included in the new **Presentations** tab of the symbolic tree in which a contextual menu allows you to:

- Add the presentation
- Rename the presentation
- Show the presentation (a presentation can be displayed just by dragging it into the graphical area).
- Delete the presentation
- Redefine the presentation
- Calculate the rendering for each presentation (if you have TopSolid'Image). This new option allows you to calculate different images for different presentations saved. These images will be renamed the same way as the presentation.

Main Favorite Main set Entities Layers Presentations PRESENTATIONS Build Top Assembly Inter 1 Inter 1 Inter 2 Inter 3	h=950 500 500 150° = 1600 = 800 h=950 150° 90° = 1600 =
	= 1600 =

Presentation of the top view, with the profile and the dimension.



Presentation with perspective view, with separation level hidden.

Improvement of the background image

The **background image** is now available for assembly components.

You just have to insert the background image in the component. If the image is linked with the component dimension, it will follow the transformation of the component in the assembly (position and dimension). This improvement is very useful for decoration items.



Configuration management

The function **Help | Configuration** has been improved to give you the indication of configuration management.

The applications are separated in different tabs and you can have more information about the local and group configuration setting.

Top Solid Design	Top Solid Fold	Top Solid Wood	Top Solid Planner		
GROUP CONFIG	URATION FILES	6			
topzwoo.	cfa : file not four	nd			
	FIGURATION F	ILES			
I ENSONAL CON					
topzwoo.	crg : L:\Missler\	ConfigV614\V614	Ntopzwoo.cfg		

TopSolid'Cam 2013: What's New



This document describes the improvements made to the **machining** application of **TopSolid'Cam software**: 2013 version.

2D Milling

In 2D milling operations, it is now possible to use the clearance block to manage retracts (at the end or between passes) between drilling, pocketing and contouring operations.

Feature recognition

Pockets and slots can be recognized, even if they have no bottom face, provided they have been created using the pocket and slot TopSolid'Design feature.



Contouring

- It is now possible to add a fillet on right angles.
- Many improvements have been made for WoodCam use:
 - Customizable vertical plunge;
 - Approach at the beginning of the first pass and retract at the end of the last pass when there are many passes in Z;
 - ...



Pocketing

- When a pocket is cut in many portions (because there are some islands), it is now possible to stay on the pocket Z level and thus avoid jumps.
- TopSolid feature pockets based on multi-profiles are supported. The feature is shown as N machinable items.

nit ar	halysis to	a piece of part	None	-	Dis	play area
Pocl	ket feat	ure				
	Indix	Poc	Process	Number	WES	Top mach. nam
П	#10	groove	Sharp groove (Ro + Cont)	0/1	WCS part 6	
	#1.9	Pocketing	Pocket form	979	WCS part 6	
ы						
ы						
ы						
•				111		- F
			WCS par	t 6	-	
Op	eration si	mulation	Full manual	- Check	for useless operations	Automatic
		Ma	chine Exit	Configuration	Bec	to analyze



Drilling

- A new setting allows another security distance to be defined to secure the jumps over the obstacles (clearance distance on collisions).
- Machine cycle for threading in WFL machines has been added.
- Collision checking is available when the user drills the pocket entry points. The problem occurred when the centering tool was bigger than the drilling tool.
- The Management of the intersecting holes has been completely rewritten. Nonetheless, the former version is still available via the **Option** menu.
- The ¾ drilling supports pecking and clearing options, even in case of intersecting holes.

3D Milling

Roughing

The roughing calculation time in **pocket** mode has been improved. On complex parts, a factor 2 has been noticed.

Contouring by plunge

■ In contouring by plunge, the **Corrected** and **Original** modes are supported.

Finishing

- PPT between two curves has been improved by the addition of synchronization parameters.
- New sweeping possibility to be able to machine admission pipes. Available in 5-axis also.



■ All the limitation curves can be deleted in one click.

Super-finishing

- Possibility to use the curves limitation bounding mode (on, before, after).
- Collision checking with the tool holder has been added.

4D Milling

Parameters, like angular split values, starting C values which were defined with configuration words, are now available in the dialog boxes.

5D Milling

Swarf machining

- Better management of areas where the tool cannot go when its diameter is too big.
- Helical approach and retract have been added (also available in 5-axis contouring). Be careful, there is no collision control on these approaches and retracts.
- On specific cases, it is possible to lock the Y axis to ZERO. Be careful, in these cases, we are not sure to produce the right part because the normal vectors are modified! Conversion of a translation movement into a rotation.
- Approach and retract can be sliced according to the initial and final tangents of the curves to machine.
- Retract modes between passes and at the beginning and the end of the operation have been improved (also available for 5-axis contouring).

3D to 5D

A new tilting mode keeps the tool vertically and tilts it to avoid collisions when they appear, and only when they appear.

5 axis transformation	
11: BALL NOSE MILL (ER. HEML012020B, SA50), D-12, L-26	
Chew teel helder	
Show too holder	
Axis and angular limitations Retracts Cutting conditions	
Main Projection and collision Collision tool definition	
Parameters	
Stock to leave > 0mm	
I olerance > U.Ubmm	
Side and e	
Dist.Max. between pts>10mm	
Max. length for a link > 29.8496mm	
Lead in method > Tool axis	
Lead out method > Tool axis	
V Lock angles	
Angle limitation	
Computing method > Keep vertical auto-tilting if collision	
Select Dounding euroes : 0	
Select main direction : (0, 0, 1)	
Tool tail diameter > 12mm	
Look aheas radius > 30mm	
Max. tilt angle > 45°	
Min. tilt angle > 0*	
smoorning ratio (u-40): 1	
The second	
Limitation of 5 axis tool path inside stock Activate	
Axial clearance distance > 2mm	
Radial clearance distance > 6mm	
Link movement> Automatic 🔹 🗐 💇	
Comment: 👻 🛃	
PP : 🔻	
OK Cancel Batch computing	

Basic operation sets

Each operation can be used with a no generate attribute.

Retract to the tool change point

New parameters set in which coordinate system are the coordinates input by the user. This is useful to save and apply retracts to another file. "ctp_type_coord"

Turning

Parting off

A new parameter sets the feed rate of the retraction to the retraction diameter. In the previous version, it was only possible to do that in rapid.

Driven point

• Possibility to drive a point which is not the point of the tool insert that machines the part.



Finishing operations

Spinning operations are supported. In this case, the tool must be defined as a special milling tool.



Trochoidal groove machining is supported. <u>http://www.youtube.com/watch?v=HRtSpY0SdBY</u>

Methods

- Possibility to automatically break the associativity between the method and the operations created by the method.
- Possibility to create a method by selecting more than one operation.
- Edition of an interactive method by using the same values which were input when the process has been run the first time.
- Comments and PP words of operations created by methods are now recalculated like the other parameters.
- Some parameters such as the machine-tool used or the material to cut have been added.
- Three new parameters have been added to manage holes in methods (available in **Edition | Options**). These parameters allow the hole intersections in method formulas to be managed.
- A new parameter allows the creation of one centering operation (and only one) even if more than one cylinder are drilled and this on all the method operations.
- A new operation called **form process execution** allows the call of another form method.
- Method loading time has been notably improved.
- A new lock option maintains the operations consecutive. This option is not available for form hole methods.
- New way to define local variables inside a method.

Tools

■ Management of the length extension links.



■ Management of the clamping system for the minimal tool length calculation.

Operation manager

- In the Operations List tab, it is now possible to see and edit (by double-clicking) the link movements and the tool change movements.
- The name of the ISO file can be seen and edited in the **Operations List** tab of the operation manager.

Verify

■ Integration of MachineWorks 7.1 (verification of 4&5-axis operations in turbo mode).

Miscellaneous

Simulation

■ The simulation of Andrea head has been improved.



A parameter allows you to run or not the simulation after each operation creation.

Symmetry management

A new parameter configures the way used to calculate symmetrical tool paths in contouring. In some cases, it is useful to get the exact symmetry of the original tool path. In these cases, the approach of the symmetrical operation is identical to the retract of the original operation.

User interface

The mouse scroll is inhibited in the combo-boxes of TopSolid'Cam boxes to avoid changes of settings when the user makes a graphical zoom.

Default values

- In Edition | Change default values | Base | Configuration | Parameters, 9 integer, angle, length and string user parameters have been added. The goal is to customize machining methods easily. These parameters are usable as key words in method editing.
- .CadParameters.Name allows you to use Cad Parameters in default values of the Cam application. Name must be replaced by the name of the parameter.

Saving

A new option allows the purge of intermediate stocks in order to reduce the size of the files when the user saves them.

TopSolid'WoodCam 2013: What's New



User interface

Update of menus

To provide continuous improvement of the user interface, the menus have been reorganized and updated.

Configuration of TopSolid'WoodCam

The **Help | Configuration** function has been enhanced to show more information such as the tool database used, the default value file, etc.

All operations are not up to date

When operations are not up-to-date and we use a function which may be affected by this state, the warning message has now a **Cancel** button to stop the function.



Various translations

Translations of some functions have been revised.

Machine templates

New standard machine templates

The standard machines have been replaced by generic machines illustrating the various possibilities of TopSolid'WoodCam.



Retractable stops

Stops can be defined to be retractable. A Boolean parameter can be created and used in the machining document to indicate that stops are out or not. This parameter will be automatically modified by TopSolid'WoodCam when positioning parts.

The parameter must be named *stops_activated* and its values can be:

- **0**: stops in;
- **1**: stops out.



Definition of stop origins

The window for defining origins has been revised to simplify its use.

Positioning origins : Position 1				
Part main origin				
Top				
- Eace choice				
● Top ─ Front ─ Rear ─ Right ─ Left				
THE TRUTTE INT				
Face origin choice				
💿 Bottom left 🔘 Bottom right 🔘 Top right 🔘 Top left				
Others positionings				
Display positioning origins > Position 1				
Apply origins to others positionings				
OK Cancel				

Rotating drilling ramps

Drilling ramps can be defined with a C axis and automatically recognized for operation analysis.

Tool management

Creation of special mills

The special mill creation can be done directly from the CAD tool library of **TopSolid'Wood**. In the tool magazine, use the **Automatic moulding tool creation** button, and then select the tools you want to create from the list.

All tools are created in **Normal** mode.

Tools units : TopSo	olid'WoodCam 3X Flat Table <routing tp1="" unit=""></routing>	
Units	Tool changer Prepared tools	Tool choice Saving Information
A ^	11	Tools origin
	T2	💿 Database 💿 Prepared 👝
a de la companya de l	T3	- Family choice
	T4	
	16	Types choice
A Dreen	T8	SPECIAL MILL
	T9	
	T10	Internal reference Tool off D L Material
	T11	
	T12	6
	T13	<u>.</u>
	T14	
	115	
-		
	OK Cancel	

Special tools creation		
Standard: TopSolid'Wood Family.type,variant: Profilage et contre profilage Type 1 Type 10 Family.type 2 Type 2 Type 3 Type 4 Type 6 Type 7 Type 8 Type 8 Type 9 Simple tool Simple arc Simple doucine Simple doucine Simple groove Simple groove		Created tools in D:\TopSolid\TopSolidFolder\V614\Config\Database\Draw\ PROFILAGE ET CONTRE PROFILAGE_TYPE 1 PROFILAGE ET CONTRE PROFILAGE_TYPE 1_TOOL2 PROFILAGE ET CONTRE PROFILAGE_TYPE 2 PROFILAGE ET CONTRE PROFILAGE_TYPE 2_TOOL3 PROFILAGE ET CONTRE PROFILAGE_TYPE 2_TOOL3 PROFILAGE ET CONTRE PROFILAGE_TYPE 3 PROFILAGE ET CONTRE PROFILAGE_TYPE 3_TOOL2 PROFILAGE ET CONTRE PROFILAGE_TYPE 3_TOOL3 PROFILAGE ET CONTRE PROFILAGE_TYPE 3_TOOL3
Existing tools Replace	🔘 Don't replace	Ask the question
		ок

Corrector number

You can allow the definition of corrector number to several tools in your magazine.

Document properties	
Document properties	Tools magazine
Document properties TopSolidWood properties Units General General information Service information TopSolidWood properties User information Visualisation options Visualisation options TopCorinate system Decementical tolerance Generatical tolerance	Display corrector tab
	C Display program tab
	Allow same correctors numbers
	OK Cancel

Aggregates

Tool numbers of aggregates

Tool numbers of aggregates used on the main spindle can be modified after their definition. This modification can be done using the **Equipments | Aggregates | Aggregates properties** command.

Aggregates properties		
Aggregates list		
Name	Machine reference	Tool position
My aggregate mounted on TP1		No position 👻
		No position
	OK Cancel	T3

Approach and retraction macros for aggregates

Link movements for approach and retraction are now automatically defined on the aggregate, when defining it, according to its type.

Part positioning

Part positioning

To ensure positioning and repositioning of parts in different cases, the positioning functions have been improved. The enclosing shape of the part is used to avoid positioning problems depending on the part complexity.

Automatic WCS creation has been improved as well.

Choosing the stop to use

If you defined the stops to use with the default working stations of TopSolid previous versions, you have to redefine these options using the new function.

In TopSolid'Wood, you can define working stations in addition to **Left working station** and **Right working station**. To create the link with TopSolid'WoodCam, you have to define the stop to be used for each working station using the **Misc. | Machines | Misc. modifications** command, and then the **Working stations** tab.

For multi-machining, the definition of working stations is important because they are used for part positioning.

lisc machine def	initions		- • ×
Working stations	Post-processor	Options Information	
Working station	5		
Use of station	\$		
Machine stops		Working stations	
Position 1			-
Position 2			
Position 3		Right working station	
Position 4		Back left	
		Front left	
		Back right	
		Front right	

Repositioning of "anyone" set

It is now possible to reposition an "anyone" set without switching back to one origin per part. The **On the stock** or **Global** mode is kept.

Operations

Calibrating

Starting point of calibrating operation

Calibrating the different types of parts (single, mounted set, anyone set and nesting) can now be customized for each of them.

In process associations, a different association can be done for each type of part.

A new parameter **Beginning element choice for closed curve** is available in calibrating and cutting processes to manage the starting point:

- **0**: Start in one of the angles;
- **1**: Start on the longest element;
- **2**: Start on the closest element to the center of the panel (nesting only).

Calibrating			
📄 🚊 Parameters 🔹 🔺	I		
🗊 💼 🛅 Plunge	I		
🖶 📲 Retracts	H		
E-Comments	H		
🖶 💼 Loop breaking edges	Н		
- Se Automatic attack	H		
- 🤐 Link type			
🛛 🥯 Links number			
🚽 🥯 Links height			
- Se No links			
🖳 🥯 Link slope type			
Se Link slope angle			
- Se Links cutting			
Maximum angle according to fiber direction	H		
Begining element choice for closed curve	H		
2 : Basic m operation			
Begining element choice for closed curve :			
U in an angle 1 bigest element			
Keywords Apply Reduce tree Show modified Sort parameters			
Default value = 0			
Variable name = .ComplexForm.1.first_element_type			
Do not sort: respect operations order from the process.			
OK Cancel Operations			

Links

Slope type

A new type of slope is available to create a slope up and down to avoid shocks to the machine.

Links	- • •
Create links Link type >	Repartition 👻
Link height :	5mm
Link length :	10mm
Link step :	Omm
Links number : Slope	2
Slope type >	double slope 🔹 🔻
Slope length :	Omm
Slope angle :	45°
	Select points
	Delete links
📃 Cut links	
Create	Jinka Concel



Groove, rabbet and moulding

Roughing operation

When analyzing operations on the part, only operations which still modify faces of the part are analyzed and machined.

In some cases, operations are needed to make roughing operations on the part before the final operations. In the following example, the first operation is a rabbet which removes material before the final moulding operation. The rabbet operation is totally overlapped by the moulding.



It is now possible to define a CAD operation as a roughing operation. This allows you to indicate to TopSolid'WoodCam to machine this operation, even if no more face of the operation is on the part.

This option is available in the creation window for wood operations.

Parameters		×
Tool type Saws	Routers	
Standard: TopSolid'Wood	_	Machining process>
Family,type,variant:		Operation is modifiable in assembly
Blades Bindes Double tool	l l	Roughing operation
i flat band □ Mill		Make one operation
84 Adjustable groove mill 84 Downtail mill ≡	and a second	at at the set was the

When analyzing operations, if the option is unchecked, the rabbet is not analyzed because it is hidden by the moulding.

N	Machinable	faces list pièce usinage 5/	2			
l	imit analysis	to a piece of part > None		•	📃 Displa	y area
ļ	Drill feature	Form wood milling Sugge	ested tools			
	Indix	Wood milling type	Process	Number of p	WCS	Top mach. name
	🖬 × #1	Calibrating	Calibrating	0/1	Posage pièce	
	⊡ * #2	Moulding	Moulding	0/1	Posage pièce	
					T	
	Operation	simulation	Automatic	▼ Check	for useless operations :	Automatic 🔹
		Machine	Exit	Configuration	Redo	analyze

If the option is checked, the rabbet is analyzed and machined.

Machinable fac	ces list pièce usinage 52 a piece of part>None	!	•	Display	area
Drill feature	Wood milling Vood willing type	Process	Number of p	WCS	Top mach. name
₩ * #1 ₩ * #2 ₩ * #3	Calibrating Moulding Rabbet	Calibrating Moulding Rabbet	0/1 0/1	Posage pièce Posage pièce Posage pièce	
🗌 Operation sin	mulation /	Automatic	Check Configuration	<pre></pre>	Automatic •

TopSolid'WoodCam cannot machine propagated roughing operations. Only the first operation will be machined.

User machining

User machining on component

You can define a user machining as a tool of components. This type of operation is now machined by TopSolid'WoodCam.

Groove and rabbet using a blade

Blade choice for wood operations

Choosing the blade for groove and rabbet machining can be set in order to select different kinds of spindles at first.

This configuration can be done using the **Misc. | Machines | Misc. modifications** function, and then the **Options** tab.

These options are:

- **No selected option**: The tool orientation is done based on the CAD operation.
- **Use the nearest frame**: The nearest WCS for the operation is used first to search an available blade.
- **Use the sawing unit first**: If the operation can be done with the sawing unit, it will be used first.

Misc machine definitions				
Working stations Post-processor Options Information				
Frames				
Align positionings frames				
Part positioning				
Create lateral frames				
Create frame under the part (if kinematics allow it)				
Create local frames for saw grooves and rabbets				
📝 Adapt frame				
Saw machining				
Use the nearest frame				
🔽 Use sawing unit first				
OK Cancel				

Operation with a blade on inclined plane

A new option is available to create local WCSs for wood operations performed with a blade (groove and rabbet), when the WCS is different from the main WCS, even if an existing WCS is available on the needed orientation.

This option allows you to perform inclined operations with blade on some machine types with specific security constraints.

Misc machine definitions			
Working stations Post-processor Options Information			
Frames			
Align positionings frames			
Part positioning			
✓ Create lateral frames			
Create frame under the part (if kinematics allow it)			
Create local frames for saw grooves and rabbets			
☑ Adapt frame			
Saw machining			
Use the nearest frame			
☑ Use sawing unit first			
OK Cancel			



Sawing

Sawing limitation

A new option for the sawing operation allows you to limit the length of the sawing according to the selected element (face, line, etc.).

This option is available for manual sawing by selecting the limitation type or by using processes.

1 🔽 🏲 🖬 . 💌 👒 . I	💊 🗠 🛝 📄 🐨 🖉) 🕶 🗢 º🖕 🐂 🗠		
🛍 🚬 🖀 🖬 🚵	👪 🚣 🛷 😨 🛡 🕶 🌞 🜌	🛓 - 💌 🍓 🧰 🎩	🕘 🗰 🕅	🔤 • 🗛 • 🖡
MANUAL OPERATION -	Sawing direction on a line/an edge by 2 points In	nvert tool NO 🖘 Sawing limitation	то тне ѕтоск 🛛 🚽	Select a planar face
·····			TO THE PART	
		-	TO THE STOCK	
			TO THE ELEMENT	

Three options are available:

To the part: The sawing path is extended to finish part limits.



- **To the stock**: The sawing path is extended to stock limits.



- **To the element**: The sawing path is trimmed to the dimensions of the selected element.



Driven point choice of blade according to material side

TopSolid'WoodCam can automatically select the driven point of a blade for sawing operations in order to set it on the material side.

We will have the bottom driven point when we will use the front face of the blade, and the top driven point when we will use the back face of the blade.

Document properties		
Document properties		
TopSolidWood properties	Sawing	
📣 General		
🗄 🐌 General information	Tool choice	
🛛 💆 User information		
Rendering options		
Visualisation options		
🐵 🗽 Coordinate system		
- 🗜 Tolerance reference		
i L Geometrical tolerance	Approach type	> Perpendicular 👻
N Sketch		
Constraint		
T lext/note		
Lompass		
Houting configuration	Retract type	> Perpendicular
At Condex and features analysis		
Cylinders and reactiles analysis		
Database		
Visualization		
Holes		[a
Drilings	Sawing clearance	: jumm
- Routing		
Sawing		
	Bottom part offset	- 10mm
	bottom part officer	
	Affect saw driven point on the material side	
L		
	OK Cancel	

Operation using a blade

Operation using a blade in the main WCS

An option is available to generate the canned path in the PDB for wood operations using a blade in the main WCS.

Misc machine definitions
Working stations Post-processor Options Information
Configuration Options
Drills ramp
Generate all the holes in the ISO code
4 continous routing axis
Generate the profil in the ISO code
Rails and pods
Generate only used set positions in ISO code
Saw machining (groove, rabbet and sawing)
Generate canned path if machining is in the main frame
OK Cancel

Tool path

The new parameter **Wood canned path in the PDB** available in the default values and the processes allows you to generate or not the canned path in the PDB.

Groove
☐ 1: Rough grooving
Write canned path in PDB : 0 don't write canned path 1 write canned path +
Keywords Apply Reduce tree Show modified Sort parameters Default value = 1 Variable name = .ComplexForm.1.wood_canned_path_in_pdb
Sort normally.

Pocket

Multi-contour pockets

Automatic machining of pockets done from multiple contours (text, ...) is now possible.



Pockets cut by another operation can also be machined as one pocket and not only as open pockets.

Result with version 6.13:



Result with version 6.14:



Common

Starting point on closed contour for wood operations

Until now, the starting point of wood operations was at the beginning or at the middle of the first element of the closed path.

Now you can choose the starting point by indicating a value between **0** (beginning) and **1** (end) ($0 \le X \le 1$). The value **0.5** will be a starting point at the middle of the element.

The **Start parameter of first entity** parameter is available in the default values and the processes.



Moulding with values set at 0.25 and 0.75:



Starting point of wood operations defined in CAD

The definition of the origin point of a wood operation on a closed path will be automatically used for machining. It is possible to test in the processes whether this point is defined or not in CAD.




Multi-pass tool path

Approach and retraction can be done only at the beginning and at the end of the path and not at each pass. Activation and deactivation of the cutter compensation can be done in the same way.

Contouring
4: SLOT MILL (FR-2TA-16) D:16 L:80 Corr.:4 (Tool holder : TP1)
WCS solution> TP1 Z-
Main Cutting conditions Approach/Retract Parameters Info.
Approach and retract at the beginning and at the end only Tool compensation activation/desactivation at the beginning and at the end
Lead out same as lead in
Regular je d'in
Comment:

Slope plunge

The beginning of the slope plunge is now performed from the Z level of the security distance if the operation depth value is set to 0mm.

Plunge				×	
	Inside material Outside material				
	Feed rate				
197	Rapid				
A	Slope				
	Helical				
Operation of	jepth > 0mn	n			
Inside material plunge parameters Plunge angle inside					
Helicoidal plunge angle (out material)> 3*					
	ОК		Cancel		

Slope plunge with operation depth at 0mm:

* *	

Slope plunge with operation depth at 5mm:



Breaking edges

A new option for contouring operations allows you to break edges with a radius. Parameters are the same as those for breaking edges with a chamfer.

1 : Calibration	- • •
1: SLOT MILL (FR-2TA-16) D:16 L:80 Corr.:1 (Too	ol holder : TP1)
WCS solution > TP1 Z-	- <u>*</u>
Main Cutting conditions Approach/Retract Parameters	Info.
Radiusing according to the angle	
Limit angle (between 0 and 180) > 180*	
Add loops/breaking edge	
None Coop Breaking edges	Parameters
Starting Z	and the second

ĺ	Breaking angles parameters		— ×
	Limit angle (between 0 and 180)	>	180*
	Breaking edges method	>	Radiusing 💌
	Breaking edge length	>	2mm
	Distance before breaking edge	>	25mm
	Distance after breaking edge	>	5mm
	Feed rate reduce % before breaking edge [0,100]	>	50
	Feed rate reduce % along breaking edge [0,100]	>	50
	Feed rate reduce % after breaking edge [0,100]	>	50
	OK Cancel]	

Result in purple of the breaking edges:



Movement to the clearance shape

We do not go back to the clearance shape by default anymore if we are already outside of it.

A Boolean keyword defines this behavior: **ZMI_WOOD_MACHINE_GO_ON_SECU_SHAPE**

- **0**: TopSolid'WoodCam does not move.
- **1**: TopSolid'WoodCam moves back to the clearance shape.

If the keyword is missing, the default value is set to **0**.

Retraction to the clearance shape

A new retraction type has been added to make a retraction to the clearance shape within the different routing and drilling operations.

Final height		×	4 : Drilling	
			4: FLAT DRILL (F0-FPL-10) D:10 L:50 Corr.:4 (Tool holder : TP1)	
			WCS solution > TP1 Z- 0 (0) 0 (0) 🔹 🛃	
	blana		Customization Cutting conditions Information	
	None	0	Main Geometry Holes intersection	
<u> </u>			Depth parameters	
			Depth : 10 Security distance : 2mm	
	Incremental value	0	Z min = 10mm	
			Depth to > Useful -	
			Gauge diameter : 10mm	
	Absolute value		Tip angle (hole) > 180*	
	Absolute value	0	Through hole	
			Extended depth > 1mm	
1000			Dwell : Us	
	Z Stock	0	Starting Z	
			Parameters	
			Retraction between points > Retract to clearance plane	
	Z finish		Obstacle height > 0mm	
	2 11 11 11	~	Clearance plane > 22mm	
			Clearance distance on collisions> 2mm	
			Lead in direction > XY_Z	
Cle	arance shape defined into the	part 🔍 🛛		
			Accurate drilling : Disactivated	
			Update stock	
Value > 0mm			Link movement > Defined in tool holder 🔹 🗸 🗸 🗸	
Value / John	Comment:			
	OK Cancel		PP :	
			OK Cancel	

Machining properties

The machining properties defined in CAD on operations can be used within processes using the keyword **Machining property** available in the different operations or directly in the PDB with the keyword **CAD_COMM**.



Retraction link movement to tool change point in 5-axis

The retraction link movement to tool change point in 5-axis (5XTool) has been improved to avoid useless movements.

Retraction link movement for spindles without tool changer

Retraction link movements are now used for spindles without tool changer when operations using them are not consecutive.

Operation analysis

5-axis continuous user machining

5-axis continuous user machining can be machined within operation analysis.

This new type of operation does not use any process; therefore the configuration of the CAD operation and the configuration of default values in CAM are very important to ensure a correct result.





Available tool types for operations

Default available tool types for TopSolid'WoodCam have been revised and updated according to their use.

e choice for opera	ation			
ool choice Informat	tions			
Tool types list SLOT MILL BALL NOSE MILL CHAMFER MILL			·	
REVERSE CHAMF DISC MILL 3T GROOVING TO SPECIAL MILL	ER MILL OL		=	
Used tools list			📝 Filter	
Reference	Tool type	Tool chang	Material	
🔀 FR-2TA-16	Slot mill	1/TP1	Fast steel (HS)	
🔀 FR-2TA-10	Slot mill	2/TP1	Fast steel (HS)	
🔀 FR-2TA-5	Slot mill	3/TP1	Fast steel (HS)	
•			Þ	
	Use	ed tool		
			Cancel	

Operation analysis for mounted set

Operation analysis of sets has been harmonized so that the analysis is the same for all types of set. You can now make an overall analysis of the whole set or perform analysis part by part as before.

Set analyse and machining					
Operations processing Visualisation					
Verticals operations only					
Choose operations to machine					
Calibrating	Counter-mouldings				
Apertures	Grooves				
User calibrations	Rabbets				
Slope faces user machinings	Pockets				
Continuous 5 axis user machinings	Holes				
Mouldings	Aligned holes				
Analysis					
Do all parts analysi	s again				
Machining of all the parts of a part					
Optimization					
Optimize paths					
OK //					

Additional operation types for set and nesting machining

Slope faces and continuous 5-axis user machinings have been dissociated from standard user machining.

Set analyse and machining						
Operations processing Visualisation						
Verticals operations only						
Choose operations to machine						
Calibrating	Counter-mouldings					
Apertures	Grooves					
User calibrations	Rabbets					
Slope faces user machinings	Pockets					
🔽 Continuous 5 axis user machinings	Holes					
Mouldings	Aligned holes					
Analysis						
of a part						
Do all parts analys	is again					
Machining of all the parts	of a part					
Optimization						
Optimize paths						
ОК						

Nesting analyse and machining						
All the parts are created						
Operations processing Visualisation						
Verticals operations only						
Choose operations to machine						
Cuttings	Counter-mouldings					
Apertures	Grooves					
User calibrations	Rabbets					
Slope faces user machinings	Pockets					
🔽 Continuous 5 axis user machinings	Holes					
Mouldings	Aligned holes					
Analysis of identicals parts	of a part					
Do all parts analys	is again					
Machining of all the parts of identicals p	arts of a part					
Optimization Shapes cutting						
From the smallest to the largest						
From the largest to the smallest						
Optimize paths						
ОК						

Missing tools

Missing tools are shown directly in the Suggested tools tab and no longer in the alpha bar.

Machinable	faces list pièce usinage 47	
Limit analysis	to a piece of part> None	Display area
Drill feature	Form wood milling Suggest	ied tools
Indix	Feature type	Description
#1	Calibrating	1: SLOT MILL (FR-2TA-16) D:16 L:80 Corr.:1 (Tool holder : TP1)
#2	Aperture	1: SLOT MILL (FR-2TA-16) D:16 L:80 Corr.:1 (Tool holder : TP1)
#3	Pocketing	1: SLOT MILL (FR-2TA-16) D:16 L:80 Corr.:1 (Tool holder : TP1)
#4	Groove	Tool not mounted : SLOT MILL, Tool diameter<=10mm, Spindle axis : Z
		Refresh tools list
Operation	simulation Au	utomatic Check for useless operations > Automatic Exit Configuration Redo analyze

Part limit analysis

Aperture detection now uses part analysis limit. Only apertures within or intersecting the part analysis limit will be machined.

Manual aperture machining will use contours found by the operation analysis with the part analysis limit which is used.

Machinable f	aces list pièce usinage 47					
Limit analysis t	to a piece of part > My part	imit	-	🔲 Dis	play area	
Drill feature	Form wood milling Sugge	sted tools				
Indix	Wood milling type	Process	Number of p	WCS	Top mach. name	
🖬 × #1	Calibrating	Calibrating	0/1	Posage pièce		
H #2	Aperture	Aperture	0/1	Posage pièce		
⊡ ×#3	Pocketing	Wood pocketing	0/1	Posage pièce		
🖬 × #4	Groove	Groove	0/1	Posage pièce		
				▼		
Operation simulation						
	Machine	Exit Conl	iguration	Red	o analyze	

Operation information

Additional information has been added in the operation information window. You can access the operation information window using the **Wood machining | Information** command or during the operation analysis:

- Machining property;
- Operation priority;
- Operation pass.

Operation information : * #4		
Cylinder analysis geometric ir	nformations :	*
Depth Width Componant name Componant variant Componant code Tool type Tool axis	6mm 10mm mill#V=simple mill#I=01#R=NR.top simple mill SLOT MILL 0, 0, 1 (Z+)	
Feature given informations :		E
Feature type Feature IDs Feature property Feature comment	Groove @14257	
Machining property	Special	
Operation priority Operation pass	5 1	
Feature given informations :		_
Feature IDs	@14257	-
	OK	1.

Multi-machining

Default saving sub-folders

You can define default sub-folders to save machining files, ISO files, slave parts and setup sheets when running multi-machining in **Tools | Options**, and then in **Routing configuration | Multi-machining** section.

Application configuration		×
Rendering options User information Firme/point Parameter Axes/hatch lines	Multi maching Saving subdirectories Machining files: [Machining	
Curve Sketch B-T Dimension Constraint Constraint P-T Text/note Predefined index	teo Res : (Tso	
Image: Tole Image: Tolerance reference Image: To	Slave pate : (Pate	
Component Component Simplified representation Multi draft G1 Stocks library Shape	Document Res: Sheets	
Material filtering Material filtering New Network Sheet metal : design Sheet metal : unbending Sheet metal : converting Sheet metal : converting Sheet metal : converting Sheet metal : configuration		
Pouting configuration Pout Porcess Pout Pout Solup sheets Database Pout P	Machining documents zoom>[Pats zoom	•
Tools	- DK Cancel	

Multi machining		x
Machining Saving		
Machining files		
V Save	V Close	
Overwrite	Request a confirmation	
Create sets sud-directories		
Directory: D:\TopSolid'WoodCam\Project\ <mark>Machining</mark>		Ä
Iso files		
Create sets sud-directories		
Directory: [D:\TopSolid'WoodCam\Project <mark>\Iso</mark>		<u></u>
Slave parts		
✓ Use slave parts		
		<u> </u>
Directory: D:\TopSolid'WoodCam\Project\Parts		
Documents		
Create sets sud-directories		
Directory: D:\TopSolidWoodCam\Project\ <mark>Sheets</mark>		
ОК	Cancel	

Launching options of multi-machining

BOM sorting depth and BOM filter are now stored to keep the last used options.

: 🖂 🔛		1	· · / ^	-/	V	ے ای	181 👞	- C) ••	\bigcirc	05	5		80	Т
•	崎 • a	a	چ چ		6		🧠 🚺	۲		-)	٩		8
EXPLORE	MULTI-FIL	.ES Depth	FLAT BOM	1	➡ Filter	oom by crit	eria= no filte	er 🔻	Pick o	n the do	icument c	ontaining	the ass	embly:	
$\overline{\mathcal{A}}$															

Modification of the stops to use for multi-machining

When the stops to use for positioning are modified, they are shown after the name of the part/nesting.

Multi machining					
Machining Saving					
Mode > Positioning and machining Machine positionings	Turn over on > Other stop 🔹				
Secondary> *** USER TEMPLATES ***	roas-1 oois-1 ▼	Iso file > Per part			
Selection type Select manually	EXPAND ALL	BACK-UP			
 Select parts by material Matters - Thickness > Plywood - 5 • Select by criterions Criterions Type > • Material > Plywood • Coating > • Thickness > 5mm • Property > Bottom c • • UPDATE MACHINING FACE 	ottom-Right) /Bottom-Right)				
5 Selected parts					
	ОК	Cancel			

Type of zoom during multi-machining

You can manage the zoom done at the end of the multi-machining:

- No zoom;
- Global zoom;
- Parts zoom.

Application configuration	
Rendering options	Multi machining Saving subdirectories
Aligned Sheet metal : bends, ig wizard TopSolidPlanner configuration Routing configuration Process Process Portices and restures analysis Portice between the state of the	Machining documents zoom> Parts zoom Global zoom Frank zoom
	OK Cancel

Machining name in the multi-machining journal

Names of machining files, defined by **WOO_CAM_FILE_NAME** and **WOO_CAM_FILE_NAME2**, are shown after the designation of the path in the project processing journal.

Project processing journal	- • •				
Directory > D:\Users\Desktop\Tests Versions\6.14\New Journal file > Document8#PRJ03.log					
 Informations Top (74-Top-1) Bight side [59-Right side-1, 59-Right side-2] A Left side (42-Left side-1) A Door (119-Door-1) B A Bottom (89-Bottom-1) 					
⊞-≪ Back (104-Back-1)					
С. ОК					

TopSolid'WoodCam 2013: What's New

Part name

The name of the machined part is defined by **WOO_CAM_FILE_NAME** or **WOO_CAM_FILE_NAME2** according to the positioning side of the part.

Modification					—X
Part Material	Misc 3d	5 axis	Info	Security envelope	
Update stock me	thod				
Land and a					June II
10.1					
					<u> </u>
Part name: 119-Do	or.1				
Takiname. [113-bic	0011				
		ОК	Cance		

Nesting machining

Optimization of nesting cutting

Cutting paths of nesting can be sorted from the smallest to the largest part and vice-versa. Optimization by proximity is still available.

Nesting analyse and machining					
All the parts are created					
Operations processing Visualisation					
Verticals operations only					
Charge operations to machine					
Optimization Shapes cutting By proximity From the smallest to the largest From the largest to the smallest					
Optimize paths					
ОК					

Nesting cutting

The starting point of the cutting operation is set to be as close as possible to the center of the panel to keep the outside parts held to the panel as long as possible.

This new mode is automatic but it can be deactivated in the default values or the processes.

This mode is only available if the nesting is positioned using the **Create all the parts** option.



Part visualization of nesting

When the nesting positioning is done using the **Create all the parts** option, the visualization of parts with the **Part** | **Visualize** function lists all the parts and not only one part per family.

Nesting visu	ualisation	
⊂ Rough visu	ualisation	Solid
Parts visua Door B Door B Door B Door B Door A Door A Door A Door A	lisation 2/4 3/4 4/4 1/4 2/4 3/4 4/4	
	ОК	

Operation management

Sort operation by priorities

Sorting by priorities can be automatically done after executing operation from the operation analysis.

Document properties	Routing
TopSolidWood properties	Display operations dialog box
- Units	Calibration
🞝 General	Enable calibrating at the beginning
🐌 General information	Single part Panel
💈 User information	Tool choice
Rendering options	Tool name
Visualisation options	
Loordinate system	I ool diameter : 16mm
Comparing televance	Operation type > Standard
Sketch	Calibrating type > Full
	Calibrating beginning \ Ten left
Constraint	
T Text/note	Calibrating approach > According to operation parameters
🙀 Compass	
Routing configuration	Stop at the end of the operation Stop number > Stop with depression
- 🎲 Process	Aperture
 Cylinders and features analysis 	Enable apertures at the beginning
Part	Tool name
Uatabase	
	Forced
	Tool diameter if forced : 16mm
Rails and pods	Hole minimum diameter to machine : 32.1mm
	Operation type > Standard
🥪 Sawing	
	Stop at the end of the operation Stop number > Stop with depression
Part positioning	Operations analysis
Tools magazine	Authorize operations placed under the part and using a saw
	Associate forms according to material
	Display form every tion window
	Orsplay form execution window
	Usplay reacting analysis window
	Sort operations with priorities (only single part)
	Feed change
	Maximum angle according to the liber direction : (45°

2D optimization of wood operations

2D optimization of wood operations is available on operations with closed path. Operations with open path can be selected but optimization will only use the starting point of the operation.

Operations manager		
General Operations list Cutting	g conditions A-16) [TP1] D:16 L:80 Corr.:1 (Tool holder : TP1)	
E V WCS Posage pièce ■ 1: Calibratio ■ 2: Aperture:	eusinage 31 (WCS @6 Z-) n:	
- b G3 Aperture: - b G4 Aperture: - b G5 Aperture:	Simulation	
- ⓑ ⊕ Aperture: ⓑ ∰7' Aperture:	tit axis list Load Methods	
	Printing Optimize	
Machine tool : TopSolid'WoodCam	😼 Delete Del ➡ Sort	/OODCAM) Material : Bois dur
Toolpath	Init	Cancel
	181 🕶 🤓 🗣 🚫 🔨	
2 2 4 2 2 2 1 1 2 2 2	🛡 - 📫 🛃 - 💟 🚳 📒 🏾	I 🕘 🗑 🕅 🔤 • 14 • 🔚 🚨 🕅
OK Optimization method: SHORTEST PATH Optimization start point:	AUTOMATIC 🗸 Option: NONE	✓ Check machinings can be optimized together: N0 € p

Operations manager			- • ×
General Operations list Cutting con	nditions		
SLOT MILL (FR-2TA-16) (TP1) D:16 L:80 Corr.:1 (T nage 31 (WCS @6 Z-)	ool holder : TP1)	
□ □ □ □ □ □ □ □ □ □ □ □ □ □ □ □ □ □ □	on operation (contouring):		
→ b ≪ Aperture:			
Aperture:			
Machine tool : TopSolid'WoodLam 3X	Flat I able (MISSLER SUFTW	ARE - TUPSULID WUUDLAMJ	Material : Bois dur
Times : Total = 0:01:29	Feed rate = 0:00:52	Rapid = 0:00:37	
Toolpath	Configuration	Exit Can	cel

Adapt frame

It is possible to run the **Work Coord. Syst. | Adapt frame** function after executing the operations performed with the **Wood machining | Operations analysis on a part** function.

This option is now the same as the one used for multi-machining.

You can manage this option in Misc. | Machines | Misc. modifications, and then in the Options tab by checking or not Adapt frame.

Misc machine definitions	• ×						
Working stations Post-processor Options Information							
Frames							
Align positionings frames							
Part positioning							
✓ Create lateral frames							
Create frame under the part (if kinematics allow it)							
Create local frames for saw grooves and rabbets							
✓ Adapt frame							
Saw machining							
Use the nearest frame							
Use sawing unit first							
OK Cancel							

Rails and pods

Pod angle

You can give an angle to a pod before performing automatic proposed positioning of rails and pods. An option allows you reset all pods to 0° before performing the positioning.

Configuration		—				
Exterior margin outside part	:[5mm				
Exterior margin under part	: [5mm				
Interior margin under part	: [5mm				
Maximum number of rails to use	>	4 🔻				
Minimum distance between rails	: [5mm				
Minimum distance between pods	: [5mm				
Pods under scraps at the exterior of the part						
Pods under scraps at the interior of the part (apertures)						
📝 Park unused rails						
Distribute rails under finish support face						
Posit extremity rails on side						
Posit extremity pods on side						
Reset pods angle to zero						
OK Cancel						

Rail and pod positioning on first positioning face

Rail and pod positioning for the first machining position has been improved to optimize the pod positioning using only through operations.

Results in V6.13 and now in V6.14 with a pocket underneath:



Rail and pod positioning on second positioning face

Rail and pod positioning for the second machining position has been modified using all operations and updated stock.

Rail and pod positioning on the periphery of the part

Two new options are available to optimize the rail and pod positioning on the periphery of the part.

These options are optimized for rectangular shapes.

Configuration		×					
Exterior margin outside part	:	5mm					
Exterior margin under part	:	5mm					
Interior margin under part	:	5mm					
Maximum number of rails to use	>	4 👻					
Minimum distance between rails	:	5mm					
Minimum distance between pods	:	5mm					
Pods under scraps at the exterior of the part							
Pods under scraps at the interior of the part (apertures)							
Park unused rails							
Distribute rails under finish support face							
Posit extremity rails on side							
Posit extremity pods on side							
Reset pods angle to zero							
OK Cancel]						

Results without and with these options:



Creation of the rail and pod operation

An option allows you to create the rail and pod operation even if there is no solution under the part.

Rail and pod operation after turning over the part

A rail and pod operation is now automatically created after turning over part if the **Place pods during part positioning** option is checked.

Document properties		
Document propeties TopSolidVood propeties Unit General Unit General User information Visualization options Visualization options Totacance reference Generatical tolerance Sechet Sechet Dimension Constraint TestAnte Constraint Dimension Constraint Dimension Constraint Dimension Process Process Process Protes Database Visualization Pat Dolings Saving Nesting Nesting Tode magazine	Pails and positioning Manual positioning Pats vanisation Stocks visible Pats transparency : 0 Pods colour Delimitation on support face > Outride of the pat > Patly under the pat > Patly under the pat > Patly under the pat > Automatic positioning Exterior margin under pat Exterior margin under pat Interior margin under pat Marimum distance between nals Minimum distance between nals Minimum distance between nals Pats under scaps at the exterior of the pat (apertures) Pats under scaps at the interior of the pat (apertures) Pats under scaps at the interior of the pat Posit externity pads on side Caste pods positioning generation even if no position is found Reset pods angle to zero Posit postermore Posit processor Post processor Post processor Post processor Posit	

Outside margin for rail and pod positioning

An outside margin can be defined for pod positioning under the stock outside the finish part.

Configuration		— × —					
Exterior margin outside part	: [5mm					
Exterior margin under part	:	5mm					
Interior margin under part	5mm						
Maximum number of rails to use	>	4 🔻					
Minimum distance between rails	: [5mm					
Minimum distance between pods	: [5mm					
Pods under scraps at the exterior of the part							
Pods under scraps at the interior of the part (apertures)							
✓ Park unused rails							
Distribute rails under finish support face							
Posit extremity rails on side							
Posit extremity pods on side							
Reset pods angle to zero							
OK Cancel							

Choice of rail and pod set according to the used stop

For each rail and pod set, you can define the list of stops for which the sets will be used.

When automatically creating rail and pod operation, TopSolid'WoodCam will choose the good set of rails and pods depending on the stop used.



Rail and pod definition

Rails along X axis can be defined using the rail and pod definition function. No option is required; the orientation detection is automatic.



Page: 1 / 1

Setup sheets

New setup sheets

New TopSolid'WoodCam setup sheet documents have been added.

Désignation de la pièce : Cêté droit Fichier d'usinage : l'opSolid'WoodCamSetupS Fichier iso :			-	υμ	30 Wood	IIC Cam	Désignation de la Fichier d'usi Fichi Nem de la ma	pièce : Côté droit nage : TopSelid'v er iso : chine : TopSelid'v	vloodCamSetu vloodCam 3X F	ipSheets Rails&Po
Désignation de la pièce : Cifté droit Fichier d'usinage : TopSolid'WoodCanSetupS Fichier iso :			Ordre	Туре	Nom	Position-dural	Position de la ventouse	Angle de la ventouse	Fichier dessin	Vertouse
Fichier d'usinage : TopSolid WoodCanSetupS Fichier isa :			1	Røl	Ral1	67.5				
Fichier isa :	safe und		1	Vertouse	Rail14red1		55	0		
FIGHTER FAM.	HEIS HEG		2	Vertouse	Ral1 Pod2		201.007	0		-
				Vertouse	Roll Polis		348.332	0		-
Renère de la nière - 67			-	Dol:	Rult	951 197	400			-
Référence de la nière - AF67			1	Vertrage	Ref2/Port		55	0		
Matière de la nière - contre-planué			2	Vertowe	Red2-Pod2		201.007	0		-
Fournisseur de la nière - Missler Software			8	Vertouse	Ral 2-Pod3		348.333	0	-	1
Type de la pière - Rafale			4	Vertouse	Ral2-Pod4		495	0	-	1
Traitement de la pière - Augun			3	Plat	Fol2	643.633				
Catégorie de la pièce : TeoSolid'WoodCam			1	Vertouse	Ral3-Pod1		55	0		1
			2	Vertouse	Ral3-Pod2		201.667	0		1
Dimensions du bruf : 1000 X 550 X 20			3	Vertouse	Rail2-Pod3		345.333	0		1
			4	Vertouse	Rail3-Pod4		495	0		1
Non de la machine : TeoSolid WoodCam 3X Rail	s&Pods		4	Rai	Ral4	927.5				
Désignation de la butée : Position 1			-	Vertouse	Rate-Post		55	0		-
				VERDER	Public Post		201/00/	0	-	-
Nombre total d'opérations - 5			-	Vertoose	Rate Post		410	0		-
Temps d'usinage : 0 h 2 mn 24 s			5	Bail	Rult	2702				
			1	Vertouse	Rol5-Pod1		90	0		0
			2	Vertouse	Ral5-Pod2		340	0		0
			8	Verteuse	Rults-Post3		660	0		0
			4	Vertouse	Ral 5-Pod4		910	0		0
			6	Rai	Rati	3032			-	
			1	Vertouse	Ral6-Pod1		90	0	-	0
			2	Vertouse	Ral6-Pod2		340	0		- 0
		_	5	Vertouse	Ral6-PodS		665	0		0
			-	- Cont	Rult		974	u i		
		0	1	Vertrage	RullAdort		90	0		
		fi el		Vertrene	Rel7.Pod2		340	0		
9	ннни	4 9	2	Vertouse	Rul7 Pod3		660	0		0
		н і	4	Vertouse	Rad7-Pod4		910	0	-	0
		H I	6	Rai	Rult	3052				
			1	Vertouse	Ral8-Pod1		90	0		0
1		н і	2	Vertouse	Ral 6-Pod2		340	0		0
		HI	3	Vertexae	Rail#-Pod3		600	0		0
				Vectories	Rails-Pod4		910	0		

		Wood	Cam	Non de la	machine : To	pSolid'Wcod	Can 3X Rails8	Pads				
Repère de la pièce : 47 Référence de la pièce : AF47 Matière de la pièce : contre-plaqué Fournisseur de la pièce : Missier Software Type de la pièce : Rafale Traitement de la pièce : Aucun			Dimensions dashruf : 1000 X 550 X 20 Désignation de la bartée : Position 1 Temps d'usinage : O 12 an 24 s Anthres: Tosticitéritériodation Traes									
Ca	egone ce la	piece : Topso	itig woodcan				Aureu	r: Topsoid	woodcam re	an		
names	10010	sursers .	RENDERAD	ige cas	Denete out	Long. e.ar cano	NOTICE OF BOOK	Prog. Hole.		A/368	10 K	1110
1	Callrage	10	PR-21A-16	HARE 2 TALLES	10	10	4	10800	012,055	0.05	13000	011.0
3	Raturope	28	PB-254-10	PROVIDE 2 THELES	10	50	4	10000	214.199	0.28	13900	1014
4	Personal	11	PO-PL4	FORET & FORD PLAT	8	40	1	10600	291.527	0.5	19900	012.8
1	Pargage	1	10-09-4	FORET & FORE PLAT	5	25	2	10000	457.00	65	13300	6514

Setup sheets with multi-machining

Multi-machining allows you to create setup sheets for each part.

To do this, you have to create a setup sheet folder using the **Operations | Setup sheets** function and affect this folder to your machine template using the **Misc. | Machine | Misc. modifications** function, in the **Information** tab.



Working stations Post-processor Options Information Internal reference : TopSolid'WoodCam 3X Rails&Pods
Internal reference : TopSolid'WoodCam 3X Rails&Pods
Document folder > TopSolid'WoodCam setup sheets

Post-processors

Post-processor assignment

A specific post-processor can be forced when manually generate the iso file as we already can do it for multimachining. This option could be defined within **Misc. | Machine | Misc. modifications** in **Post-processor** tab.

This option is valid for both multi-machining and manual generation of the iso file.

Misc machine definitions
Working stations Post-processor Options Information
Configuration Options
Post processor
Directory : C:\Missler\Pp
Name > MY POST-PROCESSOR
ISO files Directory:
Name :
Filter file name characters
Charaters to replace
Charaters :
Replaced with:
Charaters to delete
Number of characters for the file name (0=without limite) : 0
OK Cancel

New properties in the PDB

New variables are available within the PDB:

- Valorization value : Part.valorisation_property_value
- Total length of the tool with its attachment: Tool.ZProg

Processes

New properties for processes and setup sheets

New properties are available for processes and/or setup sheets to allow better and more advanced customization for machining.

These properties relate to wood operation properties, as well as part properties and machine properties.

A new type of draft view has been added in order to get TopSolid'WoodCam part view and a new table for rail and pod list is available as well.



Comment and PP fields

The **Comment** and **PP** fields of the different operations are now associative with processes and will be reexecuted as other fields already were.

Comment:	 •
PP :	•

Feature operation attributes

Lists are now sorted by alphabetic order in either options or process associations.

Application configuration		x
Rendering options User information Fran Vpni	Dialog box creation Window size: [25	والمحصوب المحافي ومعاور والمحافز
A Sheet m. L www. → TopSolid"Hanner configuration Pouting configuration Offices Offices	Missing machinings > Re-create Feature operation attributes Groove Moulding Rabbet screw standard process	-
	OK Cancel	1
	Operation process choice Operation process choice machining method Groove Default Moulding Default Rabbet Default screw Default standard process Default OK Cancel	

Sorting of process operation

An additional option is available to keep operations contained within a process as a single group.

These options are:

- **Sort normally**: The sort is done as if operations are independent of each other.
- **Do not sort: respect operations order from the process**: The sort will keep the operation order but operations will not necessarily be one after the other.
- **Do not sort AND do not separate machinings**: The sort will manage operations as a single operation so as to keep the order and execute operations one after the other.

ľ	Sort normally.	
4	Sort normally.	i
	Do not sort: respect operations order from the process. Do not sort AND do not separate machinings.	e
-	be not test into de not coparate indenimingo.	

Default values

Initialize default values

You can initialize default values to go back to the original configuration of TopSolid'WoodCam from the default value definition window.

Parameter file edition	n	
ternel 	hining	
		the second second
current file = TopSoli TopSolid'WoodCarr	id'WoodCam n	
📝 default file		🛃 🔁 🕞

Options and properties of the machine template

Miscellaneous options of the machine

TopSolid'WoodCam dedicated options which were in the **Misc. | Machine | Modify locally** function have been grouped together in the **Misc. | Machine | Misc. modifications** function.

Calibrating options

Calibrating options defined in the template document properties have been reorganized for greater consistency between them.

TopSolid'WoodCam options

All settings performed in the TopSolid'WoodCam options (Tools | Options) are now stored and read in the topzwood.cfg file only.

Simplification between main options and document properties

To avoid confusion between what is defined in the main options of TopSolid'WoodCam and what is defined in the properties of the document, the main options have been simplified.

Application configuration		
		Routing configuration
9 User information		
🐵 🔀 Frame/point		
- 9 Parameter		
Axes/hatch lines		
Urve		
- No Sketch		
E Constraint		
Predefined index		
🖶 🧮 Table		
🛓 🕂 🕂 Tolerance reference		
Geometrical tolerance		
T Predefined properties		
- 👘 Multi criteria filter		
🖶 🤌 Component		
Dynamic		
Simplified representation		
🗄 🗂 Stocks library		
Shape		
Material filtering		
	=	
🗄 闷 Sheet metal : design		
Sheet metal : unbending		
Sheet metal : bonding wizard		
TopSolid'Planner configuration		
- A Process		
- Cylinders and features analysis		
- 🗇 Part		
🔚 Setup sheets		
Database		
- 🤬 Verify		
있 Outils		
- 🥝 Multi machining		
	-	
		OK Cancel
	_	

TopSolid'Planner 2013: What's New



This document describes the improvements made to the **TopSolid'Planner** software: **2013** version.

Panel brush

The new panel brush allows you to change the panel type (one edge, four edges, with laminates...), change the support material, configure the edges (profile, matter, cuts and code) and configure laminates of a panel entity directly in TopSolid'Planner.



Panel configuration by panel support and edges matter and coating modification.

panel brushes.







Prerequisites prior to creating a panel brush

The panel brush, like the other brush types, requires the creation of panel categories described in the brush definition.

A panel is made up of a support, edges, laminates and options (cut type, laminates...). The support, the edge and the laminate should therefore have dedicated categories.

Creation of the matter and coating library

Creation of the matter and coating library for:

- The support;
- The edges;
- The laminates.

The process to create the edge, support and laminate's matter and coating categories is the same as the matter categories.



Planner's matter categories statement

Planner's matter categories statement in Tools | Options | TopSolid'Planner configuration | Categories for:

Support's matter and coating

Components	Matters	Panels	Edges	Edge matters	Supports	Support matters
Name		Designation				
TSW-Support-mat	erial-Gross	%Gross				
TSW-Support-mat	erial-Painted	%Painted				
TSW-Support-mat	erial-Plated-Color	%Plated Cold	Dr			
TSW-Support-mat	erial-Plated-Wood	%Plated Woo	bd			

- Edge's matter and coating

Components	Matters	Panels	Edges	Edge matters		
Name		Designation	Designation			
TSW-Edge-material-PVC-COLOR		%TSW-Edge-material-PVC-COLOR				
TSW-Edge-material-PVC-WOOD		%TSW-Edge-material-PVC-WOOD				
TSW-Edge-material-WOOD		%TSW-Edge-material-WOOD				

- Laminate's matter and coating

Laminate matters Sub com	
Name	Designation
TSW-Laminate-material-MELA	%TSW-Laminate-material-MELA
TSW-Laminate-material-WOOD	%TSW-Laminate-material-WOOD

Edge's profile model creation

An edge model contains two data: the profile geometry (which can be managed by a catalog) and the matter's categories available for this profile.

The edge's model creation starts by the standard TopSolid'Wood edge creation (see **TopSolid'Wood Help | Procedures | Creation of an edge model**). It is also possible to copy an existing edge model.

- Launch the edge's model creation function using the Planner menu, Tools | Define model | Define edge model.
- ep=2

h=19

- Select the edge's matter categories that can be used by this edge profile.

Add edge material categories TSW-Edge-material-PVC-COLOR TSW-Edge-material-WOOD	TSW-Edge-material-PVC-WOOD	
	ОК	

•

• Then select the default matter to use in the chosen matter categories.

Select a material for edge	x
PVC U - ASH	
PVC U - BEACH	and the second
PVC U - WALNUT	
PVP U - MAPPLE	W-Edge-material-PVC-WO
	PVC U - BEACH
4 Elements	
ОК	

Then, the edge model is automatically generated. It remains only to save it in your component library.





Warning: The edge model has to be saved in a library family named **edge** to be recognized during the codification generation.

Moreover, the file **lib.cfg** located in the folder *Missler\V614\z\woo\lib\TOPWOOD* has to be copied in the library root folder.

🗹 Filtrer Standard:	
Dressing Room	-
Famille, type, variante:	
🔁 Dressing Room	
🗄 🛅 Cabinet	
🗄 🛅 Component	
Config	
E Door	
🗄 🛅 Edge	
🚊 🛅 First thin edge	
📽 Flat	
📟 🕱 Quarter round	1
🗄 🧰 Thick edge	
🗄 💼 Thin edge	
🗄 🧰 Hardware	
🗄 🧰 Internal arrangement	
🗄 🧰 Material	
🗄 🧰 Panel Components	-

Edge's categories statement

• Declare in **Tools | Options | TopSolid'Planner Configuration | Categories | Edge** the different categories previously created.

Components	Matters	Panels	Edges	
Name		Designation		
TSW-Thick-edge		%Thick edge		
TSW-Thin-edge-P	VC-COLOR	%Thin edge PVC COLOR		
TSW-Thin-edge-P	VC-WOOD	%Thin edge PVC WOOD		
TSW-Thin-edge-WOOD		%Thin edge WOOD		

Filter	Standard	Family	Туре	Variant	
×	Dressing Room	edge	Thick edge PVC		

Panel model creation

Once the support's matter and the edge's model are created, it is possible to generate the panel model which will consolidate the different support's matter, edge and laminate.

 Launch the panel model creation function using the Planner menu, Tools | Define model | Define template of panel.



• Choose the support's matter categories available for this panel and the default matter.

Add support material categories		X
TSW-Support-material-Plated-Color TSW-Support-material-Plated-Wood	TSW-Support-material-Gross TSW-Support-material-Painted	 ▲ ▲

• Choose the edge profile's categories available for this panel and the profile, the code and the default matter.



The panel model is automatically generated and the panel configuration window opens with different configurations areas.

<u>Edge type</u>

	N*	Codification	Edge type - code	Length	Beginning cut type	End cut type	Modifiable
~	1		Quarter round - EP 2mm	562.0mm	Covering	Covering	X
~	2		Quarter round - EP 2mm	350.0mm	Covered	Covered	X
~	3		Quarter round - EP 2mm	562.0mm	Covering	Covering	×
~	4		Quarter round - EP 2mm	350.0mm	Covered	Covered	X

- **Checkbox**: Allows you to enable or disable the edge.
- **N°**: Edge number.
- **Codification:** Allows you to display the selected edge codification.
- **Edge type code**: Allows you to display the selected edge type and code. By double-clicking on this case, it is possible to modify the edge.
- Length: Allows you to display the edge length of the model.
- Beginning cut type/End cut type: Allows you to set the cut type on the different edge.
- **Modifiable**: If the case is checked, it will be possible to configure this edge in TopSolid'Planner. All the checked edges will be identical after modification in TopSolid'Planner.

Edge configuration

- Edges are similar: Allows you to have four identical edges on the panel.
- **Cuts are similar**: Allows you to have the same cut type on the four edges.
- **Cuts are modifiable**: If this checkbox is checked, it will be possible to configure the edge's cuts type in TopSolid'Planner. The cuts type modified are the cuts of the edge checked **Modifiable**.
- **Support matter is modifiable**: If this checkbox is checked, the support's matter and coating can be modified in TopSolid'Planner.

Advanced options

>>

As with the panel entity creation, the advanced options allow you to set the panel parameters.



Here, it is strongly advised to use the panel **Simplified representation** to ensure the snap on panel's faces and their interchangeability.

Laminate types

• To configure the laminate, first check **Make laminate** in the advanced options.

The part about the laminate is now available.

• Check a box to activate a laminate and choose the laminate's matter categories and the default matter.

_ami	inate s	hapes			
	N*	Codification	Material	Coating	Thickness
~	1		Melamine	TMaterial%Beech-un	@81=1mm
~	2		Melamine	TMaterial%Beech-un	@82=1mm

Laminate configuration

ADD CODIFICATIONS
🔽 Laminates are similar
🔽 Laminates matter is modifiable
🔽 Laminates are activated

- Laminates are similar: Allows you to have two same laminates on the panel.
- Laminates matter is modifiable: If this option is checked, the laminate's matter and coating may be modified in TopSolid'Planner.
- Laminates are activated: This option allows you to disable the laminates during panel configuration in TopSolid'Planner.
- Save the panel model in your component library.
- Once the different panel models are created and saved in the library, declare the different panel's model categories in **Tools | Options | TopSolid'Planner Configuration | Categories | Panel.**

Components	Matters	Panels	Edges
Name		Designation	
TSW-Panel-1-front-edge		%1 front edge	
TSW-Panel-4-edge		%4 edge	
TSW-Panel-front-b	back-edge	%Front and t	oack edge
TSW-Panel-front-r	ight-edge	%Front and r	ight edge

Define panel brush

Panel brush creation

Because the panel brush only applies on panels, you must first create them. The options (support material, edges, laminates...) of the created panel are not essential because they will be then modified by the brush.

Launch the brush panel function in the Planner menu, **Tools | Define panel brush**.



- **OK:** Allows you to validate the different panel brush parameters.
- **Designation:** Allows you to name the brush as it will appear in the TopSolid'Planner configuration. The different panel brushes are grouped by their designation and their categories.
 - Panels selection : To select the different panels on which apply the panel brush, two ways are available :
 - Selection by **Filter**: Allows you to use a filter to select the panel.

In case this filter chooses the parts according to a property (type, designation...); it is important to apply this property to the **panel entity** and not to the **support**. To do this, launch the **Modify element** function on the panel entity, select **Panel Process**, and then **Modify Cutting-up**. Thereby, the property will be applied to the panel entity.



It is also possible to refine the filter by adding the panel entities:

• In Tools | Options | Multi-criteria filter, right-click on the filter name and select Filter the panel entities.

0	Filter
0	Add
4	Сору
4	Filter panel entities

• Manual selection: if **No filter** is selected in the drop-down list, the **Panel(s)** option appears in the dialog bar.

Panel(s):

It allows you to select manually several panels on which the panel brush will operate.

The different panels selected by the filter or manually are displayed in red.



- Template panel is modifiable = YES/NO:

- **YES**: It will be possible in TopSolid'Planner to change the panel model. It allows you for example to switch from a model with four edges to a model with one edge or with laminates...
- **NO**: The panel model cannot be changed in TopSolid'Planner.
- Multi-thickness = YES/NO:
 - YES: The panel brush will modify all the selected panels, regardless of their thickness.



Using this option on **Yes** involves that all the panel categories using by this brush are available in all configurable thicknesses.

• NO: Only the panels whose thickness is filled in the Thickness of panels field will be modified.

Thickness of panels= 19
- Once all the information has been filled in, validate the brush with **OK**.
- Choose the panel categories which can be used by the brush.

Add categories to brush Shelf material	TSW-Panel-1-front-edge TSW-Panel-4-edge
	ок

• Then, choose the panel model used by default.

1 front edge 4 edge	Range 1	
	Range 3 Range 4	4 edge Range 2
4 Elements	ОК	

The brush is created and the different selected panels are configured as the chosen panel model.

• Save the component.

Some data about the panel brush can be modified from time to time for this component from the tree.

- In the tree, right-click | Edit set | Brush set.
- Then on the category type to modify, **right-click | Modify**.



If the panel template used is modified after being used by the brush, it is possible to reset the brush with a **right-click** on the brush line **| Reset brush.**

对 BRUSHES SET : (2)

Using a panel brush in TopSolid'Planner

The panel brush, like the subcomponent brush and the material brush, is available when inserting and modifying a component.

When configuring the component, the panel brushes are available in the **advanced mode**.



This tab available in the **advanced mode** allows you to configure the different component's panel brushes.

		Apply to all
Designation	Туре	
Side panel	Range 2	

- Apply to all: Allows you to apply the chosen parameters to all document's equivalent brushes.
- **Designation**: Allows you to show the brush name.
- **Type**: Allows you to show the name of the panel template used.

By double-clicking on the brush line, the panel's configuration window opens. It allows you to configure the panel depending on the chosen panel model and its parameters.

Select a style for Front & ba	ack			×
Front and back edge	Range 1 TMP2			
1 front edge	Range 2 TMP3			
	Range 3			
	TMP			1 front edge TMP3
	6 Elements			
Supports	Edges	Laminates	C.	uts
	Code = EP 4mm			
	ОК			

- **Panel model configuration:** Panel category choice and model selection. If during the panel brush creation it was informed that the panel model cannot be modified, this area is grayed-out.
- Support: Support's matter configuration. If in the panel model it was allowed to modify the support's matter, it is possible here to click on the preview to choose the support's matter in the different categories.
- **Edges:** Edges configuration. Allows you to simultaneously configure the different edges checked **Modifiable** in the panel model.
 - If no edge has been checked **Modifiable**, these boxes are grayed-out.
- Laminates: Laminates configuration. Allows you to configure the laminates if the laminate modification was allowed. The checkbox allows you to activate or deactivate the laminates if the laminate deactivation was allowed in the panel model.
- **Cuts:** Cuts configuration. Allows you to set the edges' cuts types if the checkbox **Cuts are modifiable** was checked in the panel model.

Advanced using

The advanced use of panel brushes allows you to drive:

- Support matter's categories in terms of their thickness;
- Edge matter's categories in terms of their dimensions (height and thickness),
- Laminate matter's categories in terms of their thickness.

Edges matter's categories driving

• In the edge's model file, edit the edge's model in the tree by launching the **Define model | Define edge model** command.



• On the Edge's matter category line, right-click | Modify.

Four options are then available:

NO TEXT	TABULATED TEXT	TABLE TEXT	Text	
---------	----------------	------------	------	--

- **NO TEXT**: This option corresponds to the simple mode; the matter categories do not change depending on the dimensions. After selecting this option, choose the matter categories and the default matter.
- **TABULATED TEXT**: This option allows you to drive the edge matter categories according to its height (parameter h in the edge which corresponds in the assembly to the panel thickness).

The Tabulated strings window opens.

• In the first **Reference value = h** column, fill in the different values which will establish the intervals.

In the example shown here, the established intervals are:



Reference value = h	String	
3	TSW-Edge-material-PVC-COLOR	
12	TSW-Edge-material-PVC-COLOR,TS	
16	TSW-Edge-material-PVC-COLOR,TS	
19	TSW-Edge-material-PVC-COLOR	
2	TSW-Edge-material-PVC-COLOR,TS	

- In the second **String** column, double-click on the line to select one or several categories to use for this interval.
 - **TABLE TEXT:** This option allows you to drive the edge matter categories according to its dimensions (parameters h and t in the edge).

After selecting this option, a window opens with a first **Value of h** field; it corresponds to the first **h** value for the first height interval.

alue of h			<u> </u>
Value: 19			
	OK	Cancel	

In the displayed table:

- The lines correspond to the different edge's thickness intervals;
- The columns correspond to the different edge's height intervals
- To modify, add or delete an interval, right-click on the column head.



• To insert or to delete a line, right-click on the line head.

11:21		17
]2;3	Insert new line	N
] 3; 4	Remove line	43
	and the second second second	

• To add edge's matter categories which correspond to a thickness and a height interval, double-click on the corresponding case.

rings tab	le		23
ep∖h	<= 19	>19	
<= 1			
]1;2]			
>2			

In the example above, the categories loaded in the selected case will be available when the edge thickness will be less or equal to 1mm and when its height will be less or equal to 19mm.

- **Text**: For advanced users, it is possible here to select a text which contains an expression to drive the categories with more parameters and options.
- Once the edge matter categories are driven, save the edge template in the library.

Panel support matter categories driving

The panel support matter categories driving starts by the support template creation with the Planner menu, **Tools** | **Define template** | **Define support template.**



• Choose the support matter categories by default and the default matter.

The chosen categories will be then modified by the driving.

Add support material categories		X
TSW-Support-material-Plated-Color TSW-Support-material-Plated-Wood	TSW-Support-material-Gross TSW-Support-material-Painted	
	ОК	

The support template is then automatically generated and the tree opens on its parameters.

• On the Support material categories line, right-click | Modify.



Three options are available in the dialog bar:



- NO TEXT: Allows you to drive the matter categories easily, without thickness condition. In theory, if the support matter categories are not driven with the Tabulated text function, the creation of a support template is not useful.
- TABULATED TEXT: This option allows you to drive panel support matter categories according to its thickness (parameter h in the support template). The settings work in the same way as the Tabulated text in the edge template.
- **Text:** For advanced users, it is possible here to select a text which contains an expression to drive the categories with more parameters and options.
- Save the support template in the standard library.

The different panel support categories created have to be declared in **Tools | Options | TopSolid'Planner** configuration | Categories.

Components	Matters	Panels	Edges	Edge matters	Supports
Name		Designation			
Gross-support		%Gross supp	oort		
Plated-support		%Plated sup	port		

With these settings, when a panel template is created, the support template categories will be required, as well as the default matter.

Add support categories		×
Plated-support	Gross-support	

Laminate template creation

Creating a laminate template allows you to drive the laminate's matter categories according to its thickness.

- Start by creating a laminate template with the Planner menu, Tools | Define template | Define template of laminate.
- Choose the matter categories and the default matter.

The chosen categories will be then modified by the driving.

The laminate template is automatically generated and the tree opens on its parameters.

- Save this new laminate template in the standard library.
- In the tree, on the matter categories line, **right-click | Modify.**



The laminate matter categories driving works in the same way as the support matter categories driving.

The different created laminate categories have to be declared in **Tools | Options | TopSolid'Planner Configuration | Categories.**

Components	Matters	Panels	Edges	Edge matters	Supports	Support matters	Laminates
Name		Designation					
Laminate-T1		%Laminate ty	/pe 1				
Laminate-T2		%Laminate ty	/pe 2				

With these settings, when a panel template is created, the laminate template categories will be required, as well as the default matter.

- 🅎	- 90
- 🐑	
	5
3	Define template of laminate

Light management in TopSolid'Planner end user

In TopSolid'Planner, a new interface allows you to manage very easily the Planner document's lights.

The **Lights** index gives a direct access to light activation/deactivation and intensity settings. The light list is updating when some components with light are included or deleted.

Each line corresponds to a component included in the Planner which contains one or several lights. The line name corresponds to the name of the included assembly which imports lights.

Switch on or switch off the line or switch on or switch off all the lights contained in the assembly.

It is also possible to vary the light intensity.



Intensity				11100
Sku intensitu		-0-	- 8	
Sun intensity	()	- 4	
		11111		
11 MA			Nelly	
in in		/	ante.	
W <	\triangleleft		> E	
11111	1		1111	
1770			Bill	
	mini	mm		
	_			
	Sun cal	culator		

A dynamic modification of the sky and the sun is now available by clicking the **Sun light** line. It allows you to set the global lighting of the document and calculate the sun position using the **Sun calculator** function.

Country : France City : Toulouse Coordinates: 43.60° *North 1.45° * Time zone: 0 Daylight saving time Date and time Date: 5 / 12 / 2012	▼ ▼ Eas
City : Toulouse Coordinates: 43.60° *North 1.45° * Time zone: 0 Daylight saving time Date and time Date: 5 / 12 / 2012	▼ Eas
Coordinates: 43.60° *North 1.45° * Time zone: 0 Daylight saving time Date and time Date: 5 / 12 / 2012	Eas
Time zone: 0 Daylight saving time Date and time Date: 5 / 12 / 2012	
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Date and time Date: 5 /12 /2012	
Date and time Date: 5 / 12 / 2012	
Date: 5 //12 //2012	
provide and a second se	
Time: 14 : 50	
System time	

Elements visualization

The new function **Elements visualization** allows you to hide some elements included in a TopSolid'Planner document.

It allows you for example to momentarily hide a door to place a shelf behind.



This function is available in the Visualization | Elements visualization menu.

Mode = HIDE

Mode= HIDE 🖘 Elements to hide:

The **Hide** mode allows you to select shown elements to hide them.

-	Mode	e = SHOW	
Mode=	SHOW \$3	Elements to show:	

The **Show** mode allows you to select hidden elements to show them.



Improving face's name entering

In an element's positioning sequence creation, during the face's name entering, a drop-down list is now available. It contains the names of the publishings defined in **Tools | Options | Components | Components management.** This makes it easier to enter the name of the face and reduces the positioning errors due to a false publishing's name.

sunation			
Result of:	QuestionSelect the left compo 🔻	Named face or edge:	Back
			Left
		ini.	Right
		0	Front
			Back
			Bottom
			Тор
			Width middle
			Depth middle
			Height middle

Centering constraint in positioning sequence

Now it is possible to create a centering constraint in a positioning sequence.

The **Automatically publication creation** function has been enhanced with the **Median publications** option. The names of these new publications can be adjusted with the other publication's name in **Tools | Options | Components |Components management.** These planes are placed in the middle of the two other publications of the direction.

OK Assembly= simplified_set

Median publications= YES 🚱 Reference frame for automatic= CURRENT COORDINATE SYSTEM



Of course, it is possible to create the publishing on an alternative set and on the current coordinate system.

In this example, the normal publishing is placed on the main assembly (all the elements), but the median publishing is placed only on the cabinet (without foots and the facades).

Automatic publications		
Publishing1 X-	:	Left
Publishing2X+	1	Right
Publishing3 Y-	5	Front
Publishing4 Y+	4	Back
Publishing5 Z-	:	Bottom
Publishing6 Z+	1	Тор
Publication7 XM	3	Width middle
Publication8 YM	3	Depth middle
Publication9 ZM	1	Height middle

When adding a rule in a positioning sequence:

- In an implicit constraint: to place a median publication on a selected component's publication, it is possible to use the **Mate or alignment** constraint. This operation is similar to the positioning in older versions with the new possibility to use the median publishing.
- A new positioning constraint had been created in the simple constraint positioning: **Plane on plane** constraint with a **Second question** checkbox which gives the possibility to ask a second question in the same positioning constraint.

Description				
Plan médian				
Question				
P1				
Second question				Contract of
Ask the question P2				
Constraint				
Plane on plane 🔹 👻	Value: 0	mm		TE LET
Source topology (pamed publications)				R. DEV
	Width middle		31.030	
			Fixed camera	Dynamic Dynamic

In this example, the **Width middle** publishing will be placed on the plane located between the two faces selected in the questions **P1** and **P2**.

Improving positioning filter

Some improvements have been made to the Planner positioning filter.

Possibility to filter the Vertical and Horizontal component's faces in a positioning.
 For the Horizontal faces, it is possible to refine the filter by filtering the Bottom and the Top face.





Positioning example with **Vertical** faces filter.

Positioning example with **Horizontal Top** faces filter.

- A **Ceiling** filter has been added. It works the same way as the **Ground** and the **Wall**, but it allows you to select the ceiling.



Version number in TopSolid'Planner Client

It is now possible in TopSolid'Planner Client to know the TopSolid'Planner version number (created by the administrator during generation) and the TopSolid core version.

Wall dimensioning improving

The wall dimensioning has been improved to makes it easier to modify. Moreover, the over dimensioning is now better managed.



🗩 DRIVERS SET : (3)

Using geometric drivers in TopSolid'Planner

The geometric drivers (especially the points) are now managed in TopSolid'Planner. To be used, the component must contain geometric drivers.

Example of a component **Triangular cover** controlled by 3 points.

The component has to be saved normally in the Planner library.

Then, as with a driver block component, an empty positioning sequence has to be created.

Positionings				
🔬 STD			4	Add a group
			e.	Remove a group
			N 💔	Modify a group
Rules				
🥘 Hide			9	Location rule
			38	Miscellaneous rule
				Dimensioning rule
				Tool rule
				Modify a rule
			/ 💻	Remove a rule
Loop	Configure	Simplified representation		

The **Hide** rule allows you to hide the component during the geometrics drivers' selection. The **Tool** rule can be added to create the component's tools.

When inserting the component in TopSolid'Planner, the geometric drivers are required and the component is included depending on the selected drivers.

Top point:





Detailed quote in TopSolid'Quote

It is now possible to show on a quote some subcomponents of an element on several quote lines. It allows you for example to display the handle type and the materials used in a component.



N°	Product	Sketch	Q.	Unit	Unit € H.T.	Total € T.T.C.
5	Recessed door kit		1,0000		41,30	41,30

In the older versions, this door was displayed on one quote line.

5	Recessed door kit		1,0000	 41,30	41,30
5.1	Cylindrical handle 512		1,0000	15,34	15,34
5.2	Glasse panel	1111	1,0000	15,16	15,16
5.3	Recessed hinge	-/e	3,0000	3,60	10,80

Now it is possible to display the door's subcomponents on several quote lines.

Displaying the subcomponents

• Start by defining, in TopSolid'Quote, the *Door* product in the first level as Subassembly. To do this, edit the item and in the **Mode** field select **Subassembly**.

	Mode	
🞯 Edit item 💦	O Unitary	Subassembly

<u>Note</u>: By default, all products are in **Unitary** mode in order not to change the current TopSolid Quote's behavior.

The door and all its subcomponents are now displayed on several quote lines.

Hiding some items

Some items do not need to be shown in the quote (such as the hinges, for example).

• To not show them, edit the item and in the **Mode** field uncheck **Show item**.

Show Item

Shifting the data in the quote To allow easier quote reading, it is possible to shift the quote's information (description,	1
quantity, price) according to its assembly level.	17 - 18
Here, it is the quote line's number which is shifted. At each new assembly sub-level, the quote line's number is shifted	1.1
• To use this function, just modify the quote reporting model and edit the information case to shift. These new variables are in Data Column I. Quote line and use the variable	1.2
with indents.	1.2.1

6		
Expression	⊞ Quote [Devis]	
Data Column	GuoteHeader [Entête du devis]	
Data Column	QuoteLine [Ligne de devis]	
System Variable	IneNumber [Numéro de ligne]	
Summary		
		1
	Boo Description [Désignation]	

Price displaying rule

TopSolid'Quote component's price displaying is now using a new rule in the **Quote detail** and **Quote preview** areas.

- Either it is only the superior component's price which is shown.
 - If, when the component is unfolded, the subcomponent's prices sum is not equal to the component's price, only the component's price is displayed.

2	Recessed door kit	I	1	29,63	29,63
2.1	Cylindrical handle 256		1		
2.2	Glasse panel		1		

In the example above, the door kit is made up of a handle, a panel and three hinges which are not displayed. So, the displayed subcomponent's prices sum is different from the door kit price so only the door kit's price is displayed.

- Or only the subcomponent's prices are displayed.

If, when the component is unfolded, the subcomponent's prices sum is equal to the component's price, only the subcomponent's prices are displayed.

2	Recessed door kit		1,0000		
2.1	Cylindrical handle 256		1,0000	7,67	7,67
2.2	Glasse panel		1,0000	11,16	11,16
2.3	Recessed hinge	-	3,0000	3,60	10,80

In this example, the displayed subcomponent's prices sum is equal to the component's price so only the subcomponent's prices are displayed.

TopSolid'SheetMetal 2013: What's New



Introduction

The 2013 version of TopSolid'SheetMetal includes many new features that will further improve your productivity. This user guide introduces the main new functions and the exercises accompanying this manual will help you to learn how to use them.

General

Import multidocs - Confirming general parameters

Purpose of the function

In this version, the user can access the general parameters dialog box when creating a part for the first time, when importing Multi-Docs TOP/PCH and/or Multi-Docs DXF/DWG/IGES.

Triggering the function

Tools | Options | Importation | Multi-Docs Top/Pch and/or Multi-Docs DXF/DWG/IGES.

nfiguration User information	Multidocs TOP > PCH
Colors	Vuse file interface
Shortcut key General	
Importation	Interface file list
Geometry	Test file extension csv Field separator TAB character Unit for thickness mm
Other options Multidocs TOP > PCH Multidocs DXF/DWG/IGES	Create PCH file in the same folder as its TOP file
Punch Cut Noving	The first line of the list file gives the names of the columns
Simulation	Header columns names Order of the columns
Post processor Lasks manager	Folder REPERTOIRE Available heids Undered columns Part Name REPERE
Display options	Path Chemin complet
	Quantity QUANTITE Path
	Thickness Epaisseur Matter Thickness
	Matter Matiere Machin
	Designation Désignation Designation
	Customer Client Command
	Commande Plane Set
	Plan Date
	Set Ensemble
	Uate Uate
	CFG#IMP#Use multiparts recognition per design life Separator # 20 Ask for each part name
	Ask general parameters for each part
	Create a Nesting Order NO

Keeping general parameters

Purpose of the function

Possibility of keeping the most recent values entered during the TopSolid session.

This concerns:

- The machine
- The matter (material)
- The matter type (material type)
- The thickness

onriguration	General parameters	
User information		
Colors	Default machin alpha2-1500w 🔻	
s Snoncul Key		
Importation	Turret	
General parameters	Automatic tool mount on turet	
Other options Multidocs TOP -> PCH Multidocs DXF/DWG/IGES	Standard turret Browne for arcade210 -	
Punch Cut Nesting	Default thick 15000mm Default shade actes inoxydable •	
Simulation Post processor Tasks manager Display options	Colouring for the parts	
	☑ Keep in dialog box previous input parameters (machine, matter, thickness, matter typ)	
	Keep in delog box previous input parameters (mechine, matter, thickness, matter typ) Copy general information from the TopSolid document	
	Keep in delog box previous input parameters (machine, matter, thickness, matter typ) Copy general information from the TopSolid' document	
	Keep in dalog box previous input parameters (machine, malter, thickness, matter typ) Copy general information from the TopSolid' document From unbend	
	Image: Seep in delog box previous input parameters (mechine, matter, thickness, matter typ) Image: Seep in delog box previous input parameters (mechine, matter, thickness, matter typ) Image: Copy general information from the TopSolid document Image: Seep in delog box previous input parameters (mechine, matter, thickness, matter typ) Image: Seep in delog box previous input parameters (mechine, matter, thickness, matter typ) Image: Seep in delog box previous input parameters (mechine, matter, thickness, matter typ) Image: Seep in delog box previous input parameters (mechine, matter, thickness, matter typ) Image: Seep in delog box previous input parameters (mechine, matter, thickness, matter typ) Image: Seep in delog box previous input parameters (mechine, matter, thickness, matter typ) Image: Seep in delog box previous input parameters (mechine, matter, thickness, matter typ) Image: Seep in delog box previous input parameters (mechine, matter, thickness, matter typ) Image: Seep in delog box previous input parameters (mechine, matter typ) Image: Seep in delog box previous input parameters (mechine, matter typ) Image: Seep in delog box previous input parameters (mechine, matter typ) Image: Seep in delog box previous input parameters (mechine, matter typ) Image: Seep in delog box previous input parameters (mechine, matter typ) Image: Seep in delog box previous input parameters (mechine, matter typ) Image: Seep in delog box previous input parameters (mechine, matter typ) Image: Seep in delog box previnters (mechine, matter typ) Image: Seep	
	Keep in dalog box previous input parameters (machine, matter, thickness, matter typ) Copy general information from the TopSolid document From unbend Add part to taskmanager Basily part machining document	
	Keep in dalog box previous input parameters (machine, malter, thickness, matter typ) Copy general information from the TopSolid' document From unbend Add part to takmanager Basity part machining document Cope part machining document	
	Image: Seep in delog box previous input parameters (machine, matter, thickness, matter typ) Image: Copy general information from the Top/Solid document From unbend Image: Add part to taskmanager Image: Basily part machining document Image: Copy general information from the Top/Solid document	
	Image: Seep in dalog box previous input parameters (machine, matter, thickness, matter typ) Image: Copy general information from the TopSolid document From urbend Image: Add part to taskmanager Image: Basity part machining document Image: Copy general information from the turet)	
	Keep in dalog box previous input parameters (machine, matter, thickness, matter typ) Copy general information from the TopSolid document From unbend Add part to takmanager Basity part machining document Close part machining document Add with the questions (to choose transformation the turret) Ignore modifications of Name of Matter of the 3D part during upadies of the punch part	
	Image: Seep in dalog box previous input parameters (machine, matter, thickness, matter typ) Image: Copy general information from the TopSolid document From unbend Image: Add part to taskmanager Image: Basily part machining document Image: Close part machining document	
	Keep in dalog box previous input parameters (machine, matter, thickness, matter typ) Copy general information from the TopSolid document From unbend Add part to taskmanager Basity part machining document Copy general information from the turnet) Information from the turnet) Information from the turnet) Information from the turnet) Information flow matching document Basity part maching document Information flow matching document Basity part matching document	
	Keep in dalog box previous input parameters (machine, matter, thickness, matter typ) Copy general information from the TopSolid document From unbend Add part to task-manager Basily part machining document Close part machining document Ask the questions (to choose transformation the turet) Ingrace modifications of Name of Matter of the 3D part during upadles of the punch part Bill of material file Designation of the names of columns	

Generating labels

Purpose of the function

Possibility of generating labels with set parameters.

Triggering the function

Tools | Options | Simulation | Labels Cards

Select the desired mode to generate the labels amongst the available modes:

- One label per part of nesting
- One label per kind part of nesting
- One label per kind part of order

To use the labels, a label template model is required in the Punchdata directory.

Lebel: cad User information Shortcet kay Description Statistics Description Constitution Constitution Description Constitution Description Constitution Consti	Application configuration	THE REPORT OF TH	X
OK Carel	Configuration Configuration Configuration Configuration Cons Shortcut key General Punch Configuration Punch Configuration Punch Configuration Configurati	Label: cod: Mode of creation of label: One label per part of nesting One label per part of nesting One label per part of nesting One label per kind part of order Proce given paper size for priving label: Paper size (Ad	
		OK Cancel	

By default, the model shall be called *Labels.Dft*.

If you would like to create a model via machine, simply rename the file as follows: *Labels_Machine name.dft*

Example: Labels_bystronic.dft

Models are available to users in the following directory: \Missler\v614\local\french\template\pch_labels\

Labels_A7V.dft	Model of a label in A7 vertical format
Labels_A4V_x8_1.dft	A4 template containing 8 labels
Labels_A4V_x8_2.dft	A4 template containing 8 labels (other model)
Labels_A4V_x16.dft	A4 template containing 16 labels
Labels_A4V_x24.dft	A4 template containing 24 labels

Example of templates:

6 To	pSolid (alpha) by Missler Softw	are - [Draft : C:\Miss	ler\V614\local\e	nglishUS\template\	pch labels	abels A4V x24.dft <	<current>> (Ass</current>	ociative mode)]
	File Edit Parameter View	v Curve Assemb	ly Dimension	Bill of material D	etailing T	ools Attribute An	alyze Piping	Bending wizard
	🖻 • 🔜 • 🚺 😓 •	- 🗙 🗐 🧳		111 🚣 🐰 4	6	& & K II	1 9 tes -	· 🔎 • 🔚
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pSo		_	1		h		4	

Extension of label files

Label files are Draft type files, with the extension **.Labs.dft**.

TopSolid 2013

The label file created for a given nesting (mode 0 or 1) takes the same name as the nesting.

TopSolid'SheetMetal 2013: What's New

Nesting order - Complex nesting		X
Parts Gaps Sheets Nesting General		
Saving Besult files name		
Saving directory	sting_rest Dossiers_Patricia\ANNEE_20	12\TESTS_614_100\GRANE
Optimizing	Criterion	Strategy
 Generated programs number 	Loss	 Maximum Fill nesting Straight front
🔘 Global loss ratio	Price	One part
Loss to reach - by decreasing parts rotation st	ep (%)	
Default nesting parameters	Modify	
Linked actions		
	Linked actions	
Matter acier Type	Thickness 2.0000mm	Machine emz3610
Crieck parts compatibility (matter and thickness)		
LUdu Save		

The label file created for a given order (mode 2) takes the same name as the order.

TASKS MANAGER		X
Parts to machine Matters to use Nests to use Parts done Nestings done Orders don	e	
Ref. D. S. P. Com. C. Mat. 742193065/C2/Caps de p. ce.3 CDE, 2020 Aluminum 742193065/Caps proceal CDE, 4040 Aluminum 742193065/Caps proceal CDE, 2020 Aluminum 742193065/Caps proceal CDE, 2040 Aluminum 6, P10 acier acier 6, P.P10 acier acier	Th. Occ. Q. Q. Mx Fan. Corp. Pio. Date Mac 4.0000rm 1 1 0 -1 No 0 15/1/2/211 433 2.0000rm 1 1 0 -1 No 0 15/1/2/211 433 2.0000rm 1 0,0 -1 No 0 15/1/2/211 433 2.0000rm 0,0 -1 No 0 15/1/2/211 433 2.0000rm 0,0 -1 No 0 15/1/2/211 433 2.0000rm 1 0,0 -1 No 0 15/1/2/211 434 2.0000rm 1 0 -1 <	Gaz T. Gaz T. 30 02 30 N2 30 N2
Current reference Reference G_P3 Designation Set Command Command Command	Family Vis Priody/0 Pat in stock Vis Date [16/10/2012	
Matter Type Thickness 2.0000mm	Turret Cut gaz	
Occurence number 1 Quantity wanted 1 Dptimized Quantity maximum per sheet	Laeometry document (Existing) CVDossiers_Patricia\ANNEE_2012\TESTS_614_	Box dimensions × 906.3816mm Y 427.2606mm
Order name Nest root der name Nesting, Test Saving directory. Sorting additional criterions for automatic placement References Designations Set Plane Use completion parts Ves No	C'Dossiers_Particia/WNNEE_2012/TESTS_614_109/LANCEMENT\ Commands @ Customers Pamilies Dates @ Cut gaz Nest parts by chronologic dates	Turing: Linked actions for nesting Production magement yetem [import from PMS] Export to PMS] Cleat tasks done
Execute Order	Create opcard before nest Validate and Close	Cancel

Manual creation function

In the **Simulation** pop-up menu, a new function can be used to create and/or print a .dft file of labels.

Function name: Create labels

Function procedure: Open a nesting document.

6 To	pSolid (alpha) by N	lissler Software	- [Nest : C:\Dos:	iers_Patricia\AN	INEE_2012\DEMOS\	PIECES_IMBRIC\J	Imbricatio	on\Nesting_1#01.meg	< <curren< th=""><th>t>> (Associative r</th><th>node]</th><th></th></curren<>	t>> (Associative r	node]	
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] 😽 🖿 I	L 🚺 🗖	🦔 🦉	😿 🍇 🌡		9 /	P 🌮	8				
OK I	vlode of creation of lat	oels One label pe	r part of nesting		rint at label n* 1	Cho	ose first lal	bel manually on template	PRINT	EXISING LABELS F	ILE	
The combo bo	νx can be ι	ised to s	elect the	label cr	eation mod	de.			One la One lal One lal	bel per part bel per part o bel per kind i	of nesting of nesting	•
									oneia	ber per kind	part of fiesti	9
	Start to print at la	abel n° 1		Choose firs	t label manually o	n template)	PRINT	FEXISING LABEL	5 FILE)			
The follow used to ir	wing field ndicate th	l can l e numb	be er	This to d	button ca irectly sele	in be us ect the fi	ed rst					

The use and the location of the first label. This field is for informational purposes and can be input using the keyboard.

label manually on the template.



When the user generates another label file, the system automatically proposes starting with label no. 8.



As soon as the label file has been created, the user can view and/or print it.

QUIT VIEW LABELS DOCUMENT PRINT LABELS DOCUMENT

If the function is called up when the current document is blank, the function only proposes printing an existing label file.

Creating label files on the fly during nesting

A new action linked with the tasks manager can be used to create label files on the fly during nesting.

Perts to machine Matters to use Nestings done Orders done Image: Parts done Des. Set. Plane Com. Customer Mat. M. Th. Occ. Q. Q.Mx. Fam. Comp. Pilo. Date Mat. Image: Parts done 7421936052.Corps de pi_ce.3 CDE_4040 Aluminium 2.0000mm 1 1 0 -1 No 0 15/10/2012 t300.	Gaz. T 30 0.2 30 0.2 30 0.2 30 0.2 30 0.2 30 0.2 30 0.2 30 0.2 30 0.2 30 0.2 30 0.2
Ref. Des. Set. Plane Com. Customer Mat. M. Th. Occ. Q. Q.Mx Fam. Comp. Prio. Date Mac. 1 7421936052;Corps de pi_ce.3 CDE_2020 Aluminium 4.0000mm 1 1 0 -1 No 0 15/10/2012 t300 1 7421936056;Corps principal CDE_4040 Aluminium 2.0000mm 1 1 0 -1 No 0 15/10/2012 t300	Gaz. T 30 02 30 N2 30 02 30 N2 30 N2 3610 0
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Set Plane Part in stock Yes	
Command (CDE_2020 Date)2E30201	
Customer Linked actions	
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Thickness 4.0000mm	
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Nest root order name Nesting_Test Saving directory C:\Do Generate common cuttings trajectories 🕑 Denerate labels nest nest nest nest nest nest nest nes	
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Oring double at circles to double parcental Plane Come Put micro attachs Cose generated documents Import from PMS Export to PMS The Put micro attachs	3
Use completion parts	2
Yes No No No Concrete reasonance Clear tasks done	
□ derenae programs V Pats colouring	
Execute Order	
Validate UK Lance Cancel	

According to the creation mode required in **Tools | Options**:

- Either a single file containing all the labels for the various types of order parts will be created once all nestings have been made.
- Or a file containing all the nesting labels will be created (as the nestings progress).

Creation | Visualization | Print from the Tasks Manager.

Two new icons are available in the Orders done tab.

The first creation icon can also be used to recreate labels in the event that the nestings are subsequently modified (add/ move a part from one nesting to another, etc.).

The action depends on the focus position in the dialog box:

- If the focus is located in the **Orders done** zone, the function will create a file of labels that corresponds to **one label per kind part of order**.
- If the focus is located in the **Composition of the order** zone, the function will create as many

label files as the number of selected order lines.

Jrder hame	0	Matter	M. Th.	Different sheets	Nbr. Full shee	ts Nbr. Global wa	te Parts nb.	Machine	Gaz Time	Date	Files path	
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mposition of th	the order											
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TopSolid 2013

If the label files already exist, it will be possible to override a creation of new labels.

All the label files requested are displayed on the screen in tile format.

The **Print labels** function prints the label file(s) that correspond to the focus position in the dialog box. On the fly printing will only be initiated if all label files have been created previously.

Labels creation	23
Recreate label	s files
One label per	part of nesting
🔘 One label per	kind part of nesting
04	Cancel

Direct import TopSolid 7 - TopSolid V6

Purpose of the function	Configuration General parameters
	🖞 User information
	e Colors Desaw inactin accord
Direct link between TopSolid V7 and	General Turret
	importation Automatic tool mount on turret
1 opSolid v6.	Standard turret Browse
	the Uther options
	Multidocs Dx4r/DWG/IGES Default thick 1.0000mm Default thate Acier
Triggering the function	t ser runch
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Tools Options Importation	Tasks manager
General narameters	Chapter and the set of
General parameters	
	Copy general information from the TopSolid document
	From unbend
Modifying these parameters	[v] Add part to tasknanager
requires restarting TenColid	Basity part machining document
requires restarting ropsolid.	Close part machining document
	Ask the questions (to choose transformation the turnet)
	Ignore modifications of Name of Matter of the 3D part during upadies of the punch part
	Bill of material file
	Designation of the names of columns
	INUM INUMBHE ICHEMIN
	(2) imports or unrole parts v/
	Exchange folder [C:\Projets\Mise à plat v7] Explorer II use direct link
	OK Cancel

<u>SMI link</u>

The **SMI** link has been improved. In previous versions and during document upgrades, V7 was intended to export the TopSMI document with the same name and in the same directory as for the initial import.

In version 6.14 and on, using the **exchange directory**, TopSolid V6 finds the TopSMI document in order to update the machining part, even if the name of the exchange document is not the same as for the initial import.

Direct link

When TopSolid V7.7 is installed on the same PC as TopSolid V6.14, the direct link can be used without having to navigate through TopSMI unfold parts.

· · · · · · · · · · · · · · · · · · ·		1
Exchange folder C:\Projets\Mise à plat v7	Explorer	🔽 Use direct link
· · · · ·	10	
I OK	Cancel	

Box available in **Tools | Options**.

Method

In version 7, unbendings must be checked in.

- In a TopSolid'SheetMetal **Part** document, use the **New part** function
- Select either the V7 Explorer button or the Current V7 Project button.

TopSolid 2013

<u>V7 Explorer</u>

In the event that TopSolid V7 is not loaded, it will be loaded in memory and the project explorer can be used to select the part to be imported.

Only files that contain unbend parts ready to be imported in V6 appear in TopSolid7 explorer.

- To switch projects, use the **Choose Project** button.
- Continue the import by selecting the machine.

TOPSOLID 7 EXPLORER 23 Project Nouveautés_2013 Pièce 140.A.0 Pièce 145.A.0 Material : Thickness : Choose project Destination directory C.\Dossiers_Patricia\ANNEE_2012\DEMOS\REX0\test Import sub directories Reproduce project tree OK Cancel

Current V7 project

Current V7 Project explorer only displays the file that contains a current unbend part in V7. This file must have been checked in beforehand in V7 to be used in CAM V6.



• Continue the import by selecting the machine.



The **Green/Red** light can be used to indicate, in real time, if the V7 part has undergone changes. If the light turns **Red**, click on the light to update the PCH part using the V7 unbending.

The light turns green.

The *real ison indicates a connection to TopSolid V7 (= TopSolid V7 activated).*

Working offline is possible (= TopSolid V7 deactived) by clicking on the 💥 icon.

The update status is temporarily unknown

Offline mode makes the import options unavailable from V7.

Importing multi-docs (using the direct link)

From the **Miscellaneous** | Import | Multi-documents | Top-→Pch menu, use the V7 Explorer button.



The project explorer can be used to select all the V7 unbendings in a file and to view all the unbendings for an entire project.



User interface

Previews

In the **Nesting Order** dialog boxes as well as in the **Tasks Manager** dialog boxes, there is the option of clicking on the Part or Nesting preview, which temporarily enlarges the view in a dialog box.

The scroll wheel can be used to zoom.



Missler Software

Sheet flatness

TopSolid 2013

Edit function

•

•

Modifying sheet flatness

When defining the sheet formats in materials management, it is possible to indicate a given state of flatness for each format (associated with a material and a thickness).

This state can be reworked once nesting is done.

When editing a nesting and if the format size

changes, the flatness state is updated according to the new dimensions.

Use the wrench to select the format on the screen.

Likewise when a change has been performed using the wrench with the Format dimensions option.

For non-standard formats that are not defined in format management, a default state shall be entered in **Tools** Options.

> Pland Pland

> > eformed

Conterrentied		
- All Camples netting - All Splitting	Collection of the chace of sheets	Optimization of the choice of

Sheets list1mm			
Length (mm) Width (mm)	2000	2000.00×1000.00 2500.00×1250.00 3000.00×1500.00	-
Price (Euros)	1		
Provisioning a	uantity 1		-
- Sheet state -			
C Flat		C Little disto	red
Distored		C Very disto	red
Border cut (n	nm)		
Left	0	Bottom	0
Right	0	Тор	0
Ok	Cancel	Remove	Add

NESTING ORDER

Expand the combo box, select Sheet state . Expand the combo box, select the new state and cl	IICK OK. NESTING ORDER DIMENSIONS OF SHEET INFORMATIONS DISPLAY SHEET MACHINED NUMBER PLANAR SHEET
en editing a nesting and if the format size nges, the flatness state is updated according to new dimensions. wise when a change has been performed using wrench with the Format dimensions option.	TopSolid (beta) by Missler Software - [Nest : C:\Dossiers_Patricia\NOUVEAUT File Manage Edit Nesting Splitting Attachs Punching Loade Paratricia Paratricia

SheetMaster

Dynamic unloading adjustment

When unloading a part or a waste with SheetMaster, it is possible to provide a delta value position gap (in relation to the punch) along the X axis (or along the X and Y axes depending on the machine).

Punch - TopSolid'Punch

Punch using the marking tool

If using the font *lettering-Punch-RC.top*, the text is broken down into small rectangles with dimensions provided according to the required text height.

If using automatic allocation, in **Tools | Options** it is possible to mark texts and the lettering and to use the rectangle tool in the **Marking** category.

Therefore, this tool could be used on other geometries of the part that do not depend on the **Lettering** function.

onriguration	Adjustments
User information	Ignore and induce segments of length lower than 1.0000mm Circles of diameters (separed by ; 13mm ; 4mm ; 5mm
Shortcut kev	Notching
General	No creation of notch but allocate each line
Importation	Limit at box triangular notch using rectangular/square tool
Punch	Tool name to use by default for notches decomposed into lines None refor coma5610
Allocation	Lines Circles
Automatic allocation	Allocate
 Adjustments 	None All All except on box Max. tool width 5.0000mm Max. tool diameter 5.0000mm Max. tool diameter 5.0000mm
- Tools order	Repeats search
Machining associated to the next	Lines search Grids search Circles search
Lut	Minimum number of holes for lines 3 for grids 4 for circles 3
Nesting	Altern the start point of parallel lines
Simulation	Recognization of TopSolid repetitions
Post processor Taska manager	Other searches
Display options	Macros search No search Keep macros in data base
	Special tools on external lines Special tools on external arcs Special tools on internal circles
	Search machinings of type line to line - Mini. number of lines to put in relation 2 - Gap maxi. between 2 lines 20.0000mm
	Trapp door evacuation
	Waster Dimensions : X mini 100 0000mm X mini 100 0000mm Put Stop it trapp too small
	coma5610 V Parts Dimensions : X mini 100.0000mm Y mini 100.0000mm V Put Stop if trapp too small
	coma3610 V Put Stop if trapp too small Search parameters
	Looma5610
	costradB610 Implementations: X minin[100.0000mm] Y minin[100.0000mm] Implementations: X minin[100.0000mm] Search parameters Barunchadanter of a punched arc[50.0000mm] Maximum dameter of a punched full circle/50.0000mm More than this value, the full circle will be nibbled with [2000 • •
	comaBEI0
	Looma/BEI0 ♥ 2 Parts Dimensions : X min[100.0000mm Y min]100.0000mm 2 Put Stop if trapp too small Search parameters Maximum denater of a punched arc[50.0000mm More than this value, the full cricle will be nibbled with 3000 ♥ Maximum overflarking for notching[6.0000mm for coma5EI0 ♥ 7 Machima an obforce with a calded tool at a certanoular tool for [coma5EI0 ♥
	LossaBEI0 Image: Parts Dimensions: X mini[100.0000mm Y mini[100.0000mm Image: Parts Search parameters Baunched arc[50.0000mm Monit Maneter of a punched arc[50.0000mm Monit Maneter of a punched arc[50.0000mm Maximum dameter of a punched arc[50.0000mm Mone than this value, the full circle will be nibbled with [2000 • Maximum dameter of a punched arc[50.0000mm More than this value, the full circle will be nibbled with [2000 • Maximum overflarking for notching[5 Image: State of the st
	Looma5610 ♥ 2 Parts Dimensions : X min[100.0000mm Y min][100.0000mm 2 Par Stop it tapps too small Search parameters Maximum dualertor of a punched arc [50.0000mm More than this value, the full circle will be nibbid with] 3000 ♥ Maximum overflarking for notching[6 0000mm for cond8510 ♥ 27 Mark tents and Nettring for cond8510 ♥
	LossaB610
	coma5610
	comatibility
	Contable[10
	Lossad5610 Impact Parts Dimensions: X min[100.0000mm Y min[100.00000mm

In version V6.14, even if there are two tools with the exact same dimensions: one classified **Normal** and the other classified **Marking**, automatic machining will exclusively select **marking** for lettering, and the **normal** type tool for punched geometries.

The font characteristics must match the tool dimensions exactly.



Strategy Changes

Improved preservation of tool order

Significant improvements have been made to better preserve changes in tool order carried out in the **Order of operations** dialog box during updates and when deleting nesting parts.

A new check box in the **Options** tab of the strategies dialog box can be used locally to return to the default order without completely resetting the strategies.

Also, the system now alerts when the previously made changes cannot be kept.

When a nesting changes and there are more zones than before (offsets), it is possible to apply the operation order changes carried out in zone 1 to the new zones.

Simplified strategies

A new option makes it possible to use simplified strategies.

By default, this mode only uses the **Internal cuttings** and **Punchings and Evacuations** categories. This makes it possible, among other things, to limit the

Application configuration			
Configuration	Sof of tools Tools categories manage Use simplified strategies Pass totaly a tool from notching category to splitting category	y when it belongs to the two categories	
Cut Cut Simulation Simulation Solution Cut recovery Solution Cut recovery Cut	[2] Define the order of tools in simulation Tools order	Circular Square Obining Disting Trapozotid Trapozotid Special loots Rectangular of simulation	B 5

tool changes when a single tool is used on both internal and external contours.

It is possible to move a tool to the **Notching and contours** category or to the **Splitting** category at any time using the **CHANGE OF CATEGORIES** button in the dialog box **Choice of strategies**.

When you reset the strategies (using the **Reinitialization** button in the dialog box) or when you reset the simulation (using the player button), it may or may not be possible to use simplified strategies.

WARNING	i	X
	Do you really war	nt to reinitialize the strategies ?
	🔲 Use simp	olified strategies
	Yes	No

Order of tool shapes

The order of tools according to "shape" is now memorized in each nesting. It appears in the choice of strategies dialog box.

e of strategies			
ategies Options Prototyp parts			
	Strategies loading Current strategy : D Strategies	Strategies saving oubleBlade_HG loading Strategies saving	
trategies Internal punching			
Internal punches	De la Falle dura	Full punches and evacuations	
Tools Strategies	Strategy : Entire sheet	Tools Strategies	
D10 Entire sheet			
C 10x10 Entire sheet			
	Traject on the sheet		
APPLY TO ALL TOOLS OF ALL CAT	EGORIES MODIFY	APPLY TO ALL TOOLS OF ALL CATE	GORIES
xternal punching Notches and contour Tools Strategies	Strategy : Entire sheet	Splitting Tools Strategies P.20-5 Entire depart	Strategy : Entire sheet
R 10x5 Entire sheet	_	R 76.2x5 Entire sheet	Verticals Horizontals
	Traject on the sheet	•	
APPLY TO ALL TOOLS OF ALL CATE(GORIES MODIFY	APPLY TO ALL TOOLS OF ALL CATEGORI	IES MODIFY
	TOOLS C	ATEGORIES MODIFICATION	
Separate orientations			Tools order
Tools Pos	ts Categories	Orientations	Square
□ R 76.2x5 Pos	t 17 Splitting	0.000*, 90.000*	Oblong Calked end Trapezoid ~ Modify
	arat	ОК	Carcel

To change the order of the shapes, click the **Modify** button.

Reminder: Tools classified as **splitters** do not take this order into account and will always be placed at the end of the list.

The default order at the creation of the first simulation is always defined in Tools | Options in the Sort of tools section.

onfiguration de l'application	1 1 1 1 1 1 1 1		
Configuration	Tri des outils Gestion des catégories d'outils		
Couleurs Accourci clavier	V Utiliser les stratégies simplifiées		
Général	Faire passer totalement un outil de la catégorie "Grugeag	es et contour" dans la catégorie "Ref	endage" lorsque cet outil appartient aux 2 catégorie
🖶 🐺 Poinçonnage 🖶 💑 Découpe			
 47 Placement 48 Simulation 	Définir fordre de passage des outils en simulation		
Trajectoire Trajectoire Tridectoire Fiches opérateurs Exquettes	Onder des aufle	Ronds Canés Oblongs Queues de carpe Trapezes	+
Geregene Geregene	Cicle des come	Triangles Outils spéciaux Rectangles	•
Gestionnaire des travaux			

Change in appearance of the choice of strategies dialog box

In order to improve the overall legibility of the tool order, categories can be positioned vertically in the choice of strategies dialog box.

Config

Internal punch	ing hes	Circlane - Entire sharet				
Tools	Strategies	Strategy : Envire ander				
D10	Entire sheet					
C 10x10	Entire sheet					
		Traject on the sheet				
Campany and				Stra	tegies loading	
APPLT I	UALL TUULS OF ALL LATEG	MODIFY		Curre	nt strategy : DoubleBlade_HG	
Full punches	and evacuations				Strategies loading	
Tools	Strategies				(and greatering)	
	course of the second se				Strategies saving	
					Strategies saving	
APPLY TO	D ALL TOOLS OF ALL CATEGI	DRIES				
			Separate orientations			
sternal punch	ing		Tools	Posts	Categories	Orientations
Notches and	contour	Course Falling shared	□ R 76.2x5	Post 17	Splitting	0.000*, 90.000*
Tools	Strategies	strategy : critile sheet				
	Entire sheet					
1.81	Entire sheet	0.000				
R 10x5	CLUMP STREET		1			
R 10x5	Cland states	Turindan Na dan d				
R 10x5	Linke steet	Traject on the sheet			Tools order	
APPLY TO	D ALL TOOLS OF ALL CATEGO	Traject on the sheet			Tools order Circular	
APPLY TO	D ALL TOOLS OF ALL CATEGO	Traject on the sheet			Tools order Circular Square Oblong	
APPLY TO Splitting	D ALL TOOLS OF ALL CATEGO	Traject on the sheet			Tools order Circular Square Oblong Calked end Trapezoid	
APPLY TI Splitting	D ALL TOOLS OF ALL CATEGO	Traject on the sheet			Tools order Circular Square Oblong Calked end Trapezoid Modify	
APPLY TO Splitting Tools R 30x5 R 20x5	Cristo tradet D ALL TOOLS OF ALL CATEGO Strategies Entire scheet Entire scheet	Traject on the sheet			Tools order Ciscular Square Oblong Calked end Trapezoid Modity	
APPLY TO Splitting Tools R 30x5 R 76.2x5	Chief these DALL TOOLS OF ALL CATEGO Strategies Entire sheet Entire sheet	Traject on the sheet			Tools order Circular Square Dolong Calked end Trapezoid Modily	
APPLY To APPLY To Splitting Tools R 30x5 R 76.2x5	D ALL TOOLS OF ALL CATEGO Strategies Entire sheet Entire sheet	Traject on the sheet			Tools order Circular Square Oblong Calked end Trapezoid Modify	
APPLY TO AL	D ALL TOOLS OF ALL CATEGO Strategies Errife sheet Errife sheet	Triject on the sheet			Tools order Circular Square Oblong Calked end Trapezoid Modify	

The configuration is carried out in **Tools | Options** in the **Simulation Sort of Tools** section.

Traitment of punches made in double			
Traitment mode	Detection of punches and remove them from s	simulation	•
Ask of confirmation			
Display options of simulation dialog boxes	4/1		
Put Strategies dialog box Verticaly (if screen p	ossible) : 💿 Yes	🔘 No	
At the opening of the Operations Order dialog	box, choice of the display mode of the operations tree :	🔘 Simplified display	Complete display

Trapp too small – Put a STOP

At the time of automatic allocation during punching, possibility of automatically evacuating the part or the wastes via the trapp using the dimensions provided. If the waste or the part is too big for the trapp, no evacuation was programmed. Version V6.14 makes it possible to position a Stop automatically in this case.

Notches	Lines Lines alloca	Arcs nibbling Kind of tool		cs nibbling and of tool		
Punch notches Non All All		ept on box		 Round Square or Rectangle 		
Split notches	Maximum tool with 5.0000mm		Imm Ma	— Maximum tool diameter 5.0		5.0000mm
Search patterns		Other sear	rchs		Turret	
Lines of punches		Internal macros		Use local turret Automatic with assistance		
Grids of punches Circles of punches Recognization of TopSolid repetitions		External macros Special tools on external lines				
		 Special tools on external arcs Special tools on internal circles Search machinings of type line to line 				
						Trapp door evacuation
Vastes Dimensions : X mini 100		0000mm Y mini 100.0000mm V		Put Stop if trapp too small		
Put micros on lines External contours	Distributed	Interna	al contours Distributed	E] With atta	ach tool

Configuration in **Tools | Options.**

፤ 🖍 🖱 🍛 🖾 🖄 🗡 🌍 😨	🔍 🌍 🛃 😰 🖽 🖃 📈 🗩 • 🖾 🖆 • 💊 🕀 • 🔳 • • • 🗸 •				
Application configuration	イススススト ひゃうちょう 目 二気日 長 ちょう				
Configuration User information Colors Shortcut key General Encodedition	Adjustments Ignore and induce segments of length lower than 1.0000mm Circles of diameters (separed by ;) 3mm ; 4mm ; 5mm Notching Not creation of notch but allocate each line V Limit at box triangular notch using rectangular/square tool				
Punch Tolerancies Allocation Adjustments Adjustments Tools order Machining associated to the nest Cut Machining Simulation	Tool name to use by default for notches decomposed into lines None for trumpf500 for circles Circles Allocate None All except on box Maxi. tool width 5.0000mm Maxi. tool diameter 5.0000mm Repeats search Grids search Circles search Minimum number of holes for lines All multiplication of TopSolid repetitions Recognization of TopSolid repetitions Circles Circles Circles Maxi. tool diameter 5.0000mm Maxi. tool diameter 5.0000mm Maxi. tool diameter 5.0000mm Circles search Circles search				
Trajectory Sort of tools Operator cards Labels cards Adjusts Other options Post processor Tasks manager Display options	Other searches Macros search Keep macros in data base Special tools on external lines Special tools on external arcs Special tools on internal circles Search machinings of type line to line Mini. number of lines to put in relation Gap maxi. between 2 lines Conscherence Wastes Dimensions : X mini 100.0000mm Y mini Put Stop if trapp too small Dimensions : X mini 100.0000mm Y but Stop if trapp too small 				

Graduated clamps

A new machine parameters option can be used to define the clamp graduations. By default the value is set to 0 which means that the clamps can be positioned freely on the ruler.

Clamps definition	Contraction and	3 /3 a a a a a a a a a a a a a a a a a a	A DECEMBER OF STREET, ST		
Clamps Number Number	3	Dimensions Length (mm) Heigth (mm)	80		
C Accept small clamps Discard between Long a	ind Small clamps	Rule positions Mini Position (mm) Maxi Position (mm)	150		
✓ Use small clamps To avoid tum-over		– Positions on stations – Station number	0		
General Caracteristics – Positionnement Programable C Manual	Curing punching work Fixed CAutomatic	C Left C Center C Rigth	C On Rule C On Station		
Mini Length between two clamps 250 Clamp bottom (mm) 10 Clamp graduation (mm) 100					
 ☐ Last clamps moveme ✓ Use default positions, ☐ Use default positions, 	nt BACKWARD loading in thrust loading centered	Default positio	ons, loading in thrust		
- Cut head - Accessibility	y area				
	Ok	Cancel			

When a non-zero value is indicated, and you have not **used the default clamp positions**, the system calculates the positions by balancing the clamps on the sheet and rounding to the nearest graduation.

When you move the dynamic clamp and when you click on the final position, the clamp is reset to the nearest graduation.

<u>Reminder</u>: If the graduation value is set to 0, the automatic automatic positioning of the clamps places them by rounding to the nearest mm. If the graduation value is set to 0, the dynamic manual move positions the value at the picked position, which is not rounded.

Turret-Color for unused tools

When you modify the turret of a part or nesting, the stations of the tools mounted but not used are indicated in orange.
Automatic punching - Micro joints

When automatic machining during punching, it is possible to position the micro joints distributed on the internal contour lines.

Settings in Tools | Options.

lion	Adjustments	
ormation	Ignore and induce segments of length lower than 1.0000mm Circles of diameters (separed by ;) 3mm ; 4mm ; 5mm	
key	Notching	
on	☑ Limit at box triangular notch using rectangular/square tool	
cies	Tool name to use by default for notches decomposed into lines None 💌 for trump/500	-
on tic allocation	Lines Circles Allocate Maria tradición (E. 0000	
ustments	None All All except on box Maxi. tool width 5.0000mm Maxi. tool diameter 5.0000mm	
is order iount ng associated to the nest	Repeats search Lines search Circles search Circles search	
	Minimum number of holes for lines 3 for grids 4 for circles 3 Aftern the start point of parallel lines	
n Cessor	Recognization of TopSolid repetitions	
nager	Uther searches Macros search No search	
otions	Special tools on external lines	
	Search machinings of type line to line · Mini number of lines to put in relation 2 · Gap maxi, between 2 lines 20,000	
	Trapp door evacuation Trapp door evacuation Vastes Dimensions: X mini 100.0000mm Y mini 100.0000mm Put Stop if trapp too sm trumpf500 Vastes Dimensions: X mini 100.0000mm Y mini 100.0000mm Put Stop if trapp too sm	nall all
	Search parameters	
	Maximum diameter of a punched arc 50.0000mm	
	Maximum diameter of a punched full circle 50.0000mm More than this value, the full circle will be nibbled with 3.000	•
	Maximum overflanking for notching 6.0000mm for trump/500	•
	Machining an oblong with a circular tool and a rectangular tool for trumpf500	•
	Mark texts and lettering for trumpf500	-
	Put micros on lines External contours Distributed Internal contours With attach tool	
	Dangerous wastes detection	
	Dangerous wastes detection Violatic dangerous wastes detection after the automatic affectation in punch document	

During automatic machining, this option is available locally.

Notches Lines Punch notches O None All All excep		tion Arcs nibbling Kind of tool		or Rectangle		
	Maximum too	l with 5.0000mm	Maximum (or	ol diameter	15.0000mm	
Search patterns Lines of punches Grids of punches Circles of punches Recognization of TopSolid repetitions		Other searchs Internal mac External mac Special tool: Special tool: Special tool: Search mac	ros cros s on external lines s on external arcs s on internal circles hinings of type line to	Turret Use Auto	Turret Use local turret Automatic with assistance e	
Trapp door evacuatior Wastes Dimen Parts Dimensi Put micros on lines External contours) sions : × mini∏00 ons : × mini∏100.0	.0000mm Y 000mm Y n Internal co	mini 100.0000mm nini 100.0000mm ntours	Put Stop	if trapp too small f trapp too small	
On corners	Distributed	Distril	outed	With att	ach tool	



Settings for the number of micro joints according to the length of the line is carried out in micro joint management.

Misro impliance uphras by mother and ticknoss		
acier Thickness: 1.5000mm	1.5000mm 0.2000mm 0.2000mm	Add Distributed micro attachs Length Number 112.0000mm 1 300.0000mm 2
Default positioning on part	Positions of micro attachs during Autor	natic Splitting
On cornerIntermediate	Default intermediate micro junctions nu Line dimension (mm) Nu Small 5.0000mm 1 Medium 5.0000mm 1 Bin 5.0000mm	umber by line length umber of m-junctions
On both	5.0000mm [1	ancel

Micro joints are only distributed if the item does not already have micro joints at the start of automatic machining.

Special tools on points along profile

The function can be used to place a tool on along profile, along a profile that has been expanded for the use of special tools.

Option: Along Segment

TOOL=OVAL SAME TOOL AS DISTRIBUTED	EXTREMITY SEGMENT (with collision control)	ALONG SEGMENT
------------------------------------	--	---------------

- After having selected the tool, click geometry and then select the tool segment that will slide on the geometry.
- Click the final position.

Option: Extremity Segment (with collis	sion control)	
TOOL=OVAL SAME TOOL AS DISTRIBUTED	D] EXTREMITY SEGMENT (with collision co	ontrol

- After having selected the tool, click on the geometry nearest the desired extremity and then select the tool segment that will be used.
- Then click **OK** to confirm.

In the event that the geometry has been modified, the positioning remains associative.

Induced machining (automatic punching)

During automatic allocation during punching, it is possible to ignore segments whose length is less than a certain size by setting them induced.

Settings in **Tools | Options.**

Janore and induce seaments of length lower than 1.0000mm	
	Circles of diameters (separed by ;) 3mm ; 4mm ; 5mm
Notching	
Limit at box triangular notch using rectangular/square tool Tool name to use by default for notches decomposed into lines None	← for trumpf500
Lines Allocate None All All except on box Maxi. tool width 5.00	Circles 000mm Maxi. tool diameter 5.0000mm
	Notioning No creation of notch but allocate each line V Limit at box triangular notch using rectangular/square tool Tool name to use by default for notches decomposed into lines Lines Allocate None All C All cxcept on box Maxi. tool width 5.0

If the value is set to 0, all segments will be machined.

Likewise, it is possible to define the values of the circle diameters that will be ignored and set induced (separate each value using the ";" character)

Simulate the tool change before repositioning (with head) if deformation

On TRUMPF machines that do not have the jack tightening option, you are required to tighten the sheet using the head when repositioning. However, if the tool in the head is mounted on the multitool or even if this tool is a special, not "normal" tool (e.g. deformation), it is necessary to mount an adequate tool in the head before repositioning.

In this case, the machining simulation is now compliant.

Automatic Splitting - Micro joints

The function can be used to create automatic splitting that now makes it possible to take into acount micro joints already present on parts.

New Without evacuation option from the function:

When launching the automatic	Splitter parameters	<u> </u>
splitting function, a new button	General Splitters Micro joints	
WITHOUT EVACUATION appears at		
the end of the line.	Stop on geometry ✓ On left side ✓ On right side	Borders to punch
This option opens the dialog box that can only be used to select the edges to be punched and the tools.	 ✓ On top side ✓ On bottom side ✓ On all sides 	V Right V Top V Bottom
	Save OK	Cancel

No evacuation or micro joint will be added.

Splitter para	ameters Splitters Micro	joints				X	
-Vertical :	splitters e blade	Common blade	Horizon Oub	al splitters le blade	🔘 Common bla	de	
Left	R 76.2x5	RECTANGULAR - Width 5.0000m	n Top	R 76.2x5	RECTANGULAR 👻	Width 5.0000mm	
Right	R 76.2x5	RECTANGULAR - Width 5.0000m	n Bottom	R 76.2x5	RECTANGULAR 👻	Width 5.0000mm	
Common	R 76.2x5	RECTANGULAR - Width 5.0000m	m Common	R 76.2x5	RECTANGULAR -	Width 5.0000mm	
🛛 🛛 Autom	natic choice		🔽 Auton	natic choice		, .	
Overflankings Image: Construction of the second s							
		Save	OK		Cancel		

Possibility of using standard functions to locally add, modify and delete micro joints on the splitting lines created. You can also add micro joints manually, or distributed on item. Picking can be carried out on the splitting line created or even directly on the part geometry.

<u>**Restriction**</u>: Distribution options on contour, positioning in corners and searching for part corners are not possible if the automatic splitting lines were created previously.

The **machining** or **part** propagation function can be used to transfer micro joints of the splitting lines of a part to the splitting lines of other similar parts of the nesting.

If the micro joints were positioned on the parts prior to creating the automatic splitting with the **without** evacuation option, these micro joints will be taken into account.

It will be possible to modify the size locally, to add, delete and propagate these micro joints.

Micro joints option of the function:

This option changes the behavior if micro joints are already present on the parts on which you wish to create automatic splitting.

TopSolid 2013

By default, the micro joints present on the parts are going to be added to the new micro joints requested (in corner and/or intermediary).

A check box can be used to add new micro joints on the lines that already have them.

Cut - TopSolid'Cut

Fly cut path

Fly cut paths can be performed on certain next-generation TRUMPF machines in the thin sheet and on even cuts that are close together.

E.g. grids of rectangular holes.

These cuts can be made without lead in, the cut tool path does not operate on the full contour but passes from one contour to another.

It is essential to have defined the **FlyCut** cutting parameter beforehand.

Access to the **Cutting** menu is available in the Part document.

Two modes are available:

Manual mode:

 Click on a line or a full circ (exclude the arcs).

nvert= No Tangence length link after cutting 1	Tangence length link before next cutting 2	Select a line	
51	J		
			_
Ja ()

When you have selected a line, it is possible to indicate a tangence length link after cutting and before the next cutting, when there is a change in direction.

13030

0.0000mm

acier

TAB-TECNO METHODE

ers Lead in/out standard Lead in/out common cutting

5104

5104

5104

5104

ST010MD0

-

10

Cuttion Gaz 02

The **Invert** Yes/No button makes it possible to invert the direction of the tool path when the initial geometry does not turn in the desired direction.

Splitter parameters	X
General Splitters Micro joints	
Micro junctions values by matter and tickr acier Thickness: [1.5000mm Corner m-junctions width: [0.2000mm	ness 1.5000mm 0.2000mm 0.2000mm Add Modify Delete
Default positioning on part On corner Intermediate On both	Default intermediate micro junctions number by line length Line dimension (mm) Small [5.0000mm Big 5.0000mm
Do not add n	new micros tabs on lines that have still micros tabs



Copy One...

Copy AlL.

0.5

0.5

0.5

TYPE-PENE TYPE-COUPE VITESSE

300

100

500

100

3.00mm

ve dillings manag

Copy line...

Add

Rem



Example of circles

Automatic mode:

Automatic mode operates via a strict selection of lines or full circles.



During simulation, possibility of not viewing the links between cuttings.

Simulation

Order . Subroutines. Adjusts.. Prototypes.. n 🖪 ΣS Simulation Configuration Visualization mode Information displayed 💿 normal 🔘 None 🔘 Step by step Minimum When the machine allows, Sequence by Sequence 🔘 Full post-processor installation is Timer 0 🖓 200 0 Corresponding speed Maximum Slowly 🔲 Show path 📝 Show cut links Show Turret / tools set or Head 0K Cancel

23

necessary.

TopSolid'SheetMetal 2013: What's New

Settings in **Tools | Options**.

Configuration	Priming/Exit			
User information	Preview of leadin/leadout in	catalog		
Shortcut key	Preview of leadins/leadouts i	n fit rules dialog bo	ж	
Importation	Check unsafe of priming/exit	s into matter		
Punch Cut	Make safe priming/exits out of Standard priming/exits	matter		
General	Rotate of gap angle	30.000*	🔽 Change vari	ant to reduce size
	(Unly for type line)			
Insertion elements	Priming/exits on attachs	30.000*	🔽 Change vari	ant to reduce size
Machining of the part	(Unly for type line)			
Micro joint	Fit on point on curve defined	in design docume	nt	
	Parameters depending of mach	hins		_
Simulation	for		13030	-
 Post processor Tasks manager 	Configuration of priming/exit	🔲 with exit		
Display options	Priming full line	Exit full line		
	Model	_ Change] Model Variant	Change
	Priming on corner		Exit on corner	
	Model	- Chavar	Model	Change
	Variant	Unange	Variant	Lhange
	Priming on attachs		Exit on attachs	0
	Model	_ Change	Model	Change
	Variant		Variant	
	Finning on opened curve		Fit exit retracted	
	Length 0.0000mm		Length 0.0000mm	
	Strong thickness lead in Automatic	change of param	eters on given segment of I	ead in
	Index of segment (U = use of p	arameters in tables	JIU	
	Hy cut of lines on direction ch Tangence length link after cutt	ianges ting 1.0000mm		
	Tangenee length ant diter out			

Lead in tracks –Saving Format as waste

In the **Extract waste from format** function, possibility of saving the format as waste in order to reuse this format for a future nesting. In the previous versions, only the core/cavity of the external contour parts were memorized and it is possible, during the next nesting, that the parts were positioned where the previously positioned part lead ins were. This happened when the size of the lead in was greater than the gap between the parts.

Example of nesting to be saved as waste:



Cutting parameter access

When you have an empty part document or nesting document on the screen, it is possible to call up the **Cutting parameters management** function from the **Management** Menu. Select the machine as well as the cut gaz (associated with the machine) whose parameters you would like to input or edit.

TopSolid (beta	a) by Missler	Software -	[Punch/Cu	t : Docum	ent5 < <c< th=""><th>urrent>> (A</th><th>ssociati</th><th>ve mod</th><th>e)]</th><th></th><th>-</th></c<>	urrent>> (A	ssociati	ve mod	e)]		-
File Mar	nage Edit	Punching	Attachs	Loader/l	Jnloader	Miscellane	ous T	ools l	Vesting	Cutti	ng
🗋 📩 🔗	-	i 🖍 🕯	n 🤤) 🗙	ちい	3	9	3		
s 🛐 🧕	;	<u>b</u> <u>b</u>	8 🌫	1	🦻 💽	6		1	8	-	y
]											
Select n	nachine and	cutting gaz		×							
Mach Amad	ine a F1		Used gaz								
alpha fo301 lc301 ti3030 trumpf	3 5 5 600		Non specif	ied gaz							
	OK		Cancel								

Move a lead in when adding micro joints on a cutting path

When you add micro joints on a previously machined contour, it is now possible to automatically reset the lead in/lead out on the first micro joint of the contour.



Configuration in **Tools | Options**.

Image: Status Micro pint Image: Status Put micros on extend contours Image: Status Put on enticon on each contour to prevent swing Image: Status Put on enticon on each contour to prevent swing Image: Status Put on enticon on each contour to prevent swing Image: Status Put on enticon on each contour to prevent swing Image: Status Put on enticon on each contour to prevent swing Image: Status Put on enticon on each contour Status Post processor Image: Status	Application configuration	STREET, ALLER DE	2 4 20
Cut parameters Fut one micro on each contour to prevent swing Cuting - Mathing - Burning Machining associated to the next Machining asociated to the next Machining associa	Configuration	Micro joint Put micros on external contours On corners	Distributed
Other contour At center of arc At center of line At center of line	 General Cut parameters Priming/Exit Insetion elements Cutting - Marking - Burning Machining of the patt Machining associated to the nest Maching associated to the nest Table cutting Simulation Post processor Task manager Display options 	Put one micro on each contour to prevent swing Put on internal contour Size greater than 30.0000mm Circular contour Rectangular contour At center of small side At center of wide side Oblong contour At center of arc At center of line	Put on external contour Size greater than [0.0000mm Size lower than [300.0000mm Put micro on corner of 2 lines at 90" Tolerancy on corner angle [45.000" Length mini of the 2 lines [5.0000mm Put micro on arc Length of arc greater than [4.0000mm Length lower than [35.0000mm
DK Cancel		Other contour At center of arc At center of line When adding micro tabs on contours, put the leadin/out on the first micro tab	

Cutting head up-down times

In the aim of fine-tuning cutting machining times, possibility of configuring the time for the cutting head to go down and up.

Configuration is carried out in the Management | Machines menu in the Cutting head section.

Circular Head Length (mm) 100	- Width (mr	n) [100	Along X (mm)	o / Punch Head
Heads number 1 Step mini between heads (m	е X+ С X im) 0 Асt	on Field for o	Y- ne head 0	Along Y (mm)	0
Action length along X axis (r	nm)	Mini	0	Maxi 0	
Action length along Y axis (r	nm)	Mini	0	Maxi 0	
Maximum Head Heigth (mm)	120	' Stelke eleng		0
Default Cutting path speed ((m/s)	0.15	Stoke along	Maxis (mm)	<u>1</u>
Default Linking path speed	(m/s)	0.2		r das (mm)	1º
Head time to go Down (s)		0.5	Default drilli	cut accessibility area	
Head time to go Up (s)		0.5	 Default drillin 	ng time (s)	
Options		170	Offsets		
With grid support	Step X (mm)	70	Opposite	e to matter	•
	step r (mm)	40	Head pos	ition for links	
Mini part dimension where i	no tipp off (mm)	10	Between cu	uttings inside part	E Down
Mini dimension of dangero	us loss (mm)	0	Between pi	arts #ocho	E Down
Ontic reference device	- I col type	C Saucro	Dofeult hoir	ath (mm)	45
		< Square	- Deldar nei	gar (mm)	113
Optic device Time	Dimension (i	nm) 20	Check d	langerous path Head (Jp
Gaz management			Safety heig	ght for path (mm)	0
02 ^ 02	Add	Remove	Burning		
N2 * 1	81		Default heir	qth (mm)	0
Centering enable	· .		Default pos	sition 🔽	Down
Swivelling head (for cham	rer)		-	- 1	

The time is displayed in an informational dialog box for the part or format.

It depends on the status of the links (head up or head down). If there is not a link between two cuts, the head is considered to go up and down between the two cuts.

If the times for the head to move up and down are set to 0. In the machine parameters, the time calculated in the information box on the part will be set to 0 and the number of head up/head down movements will not be entered.

These times may appear in the DRAFT operator file of the nesting.

9	File	Manage	Edit I	unching	Attachs	Loader/Un	loader M	liscellaneo	us Tools	Nesting	Cutting	Autre	Window He
		1	• i				׼		29	3	🐑 🔟	8	K J.
150	3	8	34 8	5 💰 e	5 🖒	8		2 🖳	4	1 4	iii 🖧 1	+ -7	* 🗞 🕞
WHOL	e par	T Select a	machinin	g									
				Information	1 on punc	h cut part							7
TopSolid ' SheetMetal				C:\Dossiers General Area: Mass: Perimeter: Box dimen Number of Machining Luting Lead in Down/up I Stop Total Used gaz [sions: of micro atta g times [4,000 [0.557 [8,000 [0.000 [16,55] [2]	NOUVEAUTI 63 11 5 12 ach Total 2 8 Cuting 12 Cuting 13 Linking 13 Laed in 13 Numbe	2013/PIE 383.905mm ² 33.4234g 79.4677mm 20.0000mm ¹⁰ On nings path length path length number r of head mo	CES-2013 72,5000mr external cc	Entrée-Micr m mtour 2 [607.067 [111.307 [4 Up]4	o, pch		~	

🖉 TopSolid (beta) by Missler Software - [Punch/Cut : C:\Dossiers_Patricia\NOUVEAUTES 2013\PIECES-2013\Entrée-Micro.pch * <<current>>

Fit on point on curve in the Design document

It is possible to set the fit point on a contour by creating a **point on curve** defined in design document (.top).

• To activate this possibility, look in **Tools | Options**.



Implementation in the .TOP document:



• Use the **Tools | Point | Point on Curve** function and click on the curve. The color and type are inconsequential.

In the **Manual Contouring** function, using the automatic departure point, the system examines the point on curve in the .TOP document and, if it exists, positions the lead in in this location.

This principle applies by default to the **Automatic Contouring** and **Automatic contouring and links** functions.

If the machinings are carried out directly on the nesting, the fit point is recovered by transitivity.

 TopSolid (beta) by Missler Software

 File
 Manage
 Edit
 Punching
 Attachs
 Loader/Unloader
 Miscellaneous
 Tools
 Nesting
 Cutting
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 Window
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 Image
 Image

When the lead in has been set, it will not move unless the point on curve is moved subsequently in the .TOP document However, if machining is carried out again, the fit point will take the new position.

<u>Warning</u>: if micro joints exist, by default the lead in will be set on the point corresponding to the point on curve in the .TOP document.

Taking into account of the exit in complex nestings

In complex nestings, if you request taking into account of cut allocations, the system now takes into account the point of laser entry and exit to avoid part collisions.

Display of the quantity of symetrized parts on a separate line

Free

In the nesting document, beneath the format, are indicated the names of the parts in place with their quantities. Possibility of dissociating the parts that have a symetry in X and/or Y on a different line.

This option can be configured directly in the document display options.

- Use the wrench on the format and select **Display configuration**.
- Check the **Display quantities of symetrized parts** box.

This option is not compatible with the display of added quantities. It is therefore not available if the added quantities are requested for a given nesting.

Configuration possible from **Tools | Options.**

Tools | Options | Nesting | Nesting information display.



6 TopSolid (beta) by Missler Software - [Nest : Document10 * <<current>> (Associative mode

File Manage Edit Nesting Splitting Attachs Punching Loader/Unloader

Copy Schedulings

The **Same machining as** function in **Multiple** mode now offers the option of copying schedulings included in the group of items whose machinings you wish to copy.

Complete Rectangular Nesting

Possibility of adding parts in the rectangular nesting by "completing" it without replaying the entire nesting.

esting order - Rectangular Ne	sting			
1ode Complete nesting		Recompute	nesting	
Parts Gaps Sheets Nestin	ig General			
		Ref.	Q.	
		Point sur profil	1	
				Search
				Remove
				* +
				C
				Colouring
Box dimensions				Auto.
X 120.0000mm Y 72.5	000mm			
Current part				
Wanted 1		Done 1		÷
Priority 0		Completion pa	nt	
Rotation Family	2	572 S 2		
	PIECE 90° (2)		•	
Rotations		00.00	000	
	000* 90 000*			
Ju. Sumetries	000-30.000			
Along horizontal axis				
Along vertical axis				
latter acier T	vpe	Thickness 3.0000n	nm Macł	nine ti3030
Check parts compatibility (matter	r and thickness)			
Load	Save	ОК		Cancel

In cluster format from the Manager

When you use a format in the Tasks manager, there is the possibility of creating a **Cluster** type sub-format from a part that appears in the Manager.

After the creation of the sub-format and its validation, the part quantity of the sub-format is subtracted from the Tasks manager.

rdre de placement - Placement rectang	gulaire		
Node Compléter placement	🗖 Recalcu	ler le placement	
Pièces Écarts Formats Placement (Général		
	Réf. Pièce_ES	Q. 10	Ajouter
			Rechercher
			Supprimer
			Coloriage
Dimensions du déplié X [150.0000mm Y 200.0000mm			
Pièce courante Quantité			
Demandée 10	Réalisée 1	0	
Priorité 0	Pièce boucl	ne-trou	
Libertés	CE 90° (2)	•	
- Rotations		0000*	
			- +
Svmétries	00		
Selon l'horizontale			
Selon la verticale			
	é :		40000
tatiere jacier Lype 7 Vérifier la compatibilité des pièces (matièr	E paisseur 3.00 e et épaisseur)	UUmm Machir	ie[03030
Charger			Annular
	niegisuer	UK	Annuler

Optimize Part Quantity

Possibility of making an order from the part manager in optimized quantity. The quantity is not entered and the nesting module will provide the quantity made on the requested format.

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1 7421936052:Corps de pi_ce.3				CDE_2020		Aluminium	4.1	.0000mm	1	1		0	1 No	5	0	15/10/2012	13030	02
7421936056;Corps principal				CDE_4040		Aluminium	2.1	.0000mm	1	1	- 9	0	1 No	1	0	15/10/2012	13030	N2
2 7421936105:Corps principal				CDE_4040		Aluminium	2.1	.0000mm	1	Opt.			1 No	£	0	15/10/2012	13030	N2
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D p2						acier	21	.0000mm	1	1		0	1 No	1	0	23/12/2012	trump/500	1
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P-Marquage						acier	21	.0000mm	1	1		0	1 No	1	0	20/12/2012	enz3610	
1 B						acier	2.0	.0000mm	1	1		0	1 No		8	20/12/2012	13030	02
toto						acier	2.1	mm00000	1	1	- 31	0	1 No		0	20/12/2012	13030	02
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TopSolid 2013

This gives access to the **Optimize quantity** box in the nesting order dialog box

<u>Warning</u>: Mixing parts on a single format in optimized quantity and for parts in set quantity is not possible.

A block is carried out before starting the orders.

If you would like to launch several nestings at once with a part in optimized quantity for each one, it is necessary to check the additional grouping criterion box **References**.

TopSolid'SheetMetal 2013: What's New

	Ref.	Q.	Nesting type
	Pièce_ES		Matrix
Box dimensions ×150.0000mm			 Rectangular Nesting Complex nesting Complex nesting Colouring Auto.
urrent part Juantity /anted[1 Maxim	um in sheet	Done 0	☑ Optimize quantity
Priority to larges parts			
ority U		npletion part	_
Use holes to Nest other parts Rotation Family		Part use common cuttin	19
Totation 1 analy		*	
Rotations			
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Symetries Only around vertical axis	Auth	orized around vertical a:	xis
Maximum number of differents parts per	sheet	Ĩ,	

Cut gaz grouping criterion

The tasks manager now manages cut gaz for cutting machines.

The **Cut Gaz** field, the column and the additional grouping criterion are available in the Manager dialog box.

Ref.	Des.	Set.	Plane	Com.	Customer	Mat.	М.	Th.	Occ.	Q.	N	Q Mx.	Fam.	Comp.	Prio.	Date	Mac.	Ga
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7421936056;Corps princip.	le le			CDE_4040		Aluminium		2.0000mm	1	1		0	-1	No	0	15/10/2012	ti3030	N2
7421936105;Corps princip	le			CDE_4040		Aluminium		2.0000mm	1	Opt.			-1	No	0	15/10/2012	ti3030	N2
PIECE_STOP_MULTI_d						acier inoxydable		1.5000mm	1	1		0	-1	No	0	23/10/2012	emz3610	
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attaches						acier		2.0000mm	1	1		0	-1	No	0	24/12/2012	trumpf500	
Fly Cut						acter		3.0000mm	1	1		U	-1	No	0	26/12/2012	13030	02
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If the additional grouping criterion **Cu Gaz** is not activated, parts with differen gases may be mixed.

If the box is checked, orders will be distinguished.

To customize the cut gaz position in the order name, use **Tools | Options**.

The gas will be preceded by # G=.

	onfiguration	General
	User information	
÷	Colors	Nesting type
- B'	Shortcut key	Complex resulting
	Juneral	Order name management
	Punch	Index orders name (#X=)
÷	Cut	Start indexing at the first use of order name
	Vesting	Index the name during modification of the order
	Post processor	Format the index number Number of digits
þ.9	Tasks manager	V Order name personalisation
	General General	Available Informations Name composition
÷	Display options	#M : Matter #C : Customer #N : Matter Tune #G : Cut gaz
		#T : Thickness
		#U: Command #S: Set
		#P : Plane #P - Paference
		#D : Designation
		#F: Family
		Mort Police
		Always concatenate matter and thickness to the name of the order
		(only if order counter is not used)
		Renum nest order always enable
		Add created nestings at the top of the list
		Minimize the size of emty columns in dialog box
		Parts return to use Parts guantities changes
		🔘 Ignore parts quantities changes 🔘 Take care of parts quantities changes 🔘 Ask confirm
		Activate Automatic numbering of designations of matters to use
		Last delivered number 0 Step increase number 1
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The **Search of parts** function in the **File** menu includes the cut gaz filter.

If no filter on the machine is active, all gases used on all machines are shown in the combo box. Otherwise the list of gases is limited to those of the selected machine.

When the cut gaz is modified in the PCH part and if the part is already present in the Manager with a different gas, the system suggests updating the gas in the Manager.

Date Column of the Tasks Manager

A new **Date** column has been added in the **Nestings done** tab.

acier 1		Length	Width	Q. Loss	%) Date	Mac.	Gaz Nestin	na file
	.5000mm	1250.0000mm	1000.0000mm	4 25.9	24/10/2012	13030	02 C:\Do	ssiers Patricia\ANNEE 2012\TESTS 614 100\LANCEMENT\Nesting Test#C=MISSLER#G=0
nter 🕺 🖉	.5000mm	3000.0000mm	1500.0000mm	1 28.9	24/10/2012	83030	02 C:\Do	ssiers_Patricia\ANNEE_2012\TESTS_614_100\LANCEMENT\Nesting_Test#C=MISSLER#G=0
acier 1	.5000mm	180.4353mm	1500.0000mm	1 83.7	24/10/2012	83030	02 C:\Do	ssiers_Patricia\ANNEE_2012\TESTS_614_100\LANCEMENT\Nesting_Test#C=MISSLER#G=0
acier 2	.0000mm	2000.0000mm	1250.0000mm	7 54.1	05/11/2012	emz3610	C:\Do	ssiers_Patricia\ANNEE_2012\TESTS_614_100\LANCEMENT\Nesting_Test1#01.meg
acier 2	.0000mm	2000.0000mm	1250.0000mm	1 53.9	05/11/2012	emz3610	C:\Do	ssiers_Patricia\ANNEE_2012\TESTS_614_100\LANCEMENT\Nesting_Test1#02.meg
acier 2	.0000mm	2000.0000mm	1250.0000mm	1 58.9	05/11/2012	emz3610	C:\Do	ossiers_Patricia\ANNEE_2012\TESTS_614_100\LANCEMENT\Nesting_Test1#03.meg
						m		
ent reference	Machin	»(H3O3O		Cu	gaz 02			
tter scier	Machin Quantit	e (#3030 4		Cu	gaz[02			The second se
erent reference	Machin Quantit Loss (%	13030 4 [25.94		Cu	gaz 02			ii aa
	Machin Quantit Loss (% Machin	113030 4 25.94 ng time 14075.54	38: (3518.8859:	Cu × 4)	gaz 02			
	Machin Quantit Loss (% Machin Nesting	+ 113030 4 [25.94 ng time [14075.54 file [C:\Dossiers_]	38s (3518.8859s 3atricia/ANNEE	Cu × 4) 2012\TES	gaz 02 S_614_100\LAN	CEMENTING	esting_Test#C	

Tasks Manager Backup

The tasks manager is archived as soon as it is saved. The backups are kept and there are three of them in the file: \$PUNCHDATA\Archives_TMan.

In the event that the Manager is damaged, it is possible to restore a backup.

Example:	
Tasksman.tmn#B=-1.bak	Archive the most recent
Tasksman.tmn#B=-3.bak	Archive the oldest

To restore a backup, copy the filen\$PUNCHDATA\Archives_TMan\tasksman.tmn#B=-1.bak into \$PUNCHDATA\Archives_TMan\tasksman.tmn.

This operation must be carried out when TopSolid is not running.

Additional criteria - Search of parts

A filter has been added to the **Search of parts** function to find parts by name.

This filter meets the Windows search concept by supporting the "*" character (e.g. *P*, by default you search for all parts which comes back to *).

	Folders C:\Dossiers_Patricia\N Scan sub folders	OUVEAUTES 2013\PIECES-201	3 🔹 🦲		ets Assiers_Patricia\N	OUVEAUTE	S 2013\F	TEGESS	2013 (Sub	
		Clear list of folders			Rem	ove F	lemove al			
General information	1						1	c		
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		Part name filter (Ex: *abc	cd* ; * for all) [*] P*		Sear	:h				
sults Iame Path		D	SPCC	Mat M Th	Mass	Mac	Gaz () Far	Date	
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Additional search criteria are also available.

The criteria that you would like to see added in the dialog box are defined in **Tools | Options**.