

TopSolid v6.15 What's new

TopSolid 2014

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What's new in TopSolid'Design v6.15



This document describes the new features made to the **design** application of the **TopSolid'Design** software in the version **6.15**.

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
Visual Studio C++ Redistributable	2012
Sentinel driver	7.5.8
Sentinel RMS License Manager (floating license manager)	8.5.3

Configuration

Configuration styles

Customization of **TopSolid** has been further improved. In addition to the configurable contexts in previous versions, it is now possible to define **styles** that contain the menus, functions and contexts to be used.

This makes it possible, for example, to create a style for each application (**TopSolid'Design**, **TopSolid'Draft**, **TopSolid'Mold**,...).

As with context customization, styles can be created using the **Tools** > **Customize** function and it is possible to switch from one style to the other using the **Tools** > **Style configuration** function.

For each type of document, the last style used is saved so that it can then be automatically used when creating a new document.

Customize	×
Styles: Mechanical base New Remove	Save
Text menu Contexts Commands	
⊡ Design	
÷	
Edit	
🖶 🖤 Parameter	
Curve	=
🛓 🖉 Shape	-
Assembly	
E Kinematics	
Dynamics	
I I I I I I I I I I I I I I I I I I I	
Attribute	
Univing elements	
Elements visibility	
Layer	
Croce batching nattorn	
	-
Restore Close	

Style definition using the **Customize** function.



Style selection using the Style configuration function.

Document

Import project

The File > Import project function now proposes to open the project just imported.

Links in the tree

The new **Append links** contextual function is used to create a tab that displays the document links. The status of documents is shown using icons and symbols:

- Green padlock: The document is locked by the current user.
- Red padlock: The document is locked by another user; the name of the station that locks the document is displayed after the name.
- Yellow flag: A more recent version is available on the disk.
- Modified documents are displayed in bold and preceded by the "*" symbol.

The contextual menu allows you, among other things, to update or open documents, open the Explorer, etc.

Main	Favorite Main set Entities Layers
10	Edit
	Edit sets
	Edit boms
1000 1000	Append index
	Append presentations
	Append links
10001 14=-	Import indexes



Addition of the Links tab and example of links.

Concurrent access to documents

For users who want to work together on a same project, the **Document** > **Others** section of the **Tools** > **Options** function provides a new file locking mode: **Lock when displaying + Secure local save**.

This mode corresponds to the lock when displaying mode with no changes allowed and no possibility to save a file by overwriting a more recent version on the disk and saved by another user.

When opening a document, a blue frame around the graphical area indicates that the document is locked by another user.

Configuration TopSolid [®] Wood configuration Display Document	Others
Save-Directories Others Picking Colors Visualisation options	File lock mode
Shortcut key	Modification
Rendering options User information	Behavior Warning Banning
Parameter	Cock when displaying + Secure local save

New locking mode setting.

User interface

Tree

The unfolded nodes of the symbolic tree remain open when the tree is displayed again.

Orientation

In the status bar, the new **Ax=yes/no** mode is used to automatically reposition the view along the nearest axis. This setting is saved between two sessions.



Presentations

When creating a presentation, the **COMMENT** option allows you to associate a description with the presentation. When the presentation has been created, this comment can be entered from a contextual command.

Compass

The compass configuration information available in File > Properties has been relocated in Tools > Options.

Selection

The mode for selecting faces by color has been added to the Shape > Other shapes > Copy face, Shape > Surfacic/boolean operations > Smooth and Shape > Mechanical/other operations > Color functions.

Selection of an axis on a coordinate system created on the fly

When an axis is asked, for example when creating a coordinate system on 2 axes, the **THROUGH POINT** option allows you to specify the axis by selecting 2 points. It is now possible to create a coordinate system on the fly so that one of these axes can be selected.

Parameter

Selection of the parameter to be modified

When modifying a parameter, for example using the **Modify parameter** function, when several parameters are found on the selected element, these are available as buttons in the dialog bar. For greater clarity, these buttons now display the parameter values.

DIAMETER=12mm HEIGHT=102.5mm Modification of a cylinder's parameters.

Expressions

The **WHEN**, **OR** and **AND** functions have been optimized to calculate only the necessary expressions. For example: WHEN(length==0,0,100/length)

- if length = 0, the 2nd part of the expression (100/length) is not calculated, which in this case avoids a division by zero.

Contour

Horizontal or vertical link

During contour creation, after entering a through point, the **Link=AXES (Z)** option allows you to get a segment parallel to one of the coordinate system's axes. This option is now available by indicating a starting point, then a profile, and vice versa.

Invalidity

A contour built on invalid elements now becomes invalid too.

Sketch

Visualization of sewn curves

In addition to show the ends of open curves, the **Show extremities** function now hatches the closed sewn curves in order to improve their visibility.

Each area is hatched with a different pattern, color and angle; five patterns are used with pastel colors in order for curves to be readable. These hatchings are hidden during a dynamic movement (zoom or rotation).



Fillet

When the value is too large or segments are tangent, a yellow circle is displayed on the vertex where the fillet cannot be performed.

The new **Mode=Local/Global** option is used to create a fillet on all angles of a sewn curve. A different radius can be specified for internal and external fillets.



Examples of fillets in **GLOBAL** mode, with same radii (left), with different radii (center), and with a too large radius (right).

Repassing

When creating a contour by passing over existing entities, leaving the cursor over a segment for one-half second allows this segment to be defined as a reference. In addition to the protractor, TopSolid then proposes a construction line parallel to the segment.

Simply click on this line to create a "parallel repassing" with a dimension imposing the constraint.

It is possible to build on segments of another sketch or on edges of a shape.



When creating the right contour, hovering your mouse over the red segment (1) allows you to display the axis (2) in order to hook on it.

Rectangle

The new **3 POINTS** option is used to create a rectangle by indicating 3 points; the rectangle thus obtained is constrained by 2 dimensions and perpendicularity constraints, which allows it to rotate around a vertex.

Curves/Other operations

Sewing

In non-associative mode, in order for the resulting curves to remain on the same level as the original curves, the **Check layer of curves** option must be checked and the value "-1" must be entered in the **Layer** field. It is also possible to keep the original attributes by checking the **Check color of curves** and **Check linetype of curves** options.

Curves/Manage

Superposed curves

Like for sewing, in order for the resulting curves to remain on the same level as the original curves, the **Check layer of curves** option must be checked and the value "-1" must be entered in the **Layer** field.

Shape

Extruded shape

The arrow for reversing the direction takes better account of the zoom in order to be visible on flat parts such as panels.

Enclosing shape

The **Optimize=YES/NO** option has been added in the case of an enclosing cylinder with a given axis; it accelerates calculation times in most cases.

Copy faces

In **COMPOSITE SURFACE** mode, the **Follow=EXISTING OPERATIONS/SUBSEQUENT OPERATIONS** option has been added. Moreover, the function ergonomics has been modified; the internal loop removal, trimming and merging options are now available in the advanced options that can be accessed using the button >>.

Mode=	COMPOSIT SURFACE	▼ Foll	low= EXISTING	OPERATIONS 🖘	Follow tangent faces=	NO 🖘 Hide parts:	YES 🖘 >	Select faces:
_					Remove internal loops=	YES 🖘 Trim with a	a curve= NO 🐔	Merge= YES

Knurling

It is now possible to knurl a planar face. This enables it to be associated with a knurling dimension in the draft.

Drilling

Major improvements have been made to the drilling function:

- Blind hole with global offset:

In previous versions, the offset was calculated in relation to the face intersecting the drilling axis. The new **Global offset** option takes all the part faces into account to calculate the offset (bottom faces + side faces).



Examples of drillings with offsets: on the left the offset mode, on the right the new **Global offset** mode.

- Drillings on circles:

For documents coming from a 2D import (AutoCAD DXF, DWG or others), the new **CIRCLES** option available in the **COORDINATE SYSTEM OR SKETCH** mode allows you to quickly create drillings on circles by automatically retrieving the diameters.

After selecting the circles and the reference face, the drilling type selection window is displayed for each circle. The **Merge=YES** option is used to identify circles with the same diameters in order to display the drilling type selection window only once.

The concentric circles are automatically recognized so that the spot faced holes can be performed.



Retrieving a draft document.



Rebuilding the initial shape.



Creating the drillings on the circles with recognition of identical circles.



Result after a few clicks.

- Ghost drilling:

In order to optimize the calculation time, in particular when inserting a component with a huge number of holes, it is now possible to create "ghost" drillings. A ghost drilling is a drilling without geometry; it is represented by a circular edge centered on its coordinate system and is positioned on the reference face. The circle diameter corresponds to the hole diameter.

The **Ghost drilling** mode is only available for **holes** performed in the tools of a component.

From the **TOOLS SET** in the symbolic tree, the **Define ghost tool** contextual function activates the ghost mode using a Boolean parameter (1: ghost mode enabled / 0: ghost mode enabled) which is automatically declared as an optional driver.

Then, when including the component, the Boolean driver is used to choose whether to create the component processes in ghost mode.



Example of a sheetmetal with a component performing 3500 holes in "ghost" mode. Increased performance by more than 50%; the draft is created almost instantly.

Smoothing

The **WITH A TOLERANCE** mode has been added to the face smoothing operation. This mode only asks for the faces to be smoothed and a tolerance. Both selection by window and selection by color are now supported. The value of the maximum deviation is shown by a point on the surface and a message is displayed in the Alpha bar.



Example of smoothing with tolerance.

Components

Addition of driver parameters

The management of driver parameters has been enhanced. Drivers can now be added to a component already included in an assembly.

After selecting a component using the **Modify element** function, the **PARAMETERS** option detects that the component template contains new driver parameters and provides a new **ADD THE NEW DRIVER PARAMETERS** button.

Using this button, the new parameters are then directly accessible from the modification window displayed by the **CONFIGURE SINGLE** option.

Secondary drivers

To make sub-component libraries easier to create, it is now possible to define secondary drivers.

Drivers defined as secondary drivers (identified by the icon $\stackrel{\text{def}}{=}$) on sub-components can be modified at several assembly levels. This allows parameters specific to each sub-component to be used without having to redefine them as drivers in the assembly.

The secondary driver parameter becomes useful only at the second level of inclusion; when a sub-component is replaced by another one, secondary drivers specific to the sub-component are then required.

Tolerances in catalog files

Values can be entered with tolerances for parameters in the catalog file of a standard component. Just like when entering tolerances in **TopSolid**, the values must be enclosed in brackets; they will be taken into account for the operations generated by the component processes.

	А	В	С
1	\$code	diameter	depth
2	Ø10x12	10[H7]	12[-0.2,+0.15]
3	Ø10x15	10[H7]	15[-0.2,+0.15]
4	Ø10x20	20[H8]	20[-0.2,+0.2]
.5			

Tools

Bisectrix coordinate system

The new **Bisectrix coordinate system** function is used to create a coordinate system at the intersection of 2 planes or 2 directions.



Example of a bisectrix coordinate system (in red) created by selecting the directions shown by gray arrows.

Dimensioning

The **Type=Angular/Linear** option has been added. In the case of parallel or quasi-parallel lines or axes (whose angle value is less than 10⁻⁶ degrees), it allows you to get an angular dimension instead of the linear dimension.

Attributes

Material

Transparency scales for material definition and those for texture definition have been standardized. In both cases, values can now vary from 0 to 10.

TopSolid'Image

Light and shadow settings

When creating a light, settings for **light** and **shadow** have been separated to make the configuration easier. Moreover, the default values of the different settings (**intensity**, **shadow opacity**, **shadow diffusion**...) have been modified in order to provide lights that can be used directly.

Reduced image calculation

A new reduced image calculation mode enables you to calculate the selected image (in **Global** or **Partial** mode) 75 or 50% smaller. This allows you to get a preview of your image more quickly in order to adjust light intensities.

Brightness adjustment

The new **TONE MAPPING** mode almost instantly adjusts the brightness of an image once it has been calculated.





Brightness adjustment of an image.

Draft

Dimensioning

The **Type=Angular/Linear** option has been added. In the case of parallel or quasi-parallel lines or axes (whose angle value is less than 10⁻⁶ degrees), it allows you to get an angular dimension instead of the linear dimension.

Drilling table

Drillings can now be filtered according to their diameter/radius values. This mode can be accessed from the function's advanced options.



Filtering by diameter/radius.

Moreover, the through all and through one holes are now differentiated in the depth column.

Table export to Excel

The **SEVERAL TABLES** option has been added to the **Bill of material** > **Export tables** function. It is used to export the different types of tables (table, drilling table, dimension table, bill of material,...) from a draft document to an **Excel** document.

By default, each table of the draft document is exported into a different workbook of the **Excel** document. The names of **Excel** workbooks as well as the position of tables can be customized by using a template.

oort tables				X
Template NAVIGATOR	Template file : C:	\Missler\Config\Templates\Exce	elT emplate.xlsx	
Tables	Types	Template sheets name	Destination sheets name	Titles
@184	aligned table of drill dimensions	Drill	TEMPLATE SHEET NAME	YES
@187	aligned table of drill dimensions	Drill	TEMPLATE SHEET NAME	YES
@190	aligned table of drill dimensions	Drill	TEMPLATE SHEET NAME	YES
@193	aligned table of drill dimensions	Drill	TEMPLATE SHEET NAME	YES
@196	aligned table of dimensions	Dimension	AUTOMATIC	YES
@519	aligned table of dimensions	Dimension	AUTOMATIC	YES
Destination NAVIGATOR	Destination file : C	∖ProjectsViews.xlsx		
	(OK Cancel		

Setting window for table export.

Visualization of differences

When the **File** > **Properties** - **Projection parameters** - **Views create 2d curves** option is checked, it is possible to view the differences when regenerating a document. Differences are displayed in a specific layer and elements that have changed are shown in magenta.

These differences can now be identified when the document is updated, by also showing the dimensions and notes that have been modified.

In the tree, the current view and the previous view are respectively stored in the **Resulting elements** and **Previous elements** sections. In the **Previous elements** section, elements that are different before and after update are visible, others are hidden.



Visualization of differences.

Title block

It is now possible to insert a multi-line text (note) in a title block.

Interfaces

VRML import

The VRML interface has been added. It allows you to import the surface shapes contained in the VRML documents and eventually to sew them.

Step export

When exporting a document in Step format, a drop-down list now allows you to choose the file extension (STEP or STP).

Interfaces have been updated in order to support the following formats:

Import AutoCAD 2012	
AutoCAD 2012	
Acis R24	
Inventor 2014	
Catia V5 (Datakit) R7 - R2	3
Catia V5 (Spatial) R6 – R2	23
Parasolid V26	
Pro/Engineer (Datakit) 2000i	
Creo 2.	0
Pro/Engineer (Spatial) 16 – W	ildFire5
Creo 2.	0
Google SketchUp 8	
SolidWorks (Spatial) 98 - 20	13
Unigraphics (Datakit) NX8.5	
Unigraphics (Spatial) NX8.5	
Export	
Acis R24	
Catia V5 R6 – R2	23
Parasolid V26	

What's new in TopSolid'Progress v6.15



This section describes the new features made to **TopSolid'Progress** in the version **6.15**.

Part preparation

Deletion of the AutoForm unstamping module (replaced by FTI)

The AutoForm unstamping option has been removed and replaced by the FTI module.

Former unstampings or former imported parts which used this option to be recalculated will be replayed without change, unless they are explicitly converted into FTI unstamping via the modification command. Additional information necessary to the calculation with FTI can then be required (the stamping direction, principally). The calculation result may be significantly different.

Concerning the codes, a FTI unstamping module must be requested in place of AutoForm module(s).

Improved selection of boundary edges

New options are available when selecting the boundary edges:

- Automatic selection of opposite edges to selected edges.
- Automatic selection of lateral edges.
- The **Cut face** option can be used at any time and no longer only at the beginning of the selection.



These options are available in the following commands:

- Define guiding edges
 - Remove a piece of the part
- **Transform a piece of part**
- Cut part
- **Second Flange unfold**
- - Trim and thicken
- Lateral extension

Improved Leading surface command

A new type of surface is available, the **Lofted** mode, in addition to the existing **Swept face** and **Extend faces** modes.

This new mode becomes the default creation mode as it provides better quality surfaces.



Lofted mode

Swept face mode

A preview of the surface is now available when creating the surface.

Strip

Import part 🕌

When importing a part from a station, several import modes are now available:

- **PROJECT**: Profiles of the part selected on the station are projected to create the strip. Only the contours of the part edges are shown (internal edges are ignored).
- **DUPLICATE**: This mode corresponds to the former import mode.
- **UNBEND**: The part of the station is unbent in the same way as a sheetmetal part.

Flange unfold 🗹

The creation of the automatic surface has been improved. The **Lofted**, **Swept face** and **Extend faces** creation modes are available (see **Leading surface** command).

It is possible to set an extension direction by point, just as it is possible to specify a unique extension direction.

In the flattening parameters, it is now possible to **create the boundary curves** corresponding to the boundary edges. Especially when the area to process includes internal cuttings, those cuttings will be applied to the result.

Meshing tolerance : @1012=07mm Maximum element scale : @1013=0.5mm Forming process For strip For stamping Material properties Materials > Default material Young's modulus : 210000MPa Yield stress : 231MPa Yield stress : 231MPa Plastic strain ratio (r-value): 1.87 Specific weight : 7.8 Poisson's coefficient : 0.3 FORMING FORMINC FORMINC FORM	Meshing tolerance: Imaximum element scale: Imaximum element scale:		Flattening parameters	
 For strip For stamping Material properties Materials > Default material Young's modulus: 210000MPa Uniform elongation (Ag) : 0.17 Plastic strain ratio (r-value): 1.87 Specific weight : 7.8 Poisson's coefficient : 0.3 Create boundary curves FORMING FECHNOLOGIES OK Cancel 	 For strip For stamping Material properties Materials > Default material Young's modulus: 210000MPa Uniform elongation (Ag) : 0.17 Yield stress : 231MPa Plastic strain ratio (r-value): 1.87 Specific weight : 7.8 Tensile strength : 247MPa Poisson's coefficient : 0.3 Create boundary curves FORMING TECHNOLOGIES INCORPORATED INK Cancel 	Meshing tolerance Maximum element sca Forming process	: @1012= <mark>0.7</mark> mm le : @1013=0.5mm	ces 🗸
Material properties Materials > Default material Young's modulus: 210000MPa Yield stress : 231MPa Tensile strength : 247MPa Create boundary curves Create boundary curves FORMING FORMING FORMING FORMING CREATED OK Cancel	Material properties Materials > Default material Young's modulus: 210000MPa Yield stress : 231MPa Plastic strain ratio (r-value): 1.87 Specific weight : 7.8 Poisson's coefficient : 0.3 Image: Create boundary curves Image: Create boundary curves	For strip	 For stamping 	
Young's modulus : 210000MPa Yield stress : 231MPa Tensile strength : 247MPa Create boundary curves FORMING FORMINA	Young's modulus: 210000MPa Yield stress :231MPa Yield stress :231MPa Specific weight :7.8 Poisson's coefficient :0.3 FORMING FORMING FECHNOLOGIES INCORPORATED OK Cancel	Material properties Materials > Default m	laterial	~
Yield stress : 231MPa Plastic strain ratio (r-value): [1.87 Tensile strength : 247MPa Poisson's coefficient : 0.3 ✓ Create boundary curves FORMING FORMING FORMING FORMING FORMING Image: Composition of the strength : 0.3 ✓ Create boundary curves Image: Composition of the strength Image: Composition of the strength : 0.3 Image: Composit	Yield stress : [231MPa Plastic strain ratio (r-value): 1.87 Specific weight : [7.8 Tensile strength : [247MPa Poisson's coefficient : [0.3 Create boundary curves FORMING FORMING FORMING TECHNOLOGIES INCORPORATED OK Cancel	Young's modulus : 21	0000MPa Uniform elongation (Ag) : 0.17	
Specific weight : [7.8 Tensile strength : 247MPa Poisson's coefficient : 0.3 Create boundary curves FORMING	Specific weight : [7.8] Poisson's coefficient : [0.3] Create boundary curves FORMING TECHNOLOGIES INCORPORATED OK Cancel	Yield stress : 23	Plastic strain ratio (r-value): 1.87	_
Create boundary curves FORMING FORMIN	Create boundary curves FORMING FORMING TECHNOLOGIES INCORPORATED OK Cancel	Tensile strength : 24	Specific weight : 7.8	_
Create boundary curves FORMING TECHNOLOGIES INCORPORATED	Create boundary curves FORMING TECHNOLOGIES INCORPORATED OK Cancel	rensie sterigti . 24	Poisson's coefficient : U.3	
FORMING TECHNOLOGIES INCORPORATED DK Cancel	FORMING TECHNOLOGIES INCORPORATED OK Cancel	Create boundary c	urves	
			PORMING TECHNOLOGIES INCORPORATED OK Cancel	
			00	

Part to stamp

Unstamping with boundary curves

Improved management of 3D strips

When creating a 3D strip, a **3D strip** set is automatically created. This set allows you to strictly isolate elements of the **"real" strip** (the one which is built), especially for projections **in drafts**, or the **kinematic**, from elements used to create it.

This set does not exist when the document is created; it can be created if needed.

Processes



In PROCESSABLE mode, the multiple selection has been disabled to prevent handling errors.

Processes

It is now possible to delete empty operation folders generated through the creation of a **process** from the construction tree.

Components

Improved Reproduce command

The tools for selecting target components have been enhanced. Invisible components can now be selected.

Configuration of components

It is now possible to **copy/paste** the default **clearances**, **standard machinings** and **attributes** of a component to another component of the standard library.

Management of properties of compone	ents	×
PGS_MISUMI PGS_PEDROTTI PGS_RABOURDIN PGS_STRACK PGS_STRACK Accessories Cutting Fixing Dowel pin Screw Countersunk head screw Countersunk head screw Screw Screw Shoulder bolt Socket head cap screw(10.9) Socket head ca		
Standard reference	Carrel	-
Copy paste Copy Paste ✓ Clearances ✓ Standard machinings Clearances associated to processes Standard machinings Default attributes Color	Component attributes	
☑ Predefined color		
Colour :		
Layer		
Predefined level		
Layer: -1		
OK Cancel		

Definition of components

A new type of process is available in the definition of the **TopSolid'Progress** standard components. A component with a **not through countersunk hole** can now be created as a process.

Tools and dies



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 round values, easier to manage for machining.

Cutting die oper	ration parameters
Dimensions Machining	process
Туре	
One part	Two parts
l l	
First part	
Clearance (on radius):	: 0.1mm
Height:	: 10mm
Draft	0.5
Without Angle	Export length () Slope
Angle:	: 1*
Export length:	: Umm
Height:	: jUmm
- Second part	
O Shift	Uiameter
Shirt:	: Umm
Diameter: Minimum Val	ue: 10.55mm : 12mm
Without Angle) Export length OSlope
Angle:	: 1*
Export length:	: 1mm
Height:	: Omm
Extend	
On top:	: Omm
At bottom:	: Omm
Authorize machining	mode for offset computation
OK	Cancel

Progressive die kinematic

The kinematic now takes into account the strips built in **inverted** direction, with their progression direction along x-.

If the document contains a **3D invert strip**, this strip is ignored unlike the "**real**" strip.

A strip inserted as a **component** in a new document is now recognized as a strip in the kinematic of the progressive die.

Draft



It is possible to add the strip projection **to all open views** by modifying these views.

What's new in TopSolid'Cam v6.15



This section describes the new features made to **TopSolid'Cam** in the version **6.15**.

2D milling

• Optimization of milling toolpaths (like for drillings).



Turning

• Automatic synchronizations according to the Operations manager.



3D milling

New roughing.



- New algorithms for left material machining.
- Morphing from or to a point.



Verification

• Collision control with the finish.

WCS

• Information on the work plane and PP words is now provided in the WCS manager.

Part name: pièce usinage 2 Material: XC38							
mment	Spindle direction	Axe_B (TABLE)	Origin WCS	Work plane	PP		
Posage pièce usinage 2	Z·	0		XY Z-			
Z- Axe_B 90	Z·	90		XY Z-			
Z- Axe_B-90	Z·	-90		XY Z-	MAB2014		
		ОК					
					8		

Parts

• When including a part (for 3D milling), holes (ejector pin) and/or operations can now be deleted.



• Draft analysis.



Shopfloor documents

• The part views of operations are now in the same direction as the WCS.

Old view:



What's new in TopSolid'Wood v6.15



This section describes the new features made to **TopSolid'Wood** in the version **6.15**.

Improved sketches

Some points of use of sketches have been improved to make sketches easier to use.

Visualization of curves

When creating or modifying a sketch, the Show extremities function now makes it easy to view the closed

contours created within the same sketch.

- Open contours are shown by two dotted circles at the ends of the contour.



- Closed contours are now hatched.

Each hatching uses a hatching template and a different color in order to be identified more easily. When zooming or moving in the sketch, these hatchings are hidden in order not to overload the view.



Parallel contour on the fly

When drawing a contour in a sketch, it is now possible to directly draw a segment parallel to an element.

The function is available from a contour that is being created.



• Position the cursor on the reference element to draw the parallel segment.

Note: The reference element to draw the parallel segment can be an edge of a shape, a segment of the current contour or sketch, or a segment from another sketch.



• A new **dotted red** construction line appears. This line is parallel to the selected element passing through the last point of the contour.



• Select a point on this parallel line to draw the segment of the contour parallel to the selected element.

A length dimension is created between the reference element and the segment of the contour.

Segment length

In a sketch, when drawing a contour, it is now possible to directly enter the length of the next segment.

- From a sketch, start the Contour function.
- Click on a first point to start the contour.
- In the **Segment length** box, set the length for the next segment of the contour, then press **Enter** to confirm.

Segment length= 100mm

• Then click on the passing point of the segment.



• A length dimension is automatically created on the segment with the length you entered.



• It is also possible to use the Link = Axes (Z) option in order to set the length of the segment aligned on X or Y.

•

Link= AXES (Z)



Rectangle by 3 points

When drawing a **rectangular contour**, a new option allows you to draw a rectangle by defining it by 3 points.

- Select the **Contour** function . then **Rectangular**. RECTANGULAR
- Select the **3 points** option. 3POINTS
- Select the 3 points to draw the rectangle.



- If the selected points are vertices, coincidence constraints are created between the corner of the rectangle and the selected vertex.
- The two rectangle dimensions are also created: if the rectangle dimension is constrained by the point selection, the dimension is passive (yellow), otherwise it is active (green).

Selection of any point for the third point.







Improved Fillet function

Global fillet

A new option enables fillets to be created automatically on a whole sketch contour.

- From a sketch, select the Fillet function.
- Set Mode = Local/Global.

The **local mode** is used to create a fillet on a single selected angle. The **global mode** is used to create the fillet on all angles of the selected contour.

• Enter the value of the **fillet radius**, and then select the contour.

Note: The contour can be an open contour.



• Then click on the **Compute fillet(s)** button to create the fillets. [COMPUTE FILLET(S)]

When viewing the fillets in red, yellow circles can be drawn at some angles of the contour. These are angles on which the fillet will not be performed. The fillet is not created if:

- The angle is tangent, in which case a fillet is not necessary.
- The fillet radius is too large and it is not possible to make the fillet.


Interior/Exterior fillets

It is now possible, on a closed contour, to create fillets with a different radius if the fillet is inside or outside the contour.

- Start the **Fillet** function.
- Select the Radius inter/exter option. RADIUS INTER/EXTER
- Enter a radius value for interior and exterior fillets.

Destruction 30mm	Dedition and action 10mm	Classed assure to an alter
Hadius Interior= John	Hadius exterior= romm	Liosed curve to modify:

• Select the **closed curve** on which fillets will be applied.



Fillets are previewed in red. Fillet radii can be modified before being applied on the contour.

• Select Compute fillet(s) to create the fillets.





Coordinate systems

Selecting points to create a coordinate system axis

When creating a coordinate system, when an axis is required, it is now possible to select two points in order to choose as the direction the line passing through these two points.

This mode may be used when defining the axes manually from the **Define part** function.



- When selecting an axis for the coordinate system being created, select **Through point**.
- X+ X- Y+ Y- Z+ Z- THROUGH POINT Length axis:
- Select the first through point for the axis.

Through point:

X+

Note: The selection order of the two points provides the axis direction, but it can be subsequently modified.



• Then select the second through point for the axis.

 X:
 Y+
 Y.
 Z+
 Z.
 Direction or second through point:

- The red arrow allows you to set the direction of the created axis. Click on the arrow to invert.
- Confirm the axis direction by clicking on **OK**.

• Create the second coordinate system axis.

The created coordinate system axis (in red below) passes through the two selected points (in blue). For the example below, the axis used by the coordinate system is shown by dotted lines.



Bisectrix coordinate system

A new **Plane Bisectrix coordinate system** allows a coordinate system to be positioned directly between two planes.

For example, this makes it possible to create mitre cuts between complex parts.



<u>Note</u>: The current coordinate system's planes can be selected using the **XY+**, **XY**-... buttons, and planes parallel to the current coordinate system's planes passing through a point can be selected using the **Through point** function.



• Adjust the face direction using the red arrow, then confirm by clicking on **OK**.

Note: The directions of the two selected faces must be identical, inward or outward. The side will be used to position the X axis of the coordinate system thus created.



• Select the second face, and then adjust the arrow direction like the first arrow.



The coordinate system is positioned by default.

• Select an origin point to position the coordinate system.

<u>Note</u>: The selected point is projected onto the intersection curve of the two selected faces in order to position the bisectrix coordinate system.



Select Quit to finish creating the coordinate system.



 Perform the cuts on the parts in relation to this bisectrix coordinate system using the Shape > Trim > By plane function, since the trimming plane is the bisectrix coordinate system created before.



•

Drilling improvements

Drilling on a 2D circle

A new option to create drillings directly from a 2D circle is now available. This circle can be for example a circle from a DXF document or a circle of a sketch.

The circle diameter will be automatically measured on the selected circle.



- Select the Coordinate system or sketch option, then the Circles option.
 CIRCLES
- Set Merge drillings = Yes/No.

<u>Note</u>: The Merge drillings = Yes option enables all the drillings to have the same parameters (except the diameter specific to each selected circle).

• Select the circle(s) to be drilled using the selection.

<u>Note</u>: The different selected drillings must be on the same reference face.

- Select the reference face of the drillings.
- Select the drilling model to be performed.
- Set the drilling parameters, then confirm with **OK** to create it.

<u>Note</u>: If the **Merge drillings = No** option has been selected, the **drilling model** selection window as well as the setting window are opened for each selected circle.



<u>Note</u>: When selecting the circles to be drilled, it is possible to select two concentric circles. It will then be possible to select a **spot faced drilling model**: the drilling will be performed on the smallest circle and the spot facing will be performed on the largest circle.



Ghost drillings

The new ghost drilling allows you to optimize performance on assemblies that use a lot of drilling processes. When performing these drillings, the full shape of the drilling is not displayed; only the circle on the machined face of the drilling diameter is displayed.

The **Ghost drilling** mode is available for **holes** performed in the tools of a component.

• In the template of the component which must perform the drillings, open the symbolic tree, then **edit the tool set**.





On the line of the tool, right-click > Define ghost tool.
 Define ghost tool

<u>Note</u>: The **ghost** mode can be set using a **Boolean parameter**. This parameter is used to enable the ghost mode (value 1) or leave the drilling to normal (value 0).

In the Ghost parameter field, specify the parameter to be used to drive the ghost tool.
 If there are Boolean parameters in the document, they are all available in the Parameter drop-down list.

Parameter: Ghost drilling 👻 Ghost parameter=

• Validate the parameter by pressing Enter.

The drillings contained in the tool become ghost drillings if the ghost parameter is 1.





Note: If the tool defined as ghost contains several drillings, all the drillings contained are then defined as ghosts with the same Boolean parameter.

However, it is possible to modify a drilling of a tool locally.



• On the drilling to be modified, **right-click** > **Define ghost tool**.

Parameter: Middle ghost drilling 👻 Ghost parameter= gm=1

Note: The **Reset ghost parameter** option is used to remove the ghost option from a drilling or a ghost tool.

- Select **Reset ghost parameter** if you no longer want to define this drilling as **ghost**. RESET GHOST PARAMETER
- In the **Ghost parameter** field, enter the new **Boolean parameter** to drive the ghost mode of the drilling.

When inserting this component, it is then possible to perform the machinings in **ghost mode**.



When drafting parts with ghost drillings, drillings will only be visible on the top view of the ghost drilling's r



Performance gains

Tests performed on a 2800x2070mm panel with 4698 drillings in matrix repetition (32mm spaces):

- X: 81 drillings. _
- Y: 58 drillings. _

			TELLER HORE	
,				
				M.M. Martin
	11111111111	1910 Carlos Carlos		

Tosts	In place	File size	File	Export to	Component process
Tests	propagation		regeneration	WoodWop	creation
Normal drillings	19s	5 275 KB	25s	2min 05	18s
Ghost drillings	7s	759 KB	7s	5s	6s

Improved drivers

Secondary drivers

Secondary drivers have been created to improve the use of sub-components.

Secondary drivers are defined in sub-components and can then be modified at several assembly levels. This allows you to define parameters specific to each sub-component without having to define them in all equivalent sub-components.

The secondary driver becomes useful only at the second level of inclusion.

In the example below, the **Width**, **Height** and **Thickness** parameters are present in all the models. On the left-hand panel, the **Crosspiece width** and **Panel number** parameters are specific to the model.



TopSolid 2014

Parameters must be created beforehand in the sub-components.

- Open the parameter list using **Parameters > Edit list > Document**.
- From the **Driver** column, define the parameters specific to the sub-component as **secondary** drivers.
- In the case of Boolean parameters (Yes/No), select Boolean secondary. Boolean secondary

Name	Designation	Display unit	Value	Туре	Driver
😌 w	Panel width	mm	500mm	parameter	Yes
 €h	Panel height	mm	1800mm	parameter	Yes
🖳 th	Panel thickness	mm	38mm	parameter	Yes
🖳 cw	Crosse width	mm	70mm	parameter	Secondary
🖳 n	Panel number		3	parameter	Secondary



<u>Note</u>: Secondary drivers can also be defined from the symbolic tree in the **Entities** tab. Open the **Parameters** list > **right-click** on the parameter to be modified > **Define driver** > **Secondary driver** = **Yes**.

OK Driver= YES * Secondary driver= YES *

✓ ENTITIES SET: (4)
 ✓ SHAPES: (5)
 ✓ SKETCHS: (1)
 ✓ PARAMETERS: (6)
 ✓ Panel width = 500mm
 ✓ h: Panel height = 1800mm
 ✓ th: Panel thickness = 38mm
 ✓ cw: Crosse width = 70mm
 ✓ n: Panel number = 3
 ✓ ∑ ph: Panel height = (h-2"cw-{n-1}"cw)/n = 506.667mm
 ✓ COORD SYSTEMS: (2)

Main Favorite Main set Entities Layers

Secondary drivers are displayed with a yellow +. 🥰

In the component, define the sub-components using the Assembly > Define component > Define sub-component function.



When using the component, it is then possible to interchange the door panel model.

- Start the **Modify element** function and select the component to be interchanged.
- Select Sub-component.
 SUB-COMPONENT
- Select the sub-component to be interchanged.
- Select the Interchange option. Interchange
- Select the new standard, then confirm with **OK**.

After interchanging the **sub-component**, it is proposed in the dialog bar to configure the **secondary drivers** specific to the standard selected for the sub-component.

OK Secondary drivers= FIRST LEVEL * Parameter to modify= Crosse width CONFIGURE SINGLE

<u>Note</u>: The **Secondary drivers = First level/All levels** option allows you to select the secondary drivers to be configured:

- FIRST LEVEL * : Used to display only the secondary drivers of the interchanged sub-component.
- <u>ALL LEVELS</u> : Used to display the secondary drivers of all the sub-components contained in the interchanged sub-component.
- Select the **secondary driver** to be modified in the drop-down list, then enter its new value.
- Select the **Configure single** option in order to modify all the **secondary drivers** in one go. CONFIGURE SINGLE

<u>Note</u>: The First level/All levels option is also available in the Configure single option.

Secondary drivers First level
All levels

It is also possible to access the secondary drivers of a sub-component without interchanging it.

- From the Modify element function, select the component to be configured.
- Select Sub-component. SUB-COMPONENT
- Select the sub-component to be configured.
- Select the Parameters option.
 Parameters

Door

and the panel grain orientation.

oh=1500

Example:



When modifying the secondary parameters of the door panel:

- The First level option is used to modify only the secondary parameters of Parameter to modify= Horizontal grain? _ the panel.
- The All levels option is used to modify the secondary parameters of the _ panel, as well as all the sub-components including the oculus.

lorizontal grair Opening position Parameter to modify= Horizontal grain? lorizontal gra Opening position Opening width Opening height

Improved driver block

Direction of arrows

To enable better understanding of the driver blocks, the arrows that symbolize the positioning faces now show the inside of the block.



Double wrap

The arrows that symbolize the double wrap of the double-wrapped driver blocks now show the outside of the



block. Their shape is also changed for a double arrow.



Visualization of hookings on publishings

Since version 2013, if the positioning face of the **driver block** has a publishing, the driver block hooks on this publishing to enable better interchangeability.

The hooking of the **driver block** on a **publishing** is now shown by a green arrow and a rectangle.



Sub-slope driver block

The new **sub-slope driver block** allows you to fit out volumes with an angle on the upper part.

- In a new TopSolid'Design document, start the Assembly > Define component > Define driver > Driver block function to create the driver block.
 DRIVER BLOCK
- Enter the **name** of the **driver block**.
- From the advanced parameters >>>>, set
 Sub slope = YES.
- In the **Angle** field, enter the default value of the sub-slope angle.

Note: Here, it is possible to enter a negative value to get a slope in the other direction. Only a value between -89 and 89° can be entered.



• Then design the component in the **driver block** normally.



<u>Note</u>: The sub-slope driver block can be included in **Mode = Inside a block** or **Mode = Housing**.







TopSolid 2014

What's new in TopSolid'Wood v6.15





The sub-slope component is then calculated automatically.





Table text

The table text is used to vary a text according to two parameters in a table.

For example, in the case of this shelf, two parameters allow you to set the thickness and depth of the shelf.

The table text will then enable you to choose the code of the cam to be used according to these two parameters.

- Start the **Tools** > **Text** function.
- Open the **advanced parameters** >>, and then select **Table text**. TABLE TEXT

The table text can be set using a table; one parameter is on the table columns and the second one is on the table rows.

• In the **Column parameter** field, enter the name of the parameter to be placed on the table columns, then confirm by pressing **Enter**.

Column parameter: d

In the Line parameter field, enter the name of the parameter to be placed on the table lines, then confirm by
pressing Enter.

Line parameter: th

The table is then created.

- In the window that opens, enter the first value for the Column parameter, then confirm with OK.
- Right-click on the column headers to insert, remove or modify a column.



inngs tab	he		
d∖th ≽	<= 200 V	1200-250.1 2250 Modify Insert new column Remove column	

- Double-click on the first box of the first column to create a new value.
- Right-click on a row to **insert** or **remove** a row.
- Then **double-click** in each box of the table to enter the text to be displayed when the conditions of the **row** and **column** parameters are met.
- Validate the table by clicking on **OK**.

rings ta	ble		X
d∖th	<= 250	>250	
<= 12			
12			

d∖th	<= 250	>250	
<= 12	TH12 L20	TH12 L30	
] 12; 15]	TH15 L20	TH15 L30	
] 15; 16]	TH16 L20	TH16L30	
] 16; 18]	TH18 L20	TH18L30	
] 18; 19]	TH19 L20	TH19L30	
] 19; 22]	TH22 L20	TH22 L30	
] 22; 29]	TH29 L20	TH29 L30	
>29	TH29 L20	TH29 L30	

• Click on an **alignment point** to position the text.

The text displays the value defined in the table according to the values of the **row** and **column** parameters.



In case this text must then drive a code or a **sub-component variant**, simply define the component as a **sub-component**.

Library update

The **Update libraries** function has been enriched with a new function to regenerate the updated files.

- Start the File > Update libraries function.
- Set Regenerate files = YES, then confirm by clicking on OK.

OK Regenerate files= YES 🗫

It is recommended that you save the library to be updated beforehand.

- After saving, confirm the warning message with **Yes**.
- Select the folder that contains the library to be updated, then confirm by clicking on **OK**.

The **TopSolid'Design** files contained in the library are then opened, updated, regenerated, saved, and then closed.

Component distribution

Gap management between each component

A gap management is now made directly from the component distribution. This makes it possible to generate a free interval between each distributed component.

When setting the distribution parameters, the **Gap** field is used to set this interval between each component.



This parameter can then be modified using the **Modify element** function a distributed component, then select **Distribution parameters**.

It is also possible to access this parameter by editing the distribution from the symbolic tree.

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 Image: State of the stat

on

Additional coordinate systems are also generated in the distribution:

- One before each component (in blue below);
- One in the middle of each generated interval (in green below).

Note: These coordinate systems are hidden by default. Click on Inv=Hid in the status bar to display them. Inv=Hid



These coordinate systems are used to place components in the gaps.

If a component is placed using a key coordinate system on one of the gap coordinates systems (in green on the previous page), it will then be possible to repeat it automatically in each generated distribution gap.

- Include the component in the distribution assembly.
- Set the width parameter of the component to the gap value.
- Place the key coordinate system of the component on one of the gap parameters.
- Select the **Repeat** option REPEAT, and then **Serial copy**. SERIAL COPY
- Select the Automatic option. AUTOMATIC

<u>Note</u>: If the **Automatic** option is not available, it means that the component is not positioned on one of the distribution gap coordinate systems.

• If the component contains tools, select **Automatic** in order to perform them.

<u>Note</u>: A serial component is then generated in the distribution gaps.



If the distribution is modified and the number of gaps changes, the **serial copy** is used to automatically update the component quantity.



Empty space management

When the **Distribution mode = Mark out** is used, each component is generated to the value given in the parameter and the adjustment is made on the last component.

It is then possible not to generate this component in order to create an empty space using the **Fill out mode = Void**.



The value of this empty space can now be recovered thanks to a parameter that is automatically calculated from the distribution edition in the symbolic tree. This **Space** parameter, which can be named to be used in the assembly, is then automatically updated after the distribution parameters are modified.

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Space = 246.08mm

Management of components with minimal value

When the **Distribution mode = Mark out** is used, each component is generated to the value given in the parameter and the adjustment is made on the last component. If the length parameter used in the component distribution definition has a minimum value, the **last component** is not generated if its length is less than the defined minimum value.

• In the component used for the distribution, define a **minimum possible value** on the parameter used in the **distribution positioning**.

MINIMUM POSSIBLE VALUE=300.000mm





If the space of the last component is greater than the minimum value, the last component is generated.



However, if the space of the last component is less than the minimum value, the last component is not generated, an empty space is created and the value of the **Space** parameter is calculated.





TopSolid'Wood

Integration of the Hettich 3D component library

Hardware components of the Hettich catalog now come directly with TopSolid'Wood. Based on the same principle as the **Blum** hardware components, this library contains

the 3D geometries of Hettich hardware components to allow more convenient integration and use in TopSolid'Wood.

From the installation of the 2nd **TopSolid** DVD, select **Hettich Hardware Library**.

Hettich Hardware Library

An installation of about 1,1GB will be downloaded.

Then install this application on the **TopSolid'Wood** station.

The Hettich Hardware library will then be available from the standards

Edges and laminates

Edges automatically modifiable in assembly

A new option enables you to make all the created edges automatically modifiable in the assembly.

- Open the options using **Tools > Options**.
- Open the **TopSolid'Wood configuration** > **Edge/Laminate** section.
- Check the Set edges modifiable in assembly box. 🗹 Set edges modifiable in assembly

As a result, when creating edges (Panels, Edges or Edge wizard functions) the created edges will be automatically defined as modifiable in the assembly.

💋 Panel drawer facade
- 😽 Flat
Flat
Panel









Laminate notes in TopSolid'Draft

When drafting laminates or laminate panels, a new function makes it possible to place notes on the laminates to display some specific information.

- From the draft template document, open the document properties.
- Open the **TopSolid'Wood properties > Draft** section.
- In Elements to dimension, check Laminate dimensions. I Laminate dimensions
- Then open the Laminate tab and configure the elements to be displayed in the laminate note. Laminate

Creation of laminates on a panel:

	N*	Codification	Material	Coating	Thickness
~	1	ST-12-1	MEL-WD-12		1mm
	2	ST-6-2	MEL-WD-06		2mm

Characteristics of the laminate:

Sizes	Values	Modes	Over dime	
Length	596.0mm	additional	Omm	
Width	296.0mm	additional	Omm	
Thickness	1.0mm	additional	Omm	

Edge	Laminate	
- Infor	mations to print in laminate dimensi odification	ons
🔽 La	aminate length	
🔽 La	aminate width	
🔽 La	aminate thickness	
🔳 La	aminate matter and coating	
🔽 0×	ver dimension length	
🔽 0×	ver dimension width	
0	ver dimension thickness	

- When creating the draft, start the **Wood** > Laminate dimensions function.
- Select the laminate to dimension, and then place the note.

<u>Note</u>: By default, the information displayed in the note is that which is set in the document **properties**.

Click on the Informations to print in laminate dimensions button to modify the information to be displayed.
 INFORMATIONS TO PRINT IN LAMINATE DIMENSIONS



<u>Note</u>: Laminate dimensioning cannot be made automatically when multi-drafting panels with laminates.

Modifying the part grain orientation of a component

Modifying the grain orientation of a part is now possible in the assembly.

This modification allows the part axes to be inverted, just like with the **Invert axis** button when defining an inplace part. INVERTAXIS

This rotation can be driven by a **Boolean parameter**, making it possible to invert the axes of several parts using the same parameter.

Note: The grain orientation of a part can only be modified if:

- The part has been predefined and the **Add to cutting-up** box is checked.
- The dimensions are in **Respect axis** mode.
- Open the symbolic tree.
- Open the **Main set** tab.
- Select the part(s) to drive in the assembly.



<u>Note</u>: It is possible to select several parts by holding down the **Ctrl** key.

• Right-click > Drive grain orientation.

Drive grain orientation

- Set Drive grain orientation: YES.
- In the **Condition** field, specify the **Boolean parameter** to be used to invert the grain orientation:
 - If the **Condition = 1**: The grain orientation is rotated 90°.
 - If the **Condition = 0**: The grain orientation remains as defined by the axes.

<u>Note</u>: The drop-down list is used to select an existing **Boolean parameter** directly.

```
OK Drive grain orientation: YES 🖘 Condition= sfp=0 sfp 👻
```

• Confirm the dialog by clicking on **OK**.

As a result, modifying the parameter (in place or in the assembly) inverts or not the grain orientation on part(s) driven by this parameter.





Forcing the grain orientation

It is also possible, in the assembly, to force the grain orientation of parts already driven by a parameter.

- In the assembly, open the symbolic tree.
- On the part where the grain orientation must be forced, **right-click** > **Force grain orientation**.
- Sorce grain orientation

• Set Force grain orientation = YES.

OK Force grain orientation: YES 🖘 Condition=

• Enter the value **1** to rotate the part grain orientation and the value **0** for not rotating it.

<u>Note</u>: Here, it is possible to specify a parameter of the assembly to drive the part grain orientation.

The part grain orientation is then forced and will no longer follow the modification of the model's driver parameter.



Define multiple parts

Modifying the BOM depth

It is now possible to modify the depth of the BOM displayed from the **Define multiple parts** window. The three BOM display options are then available:

- Flat BOM
- At top level
- Multi level

In **Multi level** mode, the **Expand all** and **Collapse all** buttons are used to expand or collapse all the nodes of the BOM.

Bill of materia	al level choice		1	1
Flat BOM	🔘 At top level	Multi level	EXPAND ALL	COLLAPSE ALL

BOM sorting of the Define multiple parts

It is now possible to sort the part list displayed in the **Define multiple parts** window in ascending or descending order according to a column of the BOM.

• On the column to be sorted, **right-click** > **Sort in ascending/descending order**.

Description management

Descriptions are now managed from the **Wood** > **Define** > **Define multiple parts** function. The same **description** can be assigned to several different parts in one go.

Column width management

The column widths displayed in the **Define multiple parts** function can now be set by default.

- From a Design document, edit the BOM file used for the Define multiple parts function using the Tools > Edit bom file function.
- Modify the width values of the different columns.
- Validate the BOM file with **OK**.

Title	Definition	Туре	Align	Width	Format
Index	INDEX 3D	CHARACTER STRING	LEFT	20mm	
Count	COUNT	INTEGER	LEFT	20mm	
Part_type	PART_TYPE	CHARACTER STRING	LEFT	25mm	
Designation	DESIGNATION	CHARACTER STRING	LEFT	25mm	
Reference	REFERENCE	CHARACTER STRING	LEFT	25mm	
Material	WOO_PANEL_MATTER	CHARACTER STRING	LEFT	25mm	
Thickness	PART THICKNESS	REAL	LEFT	20mm	02 mm
Length	PART LENGTH	REAL	LEFT	15mm	02 mm
Width	PART WIDTH	REAL	LEFT	15mm	02 mm

When using the **Wood** > **Define** > **Define multiple parts** function, the column width is set according to the BOM settings.

Define part

Machining and drawing default settings

The Define part settings for machining and drawing can now be set by default in the options.

These settings will be taken into account when defining new parts.

 Open the new Define part settings using Tools > Options > TopSolid'Wood configuration > Define parts.



Set the default values to be used for the Machining Machining and Drawing Drawing tabs.



Validate the options with OK.

When defining new parts, the settings of the **Machining** and **Drawing** tabs will be those configured in the **Tools** > **Options** function.

<u>Note</u>: Changes can be made to the default settings of the **Machining** and **Drawing** tabs from the **Define part** window.

Then, when defining new parts, the default settings will be those configured from the **Tools** > **Options** function.

Interface of the Cutting-Up and Machining tabs

The **Cutting-up** and **Machining** tabs of the **Define part** window have been modified to provide better access to the functions and reduce the height of the window.



Drawing	Bill of materi	al Valorisa	tion Attribu	utes Properti	es Part types
Cutting-u	p Stock	Material	Coating	Machining	Description
Mode One Later Calib	machining file al operations ration in seco	Two fil in second m nd machining	es if necess achining file g file	ary	
Positior Ident Optin Force	ning ical to sawing nized Upda ed	jup coordina te machining	te system face		
First pos	sitioning Sec	cond position	ning		
Part p	oositionning or	n machine (t Working Butée 1	op view) station		•

WoodWop interface

Optional machine configuration

The machine to be used for the **WoodWop** machining export can now be configured for all exports.

Open the WoodWop export options using Tools > Options > TopSolid'Wood configuration > Machining > WoodWop configuration.

The Machine drop-down list allows you to use a specific machine for export.

Note: Selecting a specific machine makes it possible to change some points of the export format in order to match the machine used in **WoodWop**.

Machine		
Machine >	HOMAG	
	HOMAG CF-HOMAG	
	FK-HOMAG WEEKE	

Milling lead in mode management

Two new configuration words have been added to enable an advanced configuration of the lead in mode of part calibrations in **WoodWop**.

<u>Note</u>: The configuration words are to be inserted in the **topzwood.cfg** file used. To know the location of this file, start the **Help** > **Configuration** function, and then open the **TopSolid'Wood** tab.

Calibration origin in the middle of a segment

ZX_ZWOO_CAM_CALIBRATION_CHANGE_ORIGIN 1

At the value 1, this configuration word enables the starting point of a part calibration to be placed in the middle of a segment of the calibration.



Modifying the calibration lead in distance

ZX_ZWOO_CAM_MPR_CAL_DISTANCE 0.050



50.0000	



The calibration lead in distance is available in **WoodWop** only on a closed calibration, if the **Entry** and **Exit** modes are vertical and if the **Plunge** is enabled.

Nesting: Customization of the displayed text

It is now possible to customize the text displayed on the parts of a nesting.



The text font can be customized using the **Tools** > **Options** > **Nesting** > **Nesting default parameters** > **Identification** command.



The text can be customized via a **Wood codification** named **WOO_NESTING_IDENTIFICATION** in the BOM file used to sort the parts.

Note: For additional information on using the Wood codification, refer to the Wood codification documentation.

```
NAME=IDENTIFICATION

"DEF=<WOO_NESTING_IDENTIFICATION|$DESIGNATION$-$DESIGNATION|OWNER_PRJ$-$ELEMENT_IDENTIFIER$>"

TYPE=STRING

ALIGN=LEFT

TITLE_ALIGN=LEFT

WIDTH=0.015

VISIBLE=YES

;
```

The Wood codification above will display the part designation, the set designation, as well as the part identifier.

- Open the options using **Tools** > **Options**.
- Open the **TopSolid'Wood configuration** a > **Files** + **Parts selection sorting** section.
- Double-click on the Automatic nesting line, and then select the BOM file with the WOO_NESTING_IDENTIFICATION.
- Confirm the options by clicking on **OK**.
- Perform the automatic part nesting.

The customized text is then displayed.

Note: The text follows the rotation of the part.



TopSolid'Image

Reduced image calculation

A new reduced image calculation mode makes it possible to calculate the selected image (in **Global** or **Partial** mode) 75 or 50% smaller. This allows you to get a preview of your image more quickly in order to adjust light intensities.

- Start the Image > Display function.
- Set Size = 50%/75%/100%.

Calculation in 50%

- Launch the image calculation by clicking in the document's graphical area.
- Save the image with a **right-click** > **Save/Save as**.

Time for a calculation in **Quality = 3** and **Radiosity = 3**:



Brightness adjustment

A new mode allows you to adjust the brightness of an image once it has been calculated.

- Start the Image > Display function.
- Set Adjustment = Tone mapping.

Adjustment= TONE MAPPING 🛛 👻

• Launch the image calculation by clicking in the document's graphical area.

Once the calculation is completed, when the image is displayed, a **Tone mapping** window opens.

Post treatment	X
Adjustment :	0

• Move the slider using the mouse to adjust the brightness of the calculated image.



- Validate the **Tone mapping** by clicking on **OK**.
- Save the image with a right-click > Save/Save as.

Size= 100%

75% 50%

Improved light definition

When creating a light, settings for **light** and **shadow** have been separated to make the configuration easier. Moreover, the default values of the different settings (**intensity, shadow opacity, shadow diffusion**...) have been modified in order to provide lights that can be used directly.

ights definition	X	Lights definition
Lights Shadows		Lights Shadows
Default ambient light Default front light 1 Default front light 2 Sky	Type Ambient Point Directional Spot	The light casts shadows
	Sky	Shadow opacity :
	Window	Shadow transparency : Basic 🔹
Name: Sky	Enable	Shadow softness : 🖓
Color Intensity	11-12	Shadow resolution :
-0	1	Shadow quality :
Skylight parameters Type: Clear Sun colour:	•	
Sun intensity:	0.6 257.7*	Shadow tolerance: 0
Sun altitude: Sun calculator.	46.7*	Enable volumetric lighting
🗖 Dynamic update		Dynamic update
OK Cancel		OK Cancel Apply

What's new in TopSolid'WoodCam v6.15



This section describes the new features made to TopSolid'WoodCam in the version 6.15.

Part holding

Vacuum blocks for flat table

A new function is available to manage the positioning of vacuum blocks on a flat table.

It enables you to create an operation for positioning the blocks in order to send their coordinates via the postprocessor when the CNC control is equipped with a laser positioning system.

Activating this new function:

Position the blocks on the machine table:
 Use the Assembly > Include standard function.

<u>Note</u>: A component is available as standard in the **TopSolid'WoodCam** library for example (**TopSolid'WoodCam** > Machines > Pods > Vacuum block).



• Define the vacuum block set:

Use the **Part** > **Flat table vacuum blocks** > **Create/modify set blocks** function. Select the definition origin and the blocks to be taken into account in the operation.



Create the positioning operation for vacuum blocks:
 Use the Part > Flat table vacuum blocks > Create a positioning blocks operation function.

<u>Note</u>: A part must be positioned on the machine to be able to create the operation.

General	Operations list	Cutting conditions	
Ē.	TABLE		
Ι	🗄 👯 WCS Posa	ge pièce usinage 1 (Z-ATP10 (0) CTP1	0 (0))
	- 6 6 1 8	locks positioning:	
	Contraction of the local division of the loc	NAMES - DESCRIPTION AND A DESCRIPTION OF A DESCRIPTIONO OF A DESCRIPTION O	
	States and the second		
	and shift.		
achine t	ool : TopSolid'W	oodCam 5X Flat Table (MISSLER SOFTWARI	- TOPSOLID'WOODCAM) Material : Bois

This operation can then be edited from the list of operations with a **right-click** > **Edit**. Editing this operation allows you to switch between the blocks origin and the machine origin.

Positions origin		Blocks	
Designation	×	Y	
Vacuum block	360	220	
Vacuum block	220	470	
Vacuum block	1100	355	
Vacuum block	1100	720	
Vacuum block	1880	220	
Vacuum block	2020	470	

It is also possible to display the position of blocks as a table in a setup sheet.

TopSolid WoodCam		Part designation : Arc Machining file : Pods.wod Iso file : - Machine name : TopSolid'WoodCam 5X Flat Table			
Name	X position relative to ma	chine origin	Y position relative to machine origin	X position	Y position
Vacuum block	360		220	360	220
Vacuum block	220		470	220	470
Vacuum block	1100		355	1100	355
Vacuum block	1100		720	1100	720
Vacuum block	1880		220	1880	220
Vacuum block	2020		470	2020	470

<u>Note</u>: In order to save time, it is possible in the machine template to first insert the blocks and then create the vacuum block set. Then you just need to correctly position the blocks under the part and remove those that will not be used before creating the vacuum block positioning operation.

Improved rails and pods positioning

Exterior margins are now taken into account for the rails and pods positioning under the aperture scraps using the two options for extremities.

Configuration	×
Exterior margin outside part	: 20mm
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 (ape	rtures)
🔽 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 Cance	el

Result in version 6.15:



Nesting machining

Multi-part remachining after nesting

During the multi-machining process of a nesting, **TopSolid'WoodCam v6.15** turns over each part with operations on its side and bottom faces one by one. This automates the remachining of parts which have not been fully machined during the first machining phase of the nesting panel.



<u>Note</u>: This automation of the machining is based on CAD part definition. All settings of the **Machining** tab will thus modify this part remachining process.

(Two files if necessary, Lateral operations in second machining file, First and second positioning...)

Warning: The positioning for the part-by-part remachining is based on the stock of the CAD part definition. This means working with length and width overvaluations equal to zero.
Optimization by operation priorities

It is now possible to follow the machining order of operations defined in CAD to machine a nesting. To do this, a new option is available in the nesting analysis.

A part is create	ed for each family
perations processing Visualisati	on
Verticals operations only	
Choose operations to machine	
Cuttings	Counter-mouldings
🗸 Apertures	Grooves
User calibrations	Rabbets
Slope faces user machinings	Pockets
📃 Continuous 5 axis user machir	nings 🛛 📝 Holes
Mouldings	Aligned holes
Analysis	
of identicals parts	of a part
Do all parts	analysis again
Machining of all the parts of ident	icals parts of a part
Optimization	
With shapes proximity	10 M 10
With machinings approach/re	tract points
Shapes culting	
From the smallest to the large	st
From the largest to the smalle	st
With shapes proximity	
With machinings approach/re	etract points

Note: This sorting option by priorities can result in non-optimized tool changes and rapid movements.

Remachining of straight angles

The multi-contouring has been enriched with a new option to go up in the corners. This enables the straight angles to be machined directly in the multi-contouring in a single operation. The **Clearence method in straight angles** option is available in the **Main** tab of the contouring operation.



Optimization of milling toolpaths

The 2D optimization now takes into account the start and the end of open curves to calculate the nearest toolpaths.

Without optimization:



With optimization:



Design properties available for setup sheets

The custom properties of design documents (**File** > **Properties** in a *.top file) can now be edited in a setup sheet. In previous versions, only the properties of machining documents were retrieved.

<u>Note</u>: The edition of the template does not propose these properties because at this stage of the document template definition **TopSolid'WoodCam** does not know yet which properties will be present in relation to the design file used. Detailing texts must therefore be created manually in order to enter the desired text.

Example:

• In the .top file, user information setting via File > Properties:

Document properties	User information	
🗄 🎳 TopSolidWood properties	Description	Current value
	SJOB_NAME	Missler
	🚯 Customer	Pierre
	Project	Pr01

• In the setup sheet template, creation of texts with the prefix "\$FINISH_":

First page	Body pages	
- @	한Tool.user_ref1 (Ref.) 한Tool.type_str (Tool type) 한Tool.diam (Tool diameter)	*
\$Wo \$FINI \$FINI \$FINI	odPart.Processing SH_Project SH_Customer SH_\$JOB_NAME	

• Result after generating the setup sheets:

TonSolid	Part designation : Derriere	Customer: Pierre
Ιυμουιια	Machining file : Tests dgi-Coffre.wod	Project: Pr01
WoodCam	Machine name : TopSolid'WoodCam 5X Flat Table	Job Name: Missler

New Elem.FamilyKeyRef variable in the PDB

The new **Elem.FamilyKeyRef** variable is used to provide the name of the spindle family when defining a machine. This variable is now available in the PDB. This allows you, for example, to manage the plunge of drilling blocks of some machines (Example: Holz-Her).

<u>Note</u>: This new variable does not take into account the name of spindle families that have been created before the version 6.15. The new name must be rewritten or tool units must be reset so that the name is displayed in the PDB.

<u>Warning</u>: Resetting the spindle families reassigns their default name. (Example: **routing unit** for **TP1**).

Spindles family	
Name 500 Spindle axis> Z- (0, 0, -1)	*
Image:	🚔
Properties Family > Drilling Ramp 7	[ype> Ramp X+
Approach Tool change : Zonly (Wcs change : Zonly (Select approach Select approach
Retract Tool change : Zonly Wos change : Zonly	Select retract
Family spindles list Available spindles	Spindles TV1 TV2 TV3 TV4 TV5 TV6 TV7 TV8
ОК]	Cancel

Displaying PP words and work planes in the WCS list

New information is now available in the Work Coord. Syst. > List command:

- PP words
- Work plane

WCS(s) list						
		Part name: part 1 Ma	aterial: Hardwood			
Comment	Spindle direction	A TP1 (SPINDLE)	C TP1 (SPINDLE)	Origin WCS	Work plane	PP
WCS part 1	Z	0 (0)	0 (0)		XY Z-	
Z- A TP1 -90 (-90) C TP1 0 (135)	Z·	-90 (-90)	0 (135)		XZ Y·	
Z- A TP1 -90 (-90) C TP1 -90 (45)	Z·	-90 (-90)	-90 (45)		YZ X-	
Z- A TP1 -90 (-90) C TP1 180 (-45)	Z·	-90 (-90)	180 (-45)		XZY+	
Z- A TP1 -90 (-90) C TP1 90 (-135)	Z·	-90 (-90)	90 (-135)		YZX+	

Automatic creation of frames

It is now possible to deactivate the automatic creation of frames when positioning the parts. This is very useful for the machining of complex-shaped parts or parts coming from a STL file import. This option is available in the **Misc > Machines > Misc modifications** command and is used to position parts with a lot of faces more quickly.

Misc machine definitions			×
Working stations Post-processo	Options	Information	
Frames			
🛛 Align positionings frames			
Part positioning			
Create lateral frames			
Create frame under the part (f kinematic	s allow it)	
👿 Create frames on shape			
Create local frames for saw g	grooves an	d rabbets	
Create security block			
Adapt frame			
Saw machining			
🔲 Use the nearest frame			
🔽 Use sawing unit first			
	ОК	Cancel	

ISO process

The **ISO process** function directly runs the ISO program writing when the post-processor is already associated with the machine.

Machining time

During a multi-machining, the log displays the total machining time for the project. This time results from adding up the machining time of all the treated parts.

Project processing journal			X
Directory > D:\Projets-V6\2014\News 6.15\TSWoodCam\Meubles enfant Journal file > Coffre#PRJ08.log	Ŧ	Ĺ	2.
Project D:\Projets-V6\2014\News 6.15\TSWoodCam\Meubles Log file created on mardi 19 novembre 2013 09:45:28 ZmiWoodUtilDialTree#Project machining times Total time : 0:23:58	enfant	\Coffr	e.top

Online help

The online help has been completely rewritten to enable better clarity and understanding.



What's new in TopSolid'Planner & TopSolid'Quote v6.15



This section describes the new features made to **TopSolid'Planner** and **TopSolid'Quote** in the version **6.15**.

Panel brushes in the project settings

The **TopSolid'Planner** project settings allow you to preconfigure a project before starting it. These settings are used to set the parameter values, select the sub-components and materials, and now configure panels (matters, edges and laminates).

- Create a new **Design** document.
- In this document, create a panel on any part.
- Create a panel brush.
- In the **Panel(s)** box, select the panel created before.

<u>Note</u>: To be able to configure the panels in the **TopSolid'Planner** project, the panel brush must have the same name and use the same panel categories as the brushes to be configured in the **TopSolid'Planner** products.

- Rename this brush the same way as the one in the product included in the project.
- Also select the same **panel categories**.

Example: This **TopSolid'Planner** file has a panel brush named **Side panel** which uses the **TSW-Panel-front-right-edge** panel category.





In order to configure these panels using the project settings, the brush created in the configuration file is named **Side panel** and uses the **TSW-Panel-front-right-edge** panel category.





TopSolid 2014

What's new in TopSolid'Planner & TopSolid'Quote v6.15

- To modify the panel material for the setting file, right-click > Modify on the Support material categories line
- Without making any changes, confirm the categories used by clicking on **OK**.
- Select the material category and the default material to be configured.



• Repeat this operation to configure the default edge and default laminates to be used.

<u>Note</u>: In this configuration file, it is possible to create several panel brushes in order to configure several ones by default.





- Save this file in the **Configuration** folder and rename it *topzcfgagc_ConfigName.top*.
- Create a new **TopSolid'Planner** project.

As there is at least one configuration file, the **Project settings** window opens.

• Select the configuration you want in the **Configuration** drop-down list. The parameters and sub-component, material and panel brushes created are then included in the different tabs.

Changes can be made to brushes before starting the project.

Project settings	Configuration		Chocolat	e Dressing	
	Dimensions and options	s Styles	Materials	Panels	
	Designation	Туре			
	Side panel	Range 2			
	Internal panel	Range 2			
	Shelf panel	Range 2			
	4 edges panel	Range 2			
	Front and back panel	Range 2			

• Then insert elements in the project.

Panel brushes of inserted elements are then directly configured when inserted in the project.



It is then possible, without modifying the panel brushes, to design two different projects.



Improved ergonomics

Optional parameters

The optional parameters of a component are now available in the **Optional parameters** tab of the **Advanced mode** when the component is configured.

The	compo	nent	must	: have	Optional	
paran	neters	and/	or	Boolean	optional	0
paran	neters.					6

Name	Designation	Display unit	Use	Driver
<mark>.</mark> 9₽h	Height	mm	3	Yes
🖳 tsd	Top shelf distance	mm	2	Yes
🖳 fed	Front excess distance	mm	3	Optional
🧟 re	Right excess (for sliding door)		2	Boolean optional
🧟 le	Left excess (for sliding door)		2	Boolean optional
🧟 stg	Side to the ground		3	Boolean

Panels Optional parameters

• When configuring the component in the **TopSolid'Planner** project, open the **Advanced mode**.



• Open the **Optional parameters** tab.

Optional and Boolean optional parameters are then available. It is also possible to share them if the **TopSolid'Planner** document contains parameters.

Designatio	on	Value	U.	Share
Top shelf	distance	300	mm	
Front excess distance		60	mm	
V		Plinth	11. 11	
V	Side	Side to the ground		
	Right exc			
	Left exc	ess (for slidin	g door)	

Navigating between parameters

The navigation between parameters has been improved when configuring a component from **TopSolid'Planner**:

- A left-click in the Value box directly edits the parameter.
- The **Tab** key is used to validate the parameter being edited and edit the next parameter.
- The **Enter** key is used to validate the value being edited.

Designation	Value	U.
Height	2000	mm
Right depth	600	mm
Left depth	600	mm
Opening width	500	mm
Top shelf distance	300	mm
Front excess distance	60	mm

Online help

The new **TopSolid'Planner** online help is now directly available from the icon bar. An online help tailored to **TopSolid'Planner client** functions is also available from a **TopSolid'Planner client**.

Improved draft

Draft template

It is now possible to create **TopSolid'Planner** draft templates. When creating a draft, the draft template to be used can be selected.

- Create the different **TopSolid'Planner** draft templates.
- Save these files in the **Planner** sub-folder of the **Configuration** folder.

<u>Note</u>: The name of the draft template file will be the name displayed in **TopSolid'Planner** when selecting the template.

🔒 Planner	- 0	🥼 📂 🛄 🔻
Nom	Modifié le	Туре
🔊 01- Mise en plan FR.dft	30/10/2013 11:17	Document TopSolid'Draft
02- Draft EN.dft	30/10/2013 11:15	Document TopSolid'Draft

From a **TopSolid'Planner** project, launch the **Draft** function.

If there are several draft files, they are all displayed in a drop-down list.

ΟK	Template draft=	01 - Mise en plan FR	÷
		01 - Mise en plan FR	-
		02-Draft EN	

Note: If only one draft template is available in the configuration, it is automatically selected. If no draft templates exist, the **TopSolid'Planner** default draft template is used.

• Select the template to be used in the drop-down list or confirm with **OK** if the template you want is the first one in the list.

The draft of the TopSolid'Planner project is then created.



Creating additional drawings

Additional drawings can now be created in a **TopSolid'Planner** draft. These drawings are then created from a project view, but it is possible to trim this view.

- From the **TopSolid'Planner** 3D project, open the **Draft I** tab, then select the **Trimming curve** function.
- Draw the curve enclosing the element to project.

<u>Note</u>: The curve must be closed.

- The **Stop** option is used to close the curve directly. STOP
- The **Right angle** option is used to draw horizontal or vertical lines. RIGHT ANGLE **
- The Free option is used to draw any lines. FREE **
- Enter a length in the **Segment length** box, and then confirm with **Enter** to set the length of the drawn segment.



- The **Rectangular** option is used to create a rectangular trimming directly. RECTANGULAR



• Enter the name of this trimming, and then click on **OK** to confirm.

OK Designation: Dressing L

- From the **draft** document, select the **Drawing** function.
- Set the **hook point** of the new drawing, then click on a point in the draft to position the new drawing.



•



• Select the direction to be used to draft the project.

X+ X- Y+ Y- Direction:



Select the Stop button to draft the whole project according to the selected direction.



- Select the Trimming curve button to trim the view to a curve you drew earlier.
 TRIMMING CURVE
- From the **Trimming curve** drop-down list, select the trimming curve to be used.

trimming curve=	Dressing L	•	
-----------------	------------	---	--

<u>Note</u>: You can also click on one of the trimming curves directly from the 3D project.



A new view is then created according to the selected direction. This view only contains the elements inside the **trimming curve**.

	20131028-102732 X- view Document 11/13 28/10/2013 Dressing

Select the New drawing option to create a new drawing. NEW DRAWING
 Select the Quit option so that no new drawing is created. QUIT

Fit screen and zoom on drawing

Two new functions are now available from the **TopSolid'Planner** context during drafting in order to improve navigation within the document.

Fit screen

• Click on the **Fit screen** icon to perform a global zoom on all drawings included in the draft document.

Zoom on drawing



- Open the Zoom tab icon and select the Zoom on drawing function.
 Select a drawing of the document in order to zoom in on the drawing.
- Select another drawing to zoom in on it.
- Exit the function by clicking on **Quit**.

Note

The Note function has been added to TopSolid'Planner in the draft.

- From a TopSolid'Planner draft document, select the Note icon.
- Enter the note text in the **Texts** field.

Texts	Drawer total openning Openning Push - Lash : Charge 150Kg	
-------	---	--

- Set the note parameters, then click on **OK** to confirm.
- Position the note on a drawing with a **left mouse click**.
- Select the elements to be indexed by **left-clicking**, then confirm with the **Stop** button. STOP



Openning Push - Lash Charge 150Kg

• Select the **No leader** option **NO LEADER** so that no index is positioned with the note.

The Note function will automatically restart to position a new note.

• Exit the function by pressing the **Esc** key so that no new note is placed.

Notes:

- Note default setting values can be set in **Tools > Options > Text/Note > Attributes/Initial values**.
- To set the **Creation layer** and the **Text height**, the document properties must also be modified in **File** > **Properties** > **Text/Note**.
- Predefined note texts can be created in Tools > Options > Text/Note > Predefined texts/notes.
- As TopSolid'Planner does not use layers, it is recommended to select **Creation layer**: **No fixed layer**.

Miscellaneous

Ground projection

Grounds are now projected in the **TopSolid'Planner** draft views.



<u>Note</u>: However it is possible not to project grounds during the **TopSolid'Planner** drafting by creating in the template views an exception that excludes all elements with the **Ground** designation.

View modification	Part exclusion	> EXCLUDED -
Coordinate system Exceptions Other options View	Smooth edges	> NO EXCEPTION -
Ground BY CRITERIA Exclusion	Hidden lines	> NO EXCEPTION -



Improved scale factor calculation

During the **TopSolid'Planner** drafting, the scale factor is now calculated in order to provide views that are more adjusted to the drawing.



<u>Note</u>: Scale factors are calculated based on the drawing dimension, as well as the dimension of the view(s) and title block.

A scale factor nearest lower to the calculated one is selected in the **Scale factors** list.

This list is available in a **Design document** > **Tools** > **Options** > **Multi-draft** > **Scale factor**.

0.01	E
0.012	
0.014	
0.016	-
۲ III I	

TopSolid'Planner client administration tools

TopSolid'Planner library export

When generating the **TopSolid'Planner client** installation, it is now possible not to export the libraries provided with **TopSolid'Planner**:

- Decoration
- Document template
- Textures
- From the **TopSolid'Planner** context

, start the **administration** tool.

The **TopSolid'Planner** libraries are installed during **TopSolid'Planner client** installation.

- Open the **Generation** section.
- Open the **Configuration** tab. Configuration
- Uncheck:
 - **Include decoration library** so that the decoration library is not exported.

Include decoration library

Decoration ((358)	
÷@	Bathroom accessories	
+*	Botany	
+	Construction	

<u>Note</u>: Excluding the **TopSolid'Planner** decoration library allows you to save about 200Mb.

- Include template library so that document templates are not exported.

Include template library



<u>Note</u>: If the document template library is not installed and the component libraries do not contain document template libraries, it will not be possible to create new projects on the **TopSolid'Planner client** station.

 Include textures library so that the Coating wall and floor textures and the Colour chart are not exported.
 Include textures library

Coating w	all and floor (128)	
Colour cha	irt	
: 	Bricks	
+	Colors	
+	Concrete	

• Then generate the **TopSolid'Planner client** installation normally.

Generate

When installing TopSolid'Planner on the client station, the unchecked libraries will not be installed.

TopSolid'Quote

Setting the quotations in Tools > Options

Quotation parameters have been relocated in **Tools** > **Options** to have a more optimized organization of functions.

• From the **Tools** tab, start the **Options** function.

General Bill of material Conditions of sale Currencies and exchange rates

The **Conditions of sale**, **Currencies and exchange rates**, **Bill of material** and **Reseller** options are then available.

<u>Note</u>: Depending on the license used and the rights of the **TopSolid'Quote** user, these functions may not be available:

- **TopSolid'Quote base license**: the **Bill of material** options are not available.
- TopSolid'Quote base license + non-admin user: the Conditions of sale, Currencies and exchange rates and Reseller options are not available.

Real image in TopSolid'Quote

When sending a project to **TopSolid'Quote**, it is now possible to generate previews of products in the project. This will display in the quotation images of products as they have been configured with their materials, dimensions, edges, etc.







N°	Picture	Q.	Unit. €	Total €
1		3	18,71	67,14
2		2		

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Configuring the components

<u>Note</u>: In order not to generate images on all components of the **TopSolid'Quote** bill of material, images are only generated on components with specific presentations.

- Open the library component for which an image must be generated.
- Configure the view and the rendering to be used to generate the image.
- Start the **Tools** > **Presentation** function.

🚑 Presentation

• Rename this presentation *BOM*, then click on **OK** to confirm.

OK Presentation name: BOM COMMENT

<u>Note</u>: This presentation allows you to set the camera and the rendering mode used when generating the image.

If this presentation does not exist in the component, the image is not generated.

• Save and close this file.

Configuring the BOM

• Using a text editor, edit the BOM model file used by TopSolid'Quote.

Insert the following function after the last function:

NAME=IMAGE "DEF=<IMAGE_PATH|FILE|IMAGE=4>" TYPE=STRING ALIGN=LEFT TITLE_ALIGN=LEFT WIDTH=0.015 VISIBLE=YES

• Then insert the text ,IMAGE after the GROUP_BY and ORDER_BY lists.

```
GROUP_BY {
```

```
LEVEL, DESIGNATION, REFERENCE, DESCRIPTION, PART_TYPE, PART_SUP, IMAGE
```

ORDER BY {

3

```
LEVEL, DESIGNATION, REFERENCE, DESCRIPTION, PART_TYPE, PART_SUP, IMAGE
```

- Save and close the BOM model file.
- Launch **TopSolid'Quote**.
- Open the BOM parameters using **Tools** > **Options** > **BOM**.
- ом. 🔳
- Click on the Load a bill of material model file button, then select the previously modified BOM file.
- Match the TopSolid'Quote **picture path** to the **IMAGE** column of the BOM.

Unit	Column		Action
Picture path	IMAGE	-	
	a an anna an ann an ann an an Inneanann a sa an	مملية	-المصنحيا

Confirm the parameters with Section 2.

- Apply the new BOM to the existing catalogs using the **Copy BOM parameters to some catalogs** function.
- Select the catalog to which the BOM must be applied, then confirm with Select the catalog to which the BOM must be applied.

Creating the quotation

• From the **TopSolid'Planner** project, generate the quotation.

Note: A folder with the name of the TopSolid'Planner project is created in the project folder.

This folder contains a **pngs** sub-folder in which images are stored.

• From the **Priced bill of material** tab of the quotation, the path of the image thus generated is shown in the **Image** column.

COUNT	DESIGNATION	REFERENCE	IMAGE	Prix (€)
⊡ . <mark>1</mark>		20131112-141850		492,79
	2 fixed horizontal separation kit First side Following right Recessed door kit	2 fixed horizontal separation kit First side Following right Recessed door kit	C:\Users\fra\Documents\Projets\Planner\20131112-141850\pngs\19.png C:\Users\fra\Documents\Projets\Planner\20131112-141850\pngs\38.png	72,23 49,86 297,37 73,33
 Launch 	h the Make a quotation f	function.	Following right Supplier: Dressing Room catalogue Catalog: 2014 Catalog	
The real p of the Quc	previews of products are ote detail.	shown in the product	: summary	

Launch the Quote preview function.
 Quote preview

The real images generated are then used for product previews.

N°	Sketch	Product	Q.	Unit €	Total €
1		First side	1	3,40	3,40
2		Following right	3	18,71	56,13
3		Recessed door kit	2		

Multi-user management

Working in **Client/Server** mode (used to create quotations on a database shared by several **TopSolid'Quote** stations) has been improved to help increase data security.

Refresh quotations list

When several users work on the **quotations**, the **Refresh quotations list** function updates the quotations and shows:

- The new quotations (or the new versions);
- The deleted quotations;
- The quotation status changes.

- Edit/Delete a quotation

Editing and deleting quotations has been secured in order not to be able to edit or delete a quotation opened by another user.

The message indicates the **user** who is editing the quotation, as well as the **computer** to which he/she is connected.

	TopSolid'Quote	×
Ô	This quotation is already opened in another session. Please ask the following user to close the quotation:	
	User: John Computer: PORTABLE-FRA OK	

<u>Note</u>: When a quotation is being edited by another user, it is also impossible to create a new version of this quotation.

- Edit/Delete customers

Editing and deleting customers has been secured in order not to be able to edit or delete a customer that is already being edited by another user.



Moreover, if a user tries to edit/delete a customer that has already been deleted by another user, a message is displayed to indicate that this customer no longer exists and the customer list is refreshed.



- Connection to a user account

If a user account is already used on another **TopSolid'Quote** station, using, deleting or modifying this account from another station has been secured.

•	TopSolid'Quote	
B	This user is already logged on the following machine, please connect with a different account:	
	PORTABLE-FRA	
	ОК	

Displaying the product summary

A summary of the product selected in the quotation is now displayed.

- From a quotation, open the **Quote detail** tab.
- Select a product in the basket.

A new zone below the basket displays a summary of the selected product, including:

- Designation
- Supplier
- Catalog
- Preview
- Comment

First side -----

Supplier: Dressing Room catalogue Catalog: 2014 Catalog

Comment: To provide at the second delivery.



What's new in TopSolid'SheetMetal v6.15



This section describes the new features made to **TopSolid'Sheetmetal** in the version **6.15**.

General

Machining parts search – New function

Function use

Simplification of the **Search machining parts** function.

Activating the function

A new button has been added: Add the selection of parts in tasks manager.

• Make your selection and press the button.

h of parts	of the second second	Statements of the local division of the loca	Concession of the local division of the loca							
	Folders	Clear list of folders	IVERS •	Targets C.\Dossiers_P C.\Dossiers_P C.\Dossiers_P	atricia/ANNEE_2013_ atricia/ANNEE_2013_ atricia/ANNEE_2013_ Remove Ret	TESTS_615\0 TESTS_615\0 TESTS_615\0 move al	DUT_DXF (S DIVERS (Su			
General informatic	m	. New contraction of the contraction of the								
Matter Not set		Thickness	M	achin Not set	•]	Cut gaz	Non specifie	d gaz	•)	
Additionnal criterio Designation	n Set	Plane	Command	Customer	2	Date between	1	and		-
		Part name filter (Fiv	*abodt - * for all)		Search					
sults		r ar name mer (Ex.			oodere.					
lame		Path			D. Set	Plane	OF	Customer	Mat	
Coch		C\Dostiers Patricia\ANNEE	2013 TESTS 615\CUT (10/F					acies	
ESSOUS-CIGALE-VA	LIDE pch	C\Dossiers Patricia\ANNEE	2013 TESTS 615\CUT 0	DØF					acier	
ocument3.pch		C:\Dostiers Patricia\ANNEE	2013 TESTS 615\DIVER	IS VERY CUT					acier	
wil och		C\Dossies Patricia\ANNEE	2013 TESTS 615\PIECE	ST5000					acierii	n
ni2 nch		C\Dossiers Patricia\ANNEE	2013 TESTS 615\PIECE	ST5000					acierii	n
ni3 nch		C\Dossiers Patricia\ANNEE	2013 TESTS 615\PIECE	ST5000					acierii	n
hold och		C\Dostiers Patricia\ANNEE	2013 TESTS 615\PIECE	ST5000					acierii	n
1.41.008 & LONGER	ON GALICHE och	C\Dossies Patricia\ANNEE	2013 TESTS 615\DIVER	ISVOXE CUT	KIT H	G PI 2020	0 F 2020	MISSIER	ວ່າ	
1.42.006 B CHAPE B	LOCAGE och	C\Dorrier: Patricia\ANNEE	2013 TESTS 615\DIVER	ISVDVE CUT	KIT H	G PL 2020	0F 2020	MISSIER	alu	
1-43-001 A TOLE SU	PPORT BASCULEUR oct	CADossiers Patricia\ANNEF	2013 TESTS 615\DIVER	ISVDXF CIT	кіт н	G PI 2020	D DE 2020	MISSIER	alu	
75 file(s) part fo	und				Preview					
Mass of selected	parts (817.0483g	Mass of sel	ected parts with quantities	817.0483g			00	00]	
Remove	the parts from the selectio	n Clear the s	election 📝	Show part Preview	Box dime	ensions 1990.1000mr	n Y	261.882	8mm	
	Ade	the whole list of parts in tasks	manager Add Bas or	deption of parts in tasks ma	earer] [Car	- Inne				
	Ad	a one writtle nat of parts in tasks	manager Add the se	nection of parts in tasks ma	Lar	ICEI				

Insert parts - Select multiple files

Function use

In the **Insert part** function, the **Explorer**, **Search** and **Select from tasks manager** functions allow you to select multiple files.

Activating the function

- **Insert part** function, use (for example) the **Explorer** function.
- Select multiple files.

The first part of the selection appears when the mouse hovers over it, the other parts of the selection appear in the thumbnail previews.



6 Parts to insert		120		
Look in	: 🌗 PIECEST500	0	- 🗿 🏚 📂 🗔 -	
a.	Nom	*	Modifié le	Туре
and the	Proj1.pch		25/09/2013 16:22	Document Top!
Emplacements	Proj2.pch		25/09/2013 16:22	Document Top
recents	Proj3.pch		25/09/2013 16:22	Document Top
Bureau	👜 Proj4.pch		25/09/2013 16:31	Document Top!
Bibliothèques				
Ordinateur				
(i) Réseau				
	•	III		•
	File name:	"Proj3.pch" "Proj1.pch" "Pr	oj2.pch" 🔹	ОК
	Files of type:	*.pch	•	Cancel



Ergonomics – Delete – Modify 💷

Delete Function

New options have been added in the combo.

Option to delete the following directly:

- stops
- trap doors
- unloads
- micro-attaches
- punched attaches

Example on the punching menu

Delete	PUNCHING ALLOCATIONS	•
	PUNCHING ALLOCATIONS	
	FREE CONTOURS	
	SCHEDULINGS	
	DEBURRING	
	OTHERS	
	STOPS	
	TRAPP DOORS	
	UNLOADS	
	MICRO ATTACHS	N
	PUNCHED ATTACHS	5

Example on the cutting menu

Delete	CUTTING ALLOCATIONS	•
	CUTTING ALLOCATIONS	
	LINKING PATH	
	LEAD IN/LEAD OUT	
	ADDITIONNAL CUTTING	10
	COMMON CUTTING	
	INDUCED MACHINING	
	FREE CONTOURS	
	OTHERS	
	STOPS	
	MICRO ATTACHS	

Improved detection on evacuation functions

After having refreshed the sequence view using the function, the last strike of the sequence concerning the evacuation can now be detected as well as the text.



Improved view of micro-attaches and punched attaches

Option to view a dashed circle around the attaches.

Activating the function

- Adjust the settings in Tools > Options > Display Options > Sequences.
- Check the **Draw circle around micro attach** box.
- Enter the attach height multiply factor.

Application configuration	・・・・ ひゃうちょう 目 大学日 山
Configuration	Sequences
Colors	Show the punching sequences
Importation	Show sens of punching sequences
Cut Westing Simulation	Set minimum size for sequence arrow
Post processor Tasks manager	
Sequences	Unique or micro arteachs unre witch or micro artachs Unique witch or micro artachs Unique vision arteach Multiply factor of height
Westing	

- To delete the micro-attach, proceed as follows:
 - 🛛 Delete 📝 .
 - Select Micro attaches from the list.
 - Click on the circle around the element.



Improved Modify function

After having refreshed the sequence view, with the evacuation modes now detectable and micro-attaches easily distinguishable, it is possible to **click directly on the STOP**, for example, to change it without having to change all of the machining.



Example of changing the STOP with the possibility to directly change the clearance values.

By clicking on the trap door status text or symbol, it is possible to directly change the shift or the position of the trap door.



Example of changing the TRAP DOOR.



For machines able to unload parts, by clicking on the text, the system displays the vacuum lifter and prompts you to select or change the positions of the vacuum cups.



Example of changing the UNLOAD.

Ergonomics – Creation of micro-attaches

Function use

Modification of the micro-attach creation function to make it more flexible to use.

Manual picking	-	ATTACH TOOL = carre 10 BRITTLE = NO Select a segment
Manual picking		
Position value		
At extremity		
On corner		
Distribute on segment		
Distribute on contour		
At one punch from border		
	Manual picking Manual picking Position value At extremity On corner Distribute on segment Distribute on contour At one punch from border	Manual picking Manual picking Position value At extremity On corner Distribute on segment Distribute on contour At one punch from border

Using the function

Now, the choice of the position of micro-attaches is offered by a COMBO.

The status of the combo as well as the most common values are stored during the session.

Entry fields such as "Position value" or the number of micro-attaches distributed are directly accessible.

Positionning	Position value 🗸 🗸	ATTACH TOOL =	carre 10 BRITTLE = NO	Value of position in	mm 50	Select a segment
Positionning [Distribute on segment 🔹 ATT.	ACH TOOL = carre 10	BRITTLE = NO Number=	N -	N + Keep existing= NO	Select a segment
<u>Position</u>	ning by "distribute or	<u>n contour"</u>				
1	Positionning Distribute on contou	✓ ATTACH T	00L = [carre 10] BRITTLE	= NO Number= 1	<u>N -</u>	N + Select a contour
		Additiona	l options are avai	able with the	e >> button.	

Additional options

GO BACK Keep existing micro= NO 🖅 On FULL PROFIL 🖅 First attach position= START CONTOUR 🖅 Select a contour

Tasks manager

Cancel a waste (extracted)

Function use

In the tasks manager, you have the option to cancel an extracted waste in order to return to the nesting as it was before the extraction was carried out.



Activating the function

- Use the wrench, click on
- Select Cancel extracted waste from the drop-down list.

NESTING ORDER	-
NESTING ORDER	
DIMENSIONS OF SHEET	
INFORMATIONS DISPLAY	
SHEET MACHINED NUMBER	
PLANAR SHEET	
CANCEL EXTRACTED WASTE	

The sheet returns to its original size.

The waste marking text as well as the waste have also been removed from the matters to use in the tasks manager.

Note: It is possible to use the Undo function.

Manual nesting (from the Tasks Manager)

Function use

Option, with a new function, to manually nest parts from the Tasks Manager. The quantities of parts as well as the sheet are managed by the manager.

TASKS MANAGER

Activating the function

 From the Tasks Manager, in the Parts to machine tab, check the parts to be nested that must imperatively have the same matter (or the same type of matter), the same thickness and be machined on the same machine.

Parts to r	nachine Matters to use Nests to	o use	Parts do	ne	Nestings do	ne Orders o	one															
	Ref.	Des.		S	Plane	OF	Customer	Mat.	М.	Th.	Occ.	Q.	N	Q Mx.	Fam.	Comp.	Prio.	Date	Mac.	Gaz.	T.,	*
	Proj1							acier inoxydable		1.5000mm	1	10		0	-1	No	0	25/09/2013	trumpf5000_sheet			-
1	Proj2							acier inoxydable		1.5000mm	1	10		0	-1	No	0	25/09/2013	trumpf5000_sheet			
-2	Proj3							acier inovydable		1.5000mm	1	10		0	-1	No	0	25/09/2013	trumpf5000_sheet			
$\boldsymbol{\wp}$	Proj4							acier inoxydable		1.5000mm	1	10		0	-1	No	0	25/09/2013	trumpf5000_sheet			
8	Pièce_2020					OF_ 2020		acier		4.0000mm	1	20		0	-1	No	0	05/09/2013	trumpf5000_sheet			
9	Pièce_acier_inoxy_GazNS							acier inoxydable		1.5000mm	1	1		0	-1	No	0	26/09/2013	bystronic			
9	Pièce_Inox_Gaz02						PP	acier inoxydable		1.5000mm	1	1		0	-1	No	0	27/09/2013	bystronic			
	Pièce_Alu_GazN2							Alu		1.5000mm	1	1		0	-1	No	0	26/09/2013	bystronic	N2		
1.	Pièce_Gaz02							acier		1.5000mm	1	1		0	-1	No	0	26/09/2013	bystronic	02		=
	Pièce_Gaz02						PP	acier		1.5000mm	1	1		0	-1	No	0	27/09/2013	bystionic	02		
+	Pièce_Alu_GazN2						PP	inox		1.5000mm	1	1		0	-1	No	0	27/09/2013	bystronic	02		
	DESSOUS-CIGALE-VALIDE							acier		1.5000mm	1	1		0	-1	No	0	15/03/2013	mazakspacegear	02		
	TOUR EFFEL10							acier		1.5000mm	1	1		0	-1	No	0	15/03/2013	mazakspacegear	02		
	Pièce_1010							acier		2.0000mm	1	1		0	-1	No	0	17/09/2013	trumpf5000_sheet			
	TAUREAU							acier		1.5000mm	1	1		0	-1	No	0	15/03/2013	mazakspacegear	02		
	Pièce_1010					OF_ 2020		acier		4.0000mm	1	1		0	-1	No	0	05/09/2013	trumpf5000_sheet			
	Pièce_2020							acier		4.0000mm	1	10		0	-1	No	0	05/09/2013	trumpf5000_sheet			

- In the Matters to use tab, check at least one entity with the matter and thickness of the previously selected parts. There may be other entities checked that do not have the characteristics of the parts.
- Specify an order name and check the boxes for the automatic numbering of orders (or automatic numbering of nestings).

Parts to r	machine Matters to use Nests to us	se Parts done	Nestings done	Orders done						
	Matter M.	Th. D	Length	Width	Q.	Free qty	Waste	Stored sheet	Area	Date
	304L-08-3000-1500	1.0000mm	3000.0000mm	1500.0000mm		Yes	No	No	4500000.000mm²	06/12/2011
${\boldsymbol{\wp}}$	✓ acier	1.5000mm	2500.0000mm	1250.0000mm		Yes	No	No	3125000.000mm ²	06/12/2011
2	304-PG220-08-3000-1500	1.0000mm	3000.0000mm	1500.0000mm		Yes	No	No	4500000.000mm ²	06/12/2011
9	acier	2.0000mm	4000.0000mm	2000.0000mm		Yes	No	No	8000000.000mm²	07/04/2013
1	🗹 Alu	1.5000mm	4000.0000mm	2000.0000mm	48	No	No	No	8000000.000mm ²	08/04/2013
1.1	304L-08-3000-1500	1.0000mm	3000.0000mm	1500.0000mm		Yes	No	No	4500000.000mm ²	09/04/2013
	✓ acier	1.5000mm	2500.0000mm	1250.0000mm		Yes	No	No	3125000.000mm²	09/04/2013
•	304-PG220-08-3000-1500	1.0000mm	3000.0000mm	1500.0000mm		Yes	No	No	4500000.000mm ²	09/04/2013
11.	🗹 acier	2.0000mm	4000.0000mm	2000.0000mm		Yes	No	No	8000000.000mm ²	09/04/2013
<u>e 1</u>	🗹 Alu	1.5000mm	4000.0000mm	2000.0000mm	50	No	No	No	8000000.000mm ²	09/04/2013
	acier	3.0000mm	2500.0000mm	1250.0000mm		Yes	No	No	3125000.000mm ²	16/04/2013
	🗌 Alu	2.0000mm	3000.0000mm	1500.0000mm		Yes	No	No	4500000.000mm ²	16/04/2013
	✓ acier	4.0000mm	2500.0000mm	1250.0000mm	10	No	No	No	3125000.000mm²	05/09/2013
	acier	2.0000mm	2500.0000mm	1250.0000mm	2	No	No	No	3125000.000mm ²	17/09/2013
	 acier inoxydable 	1.5000mm	3000.0000mm	1500.0000mm		Yes	No	No	4500000.000mm ²	25/09/2013
	🗹 inox	1.5000mm	3000.0000mm	1500.0000mm		Yes	No	No	4500000.000mm ²	27/09/2013
	🗹 acier	4.0000mm	2000.0000mm	1000.0000mm		Yes	No	No	2000000.000mm²	30/09/2013

• Press the Manual Order button.

Order name Use automatic ISD counter to set file names (per machine) Nest root order name LANCEMENTS_9 Saving directory) C:\Dossiers_Patricia\ANNEE_2013_TESTS_615\LANCEMENTS\	Tunings Linked actions for nesting Production management system
Sorting additional criterions for automatic placement References Designations Set Plane OF Customers Families Dates Cut gaz	Import from PMS Export to PMS
Use completion parts Yes No Nest parts by chronologic dates	Clear tasks done
Execute Order Manual Order Create opcard before nest	
Validate Validate and Close	Cancel

What's new in TopSolid'SheetMetal v6.15

In the case where there are several matters selected (having the characteristics of the parts), a dialog box prompts you to choose the exact nesting sheet.

	acier	4.0000mm	2500.0000mm	1250.0000mm	10	No	No	No	3125000.000mm²	05/09/2013	
exact	acier	4.0000mm	2000.0000mm	1000.0000mm		Yes	No	No	2000000.000mm²	30/09/2013	
	n Matter fil	le Preview		Add this pai	***					Creed	
				Add this ma	tter					Cancel	ļ

Matter typ

Q. Free qty Waste Stored sheet Area

Matter selection of tasks manage

D.. Length

Width

General information

Matières compatible:

Matter M. Th.

Matter acie

A new nesting document is created on the given machine and with the desired sheet.

In the case of a non-rectangular sheet, the option to make it symmetrical and/or turn it is given. Parts are inserted from the thumbnail previews.

• Once manual nesting is complete, manually check the nesting so that the quantity of manually nested parts is decreased in the Manager and that the sheet is also decreased in the **Matters to use** tab.

The order appears in the **Orders done** tab.

TopSolid 2014

Thickness 4.0000mm

Parts

.....

1 . . .

171

Date

8

TopSolid'Punch – Punching

Maximum overflanking for automatic punching

Function use

Option to define a maximum overflanking value for machining on lines.

Application configuration	
Configuration Subset information Colors Shortcut key General Importation Punch	Adjustments Ignore and induce segments of length lower than (0.5000mm Circles of diameters (separed by ;) Notching Not creation of notch but allocate each line Init at box triangular notch using rectangular/square tool Tackers to use that disk to sea that a face the second
Alcoation Automatic allocation Automatic allocation	Lines
Post processor Display options	Other searches Macros search No search Macros search No search Special tools on external lines Special tools on external arcs Special tools on external lines Search machinings of type line to line - Mini, number of lines to put in relation Trapp door evacuation Trapp door evacuation Parts Dimensions : X mini [100.0000mm Parts Dimensions : X mini [100.0000mm Y mini [100.0000mm Put Stop if trapp too small 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 Trapp door * Maximum overflanking for notching [6.0000mm Waximum overflanking for lines [8.0000mm] for trumpf5000_sheet *

Activating the function

- Go to Menu > Tools > Options > Punch > Automatic allocation > Adjustments.
- Go to the Lines > Allocate section.
- Enter the chosen value into the **Maxi. tool width** field.

Manual punching – Automatic tool mounting

Function use

Option to automatically mount a tool selected from the list on the turret throughout the manual machining function. The turret will remain in manual mounting for the other tools.

Activating the function

• Tool box list: tick the Mount this tool automatically if necessary box.

ools square - rec	tangular		23
rect76.2x5	76.2000mm 5.0000mm Th	Put On (indexing) (Splitter)	
rect50x5	50.0000mm 5.0000mm Th	Put On (indexing) (Splitter)	
carre 35	35.0000mm 35.0000mm Th	(0.000°)	
carre 30	30.0000mm 30.0000mm Th	(0.000°)	
carre 15	30.0000mm 30.0000mm	(0.000°)	
rect30x5	30.0000mm 5.0000mm Th	Put On (indexing) (Splitter)	
rect30x3	30.0000mm 3.0000mm Th	(0.000°) (Splitter)	
carre 25	25.0000mm 25.0000mm Th	(0.000°)	
carre 20	20.0000mm 20.0000mm Th	(0.000°)	
rect20x5	20.0000mm 5.0000mm Th	(0.000*) (Splitter)	
rect15x5	15.0000mm 5.0000mm Th	(0.000*) (Splitter)	
carre 10	10.0000mm 10.0000mm Th	Put On (indexing)	
rect10x5	10.0000mm 5.0000mm Th	Put On (indexing) (Splitter)	
rect10x4	10.0000mm 4.0000mm Th	(0.000°) (Splitter)	
carre 5	5.0000mm 5.0000mm Th	(0.000°)	
🕖 Mount automati	cally this tool if necessary		
Rect. an	d Square Decreasing order	All Ok	

Note: The checkbox status remains the same throughout the TopSolid session.

Management of ZN35 tools

Function use

Option in tool management to define a priority security area in relation to that of the station where the tool is mounted. This function has been developed for ZN35 tools which are intended to strike in an area closer to the clamps than traditional tools, avoiding a clamp-shift.

Activating the function

• Tools management > check the Clamps zone box.

A dialog box opens allowing access to the accessibility area of tool parameters.

Tools caracteristics : RECTANG	LE			
molette rect10x4 rect10x5 rect20x5 rect20x5 rect30x3 rect30x5 rect4x3 rect50x5 rect76.2x5	Definition Tool name Manufacturer reference Aucun Length (mm) 30 Width (mm)		Accessibility area of tool Accessibility to clamps (mm)	2
Add Remove Modify	Prefered stations Ist Station 1 - (A) Best 2sd Station 1 - (A) Camps zone Tool text Tool t	Text Macro	$ \begin{array}{c c} a & 0 \\ b & 0 \\ c & 0 \\ d: & 0 \\ h & 0 \end{array} $	
Remove All	<u>۲</u>		Ok Cancel	
Copy params			<u>e</u>	
Close and update	Cancel Punch C	Conditions		

In this example, only the height has been entered. Other values that have remained at 0 will be defined by the station **Management > Machine > Stations definition > Station accessibility area**.

The parameters will be:

- a=97
- b=97
- c=97
- d=97
- h=35

Support Stations definition 25 Statio 1 2 3 4 4 5	Station accessibility area (mm) a 97 b 97 c 97 d: 97 d: 97	ation on station
7 8 9 10 11 12 13 14 15 16 17 7 8 9 9 10 11 12 13 14 15 16 17 7 8 9 9 9 10 10 11 12 13 14 15 16 4 4 4 4 4 4 4 9 10 10 11 12 13 14 14 15 14 14 14 15 14 14 15 14 14 15 15 16 16 16 17 17 18 19 10 10 11 11 12 13 11 12 13 11 14 15 16 16 17 17 18 19 19 10 10 11 11 12 13 11 14 15 16 17 17 18 19 19 10 11 11 12 13 11 11 12 13 11 12 13 11 14 15 17 17 17 17 17 17 17 17 17 17 17 17 17	h 72 Accessibility Control Punching machine accessibility (mm) Display accessibility area Horizontal mini 0 Vertical mini 0 Horizontal maxi 0 Vertical maxi 0	(Modulo 90* ulator) in degrees emove ALL
Management of the Deburring function

Function use

Option to carry out deburring of a geometry of the part already machined with a tool.

Activating the function

 Adjust the relevant tool settings in Management > Tools

The new **Deburring** feature is available for round, square and rectangular tools.



In contrast to other machinings, deburring is drawn in cyan blue with the shape of the tool (round, etc.) at the beginning and end of the geometry.

Only the line or arc are drawn.









<u>Use</u>

Three operating modes are available:

- Segment mode
- Full contour mode
- Part of contour mode



Segment mode

When the segment to be deburred is selected, it is possible to restrict the deburring to the length between two punched attaches. An overflanking value can also be entered.



OK Offset = 2.5

Full contour mode

When the contour to be deburred is selected, an offset value is proposed.

This value is equal to half the width of the tool already positioned.

 If you subsequently wish to change the offset value, use the wrench and select the "cyan blue" colored line. It is also possible click the geometry where the main tool is positioned to change the offset value of the deburring tool.

Allocation caracteristics	Allocation caracteristics
Current tool Deburning	Current tool Deburring
Description Designation: [rect75.2x5] Monst on human [1,10] (indexing)	
Punching parameters Nibbling step (0-automatic) 73.073:	Deburing tool BAVURE
Forced covering between 2 punches	
Start overflanking 2.500mm	-
End overflanking 2.500mm	Offset 2 50mm
By tool width O By set width	
Operation type: [Line nibblin Optional machinin Geometrical line length: [975.0m Total machining length: [980.0m Invert sequence Use supporting curve Machined with the sheet s Optimize punch in full	2 2 3 4 5 5 7 7 7 7 7 7 7 7 7 7 7 7 7
Stop type 🖲 None 🕐 Temporised 🕐 U	r .
Trapp door number Trapp door number Shift along X 0.00mm Shift alon Trapp door tempo 0.000m	
🛞 Open And Close 👘 Open W	4
Punching Mode None	
Ok. a	Ok Cancel

• To remove the deburring function, use the **Delete** function and select **Deburring** from the list.



Part of contour mode

Click on the first segment of the contour.

The offset value is retrieved according to the tool present on the geometry.

- Reverse the direction if necessary.
- Change the offset value if necessary.
- Select the last segment to be deburred.

<u>Reminder</u>: Using the wrench allows you to change the offset value at a later stage.

Simulation – Operations order

In the case where there is no stop/unload/trap door on the geometry, in the operations order and manually, the deburring tool must be placed last.

However, in the case of deburring limited to attaches, the main sequence will be performed. As the part is held by the two attaches, deburring will be performed on the machined section (with possible overflanking). The last two attach strikes will be given to release the part.

I	Operations order	
	Operations display : 💿 Simplified	
	SIMULATION → Area1 - rd10 [0.00*] (T5] - rd5 [0.00*] (T4] - rect30x5 [xxx*] (T1] - rect50x5 [90.00*] (T3] - rect76.2x5 [90.00*] (T10] - BAVURE [0.00*] (T10] - rect76.2x5 [0.00*] (T21) - rect76.2x5 [0.00*] (T10]	t
		•

Automatic punching/Maximum overflanking

Function use

Option to define a maximum overflanking value for lines.

Activating the function

- Adjust the settings in Tools > Options > Punch > Automatic allocation > Adjustments.
- Check the Maximum overflanking for lines box.

Application configuration	1 2 2 2 M 2				
Configuration	Adjustments				
User information	Ignore and induce segments of length lower than 0.50mm Circles of diameters (separed by ;)				
- 🧬 Shortcut key - 🙋 General	Notching No creation of notch but allocate each line				
Importation Punch	Irinit at box triangular notch using rectangular/square tool Tool name to use by default for notches decomposed into lines None Tool name to use by default for notches decomposed into lines				
Allocation	Lines Circles Circles Circles None All Collected None Maxi tool width(5.00mm Maxi tool dameter(5.00mm				
Tores order Tures mount Achining associated to the next Eut Cut Nesting Simulation	Reports reach Grids search Minimum number of holes for lines 3 for grids 4 for circles 3 Altern the start point of packel lines				
☐ ☐ Post processor	Other searches Macros search No search Special tools on external incs Special tools on external incs Search machinings of type line to line - Mini, number of lines to put in relation - Gap maxi, between 2 lines 20.00mm				
	Trapp door evacuation Wates Dimensions :: X min [100.00mm] Y min [100.00mm] Put Stop it tapp too small em/3610 Parts Dimensions :: X min [100.00mm] Y min [100.00mm] Put Stop it tapp too small				
	Search parameters Maximum diameter of a punched arc [50.00mm				
	Maximum diameter of a punched full circle 50.00mm More than this value, the full circle will be nibbled with Trapp door				
	Maximum overflanking for notching 6.00mm				

Mark open curves during punching

Function use

Option to use RollerBall tools on open curves during automatic punching. In previous versions, RollerBall was only effective for curves from text or lettering.

Activating the function

- Adjust the settings in Tools > Options > Punch > Automatic allocation > Adjustments.
- Check the Mark opened curves box.

onfiguration	Adjustments	
🖞 User information 🔶 Colors 🍰 Shortcut key 👰 General	Ignore and induce segments of length lower than (0.50mm Circles of diameters (separed by ;) Notching No creation of notch but allocate each line Richard Circles and	
Punch	Tool name to use by default for notches decomposed into lines None for trump/5000_sheet	•
Allocation Allocation Automatic allocation	Lines Allocete All All except on box Maxi. tool width 6.00mm Maxi. tool dameter 5.00mm	_
Tools order Turret mount Machining associated to the nest Cut Resting Resting	Peopols search Circles search Minimum number of holes for lines 3 for grids 4 for circles 3 Altern the start point of parallel lines Reconsistion of Tosofidi receivings	
Post processor Tasks manager Display options	Dither searches Macros search No search V 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 20.00	mm
	Trapp door evacuation Wastes Dimensions : X min 100.00mm Y min 100.00mm Put Stop if trapp too small trumpt5000_sheet Pats Dimensions : X min 100.00mm Y min 100.00mm Put Stop if trapp too small	1
	Search parameters Maximum diameter of a punched arc 50.00mm	
	Maximum diameter of a punched full circle 50.00mm More than this value, the full circle will be nibbled with Trapp doc	•
	Maximum overflanking for notching 6.00mm	•
	Machining an oblong with a circular tool and a rectangular tool for trump/5000_sheet	•
	Mark texts and lettering Mark opened curves for trump(5000_sheet	-

Example: Fold lines and open curves marked with the RollerBall.

Appli



TopSolid'Cut – Cutting

Cut parameter custom colors by machine

Function use

Option to define **by machine** a custom color for each cut parameter.

Activating the function

 Adjust the settings in Tools > Options > Cut > Cut parameters > Use colours defined in cutting parameters on elements.

ipplication configuration	
Configuration	Cut parameters
🔤 💆 User information	
🛓 😍 Colors	
🖋 Shortcut key	
🚽 🧓 General	
🗄 🗗 Importation	V Use colours defined in cutting parameters on elements (permanently)
🗄 🚭 Punch	
🖕 💑 Cut	
- 🐴 General	
- 🚰 Cut parameters	Allocation mode of machining parameters
Priming/Exit	Depends of dimensions of profile that supports the cutting element
	Depends of dimensions of the cutting element
🚽 👥 Cutting - Marking - Burning	
📅 Maakinia sekkalaan	

The color is permanently assigned to the cuts of the part.

Notes:

- Links are unaffected and their color remains navy blue.
- Lead ins are unaffected and their color remains yellow.
- Lead outs are unaffected and their color remains green.
- Additional cuts (loopings, etc.) are unaffected and their color remains green.

<u>Use</u>

When cutting parameters management 📴 is activated, the default colors appear on each parameter.

Cutting parameters ma	nagement							X
General Device name Cutting Gaz Nor	tt3030 n specified gaz	Matters A acier	vailable matters	•	Copy parameter: Copy (Copy	s Dne All	Thickness Available 2.00mm	thickness Add Remove Ask for confirmation
Cutting parameters Le	ad in/out stand	ard Lead in/out	common cutting					
								Extensive drillings manage
•							÷.	Extensive strong thickness primings manage
		TAB-TECNO	METHODE	TYPE-PENE	TYPE-COUPE	VITESSE	FENTE-DE-CI	Choose colour
		Text -	Number -	Number -	Number -	Value -	Value - mm	Copy line
Contour#1		ST020MD0-025	5	20	500	10	0	
Contour#2		ST020MD0-025	9	10	300	0.4	0.18	
Internal dian		ST020MD0-025	5	20	500	10	0	Add
<mark>Internal dian</mark>		ST020MD0-025	9	10	300	5.4	0.18	Contour 👻
Engraving#1		ST020MD0-025	5	20	500	10	0	Remove
Burning#1			0	0	0	0	0	Ask for confirmation
		Cancel			Apply		Sav	ve and Exit

Hover over the parameter name (for example: Contour#1). •

The **Choose color** button becomes active.

- Click on the Choose colour • button.
- Choose the color for the desired parameters.

Note: White, black and navy blue are not present in the palette.

When creating cutting paths on a new part, the colors are permanently associated.





The cutting parameters Information/modification button also makes information appear.

Machining parameters modi	ification						— X
Cutting Gaz Non specified gaz	2						-
			Choose in cutting pa	rameters tables			
Cutting parameters							
		METUODE			MILCOL		темро
	Tab-IELNU	METHODE	ITFE-FENE		Walua	Vehie	IEMFU Value
	lext -	Number -	Number -	Number -	value -	value - mm	value - s
Engraving#1	020MD0-02S0-30-2	5	20	500	10	0	0.1
		Cancel			OK		

The colors are also shown on the path instances on the nesting.

path instances on the ting.			
date old parts After having configured the co cutting parameters, open an old p without the new function. Activate the Update machining function to obtain the defined colo	olors of the art machined parameters ors.	Cutting Autre	 Win Cut PATH Set evacuatation with trapp door Modify machining parameters Cutting parameters management Update machining parameters Display used machining parameters Show/Hide path Extremities connectivity Path graphical informations

cutting parameters, open an old part n without the new function.

Update old parts

•

 Activate the Update machining part function to obtain the defined colors.

Cutting gas by material/by machine

Function use

Option to define the default cutting gas for a given machine (function prior to version v6.15) Additional option to define the cutting gas to suggest by default for a given material.

Activating the function

• Adjust the settings in Tools > Options > Cut > General > Default gas.

Application configuration	
Configuration	General
User information	Connectivity tolerancy for cutting elements and trajectories 0.00100mm
🖋 Shortcut key	Scale of visualization of priming/exit and insersion elements
Importation Punch	
General	Line width of cutting elements
	Show extremities of trajectories
Cutting - Marking - Burning	Security distance between link path and cutting path 3.00mm
Machining of the part	W Make safe links created manually
	V Check unsafe links during simulation
Gesting Simulation Set processor	Default gas Gas for this matter
Tasks manager Display options	Default gas oxygène

<u>Use</u>

By checking the box for the first time, all materials have the same gas by default. You just need to specify the gas using the list on the right.

When importing a part, and when the machine and the material are selected from the lists, the gas combo will position itself on the gas preset in **Tools** > **Options**.

General parameters	CARLES ST.	
	Thickness 200mm Ma Machine M 30	tter typ aterial 4L-50-3000-1500 4L-80-3000-1500 54H111-10-3000-1500 54H111-0-3000-1500 54H111-20-3000-1500 54H111-20-3000-1500
	Cutting Gaz azote Cutting parameters selection By contour Liberties Rotation forbidden Symetry forbid	dden
	Importation Uportation Texts Simplify all Texts Default wanted quantity Use part holes during complex nesting	Opened lines and arcs aiven Text colour
	Ok	

Laser path ergonomics

Function use

To make laser paths easier to read, it is possible to no longer show the extremities of paths (red squares to define the beginning and end of the path).



Activating the function

- Adjust the settings in **Tools** > **Options** > **Cut** > **General**.
- Check the box Show extremities of trajectories.

Application configuration	
Configuration	General
User information	Connectivity tolerancy for cutting elements and trajectories 0.00100mm
Shortcut key	Scale of visualization of priming/exit and insersion elements 1
Importation	
Cut General	Line width of cutting elements Thin Medium Thick Very thick Very very thick
Cut parameters	Show extremities of trajectories
Insertion elements	Unsafe links - contouring Security distance between link path and cutting path 3.00mm

<u>Use</u>

• Regenerate the view of the document sequences.



Automatically contour unsafe links

Function use

In the case of potentially unsafe links between cuts made "head down", it is possible to automatically contour cuts crossed by these unsafe links.

Activating the function

- Adjust the settings in **Tools** > **Options** > **Cut** > **General**.
- Check the boxes corresponding to the Unsafe links Contouring functions.

Application configuration	
Configuration	General
User information	Connectivity tolerancy for cutting elements and trajectories 0.00100mm
Shortcut key	Scale of visualization of priming/exit and insersion elements
Importation ⊕	
General	Line width of cutting elements
	Unsafe links - contouring
📲 💀 Cutting - Marking - Burning	Security distance between link path and cutting path 3.00mm
Machining of the part Machining associated to the nest	V Make safe links created manually
Micro joint	Check unsafe links during simulation
📗 🕀 🥰 Nesting	

Check function



After having launched the check and chosen to **hide all safe paths**, a question bar allows faster access to the various options.

Move head up for all unsafe links Insert waypoints Make safe links by contouring

To automatically contour unsafe links, there are several possible options:

GO BACK Make contouring on all unsafe link	s Security distance 5	Select unsafe link to countouring	
--	-----------------------	-----------------------------------	--

 Adjust the security distance and define the link to be contoured or choose to contour all unsafe links with the security distance entered.



Manual creation of links between cuts function

₹<mark>⇔</mark>

When links are created manually between cuts, option to automatically contour if the link that has just been created is potentially unsafe.

Change start point path	Head positions	Make safe by contouring= YES	Security distance	3	Select path to link	

Detection and correction before simulation of unsafe link paths in the Meg

When the **Check unsafe links during simulation** box is checked in **Tools** > **Options**, an alert may be reported if potentially unsafe links are found.

• Check the box in **Tools** > **Options** enabling automatic contouring.

<u>Note</u>: The automatic creation of cuts and links function does not check for unsafe paths. You must use the **Check** function.

g	ALERIS
S,	1 Alert(s) reported
fe	Des liaisons dangereuses ont été trouvées dans le document
ıg	
٢S	
u	
-	

TopSolid 7 and TopSolid'Sheetmetal link

Ignore the case in material names

Function use

TopSolid'Sheetmetal ignores the case for material names from 3D TopSolid.

If a 3D part is made of **Stainless steel** and the material entered in the sheet management of TopSolid'Sheetmetal is **stainless steel** or **STAINLESS STEEL**, there will be no conflict.

Ignore major revision indications

Function use

Option to ignore major revision indications when importing parts from TopSolid V7.

Retrieval of colors from the unfolding document in TopSolid'Sheetmetal

Function use

Option to find out the colors used in TopSolid 7 on the unfold in order to configure in **Tools** > **Options** the different colors relating to "cut" contours, "marked" contours and "burns", as well as the color of "free" points on which **automatic pointing** will be applied.

• To find out the colors of the V7 unfold, go to the PCH document properties.

Each line corresponds to the different profile/point categories encountered with the color from the palette and its number.



The PCH part display presents the profiles and points of the V7 unfold with their attributes. Also shown are the **points on curves** to which primings will be attached by default.

Note: The profiles are highlighted when a line in the box is clicked.