

Tutorial TopSolid'Cam 7



MASTER YOUR MANUFACTURING PROCESS

TopSolid'Cam 7 Tutorial

© 2020, TOPSOLID SAS. 7, Rue du Bois Sauvage F-91055 Évry, FRANCE Web: <u>www.topsolid.com</u> Email: <u>contact@topsolid.com</u> All rights reserved.

TopSolid[®] is a registered trademark of TOPSOLID SAS.

TopSolid[®] is a product name of TOPSOLID SAS.

The information and the software contained within this document are subject to change without prior warning and should not be construed as a commitment by TOPSOLID SAS.

The software covered by this document is supplied under license, and may only be used and duplicated in compliance with the terms of this license.

Hard copy or digital materials provided during training or accessible on-line as part of the training represent a protected original work that is the property of the training organization. They cannot be reproduced in part or in full without the express consent of the training organization.

All the texts, comments, works, illustrations and images reproduced on these materials are protected by copyright worldwide. Any use other than that designed for the purposes of training is subject to the prior authorization of the training organization, subject to prosecution. The Customer shall refrain from using, reproducing, representing, lending, exchanging, transmitting or transferring and more generally exploiting all or part of the data and/or transferring it to another material, and from modifying, adapting, arranging or transforming it without the prior written consent of the training organization. The Customer shall also refrain from extracting all or part of the exclusion of any transferring it to another material, and from modifying, adapting, arranging or transforming it without the prior written consent of the training organization. The Customer is only granted a right of use, to the exclusion of any transfer of ownership in any form whatsoever. Therefore, only the reproduction and representation of the content authorized by the French Intellectual Property code on a screen, and a single hard copy for archiving purposes, are authorized, for strictly personal purposes, and for professional use.

The Customer also undertakes to not take part in competing, directly or indirectly, with the training organization by transferring or communicating these documents to anyone.

Version 7.14 Rev.01

<u>Note</u>: If you are experiencing any problems with this tutorial, please feel free to send your feedback and comments at <u>edition@topsolid.com</u>.

Contents

Creating a Machining Part	1
Introduction	1
Working with the PDM	1
Step 1 – Quick machining	4
Step 2 – Inserting a part in a blank machining	6
Step 3 – Quick machining with setup	10
Step 4 – Machining with a selected machine and inserting a part or setup	13
Step 5 – Assembled part and stock, setup, then machining	15
Step 6 – Modifying a stock	
2D Milling	20
Introduction	20
Machining	21
Options	21
Step 1 – Machining the upper section of the part	22
Step 2 – Machining the left section of the part	27
Step 3 – Machining the right section of the part	
Step 4 – Machining the inclined planes	32
Step 5 – Machining the center groove	35
Step 6 – Machining the outer section	
Step 7 – Machining the three steps	
Step 8 – Performing the drillings	41
Step 9 – Performing the borings	44
Step 10 – Alternative machinings	46
Conclusion	49
3D Milling	50
Introduction	50
Machining	51
Step 1 – Roughing	52
Step 2 – Re-roughing	57
Step 3 – Finishing	60
Step 4 – Re-finishing	66
Turning	69
- Introduction	69
Step 1 – Facing the end of the part	71
Step 2 – Turning	74

Step 3 – Groove roughing	76
Step 4 – Groove machining	79
Step 5 – Finishing	81
Step 6 – Threading	83
Step 7 – Hole machining	84

Creating a Machining Part

Introduction

The purpose of this chapter is to create a machining document which can be used to perform various machining operations that can be accessed from the Cam section of **TopSolid**.

We will explain in detail five different methods that will produce a similar result.

We will start with the easiest method (**TopSolid** will automatically respond to some questions for the users), then we will discuss the most detailed method (where we must answer all of the questions).

Working with the PDM

As explained in the online help and the Design support, the PDM allows you to store and monitor changes made to any files created in **TopSolid 7**.

In addition, in order to isolate all the tutorial files, we will create a new project containing all the required files.

To create this new project, proceed as follows:

- Click on the 💙 icon of the **TopSolid 7** menu.
- Select **File** from the drop-down menu.
- Select the 🐸 New Project command.

You can also click on the icon from the **Home** tab or use the **Ctrl** + **Shift** + **N** keyboard shortcut to create a new project.



The following dialog box is displayed. Rename the project *TOPSOLID*, select **Blank Template** and confirm with the green checkmark ✓.

🕙 New Project	—	×
Name:		
Template:		
in Modèles outillage		
Blank Template		
Vault:		
Coffre		~
∀ × ?		

• From the Project tree, right-click on the project name and select the **Folder** command from the contextual menu.

TOPSOLID	
🔣 📴 🗊 🖗 📚	🚬 🕰 - 🧮
	Part Assembly Machining Bill of Material
1 1 1 1 1	Open Document Consult Document Folder

By default, the created folder will be named *Folder*.

• Rename the folder *Support*.

Note: To rename a file, a document or a folder, you simply have to press the F2 key or slowly double-click on the left mouse button (such as for Microsoft Windows Explorer). Examples of folder names are provided in order to easily locate them when creating the machining.

When changing the name of a file, it will be preceded by the \mathbb{P} icon and followed by an asterisk * indicating that the document has been changed or created. Saving the document will remove the asterisk.

TOPSOLID	Ψ×
🔣 🚼 🗊 📭 🃚 🐂 🛆 - 🚍	?
🕂 🐋 TOPSOLID	
🛓 📲 References	
🗄 📲 🕎 Templates	
🗄 🕂 🖶 🕁 Favorites	
🔤 🕂 🖶 🍧 Recycle bin	
🛶 🖶 🥁 Support	

In the Support folder, you will now be able to create and machine a new part.

Given that the purpose of this chapter is not to geometrically define the part (see *TopSolid'Design Tutorial*), we will import a part. In this case, since this part was not designed in **TopSolid**, we will lose all the benefits of the integrated solution (interactive modification, use of machining information such as H7 hole, etc.).

• To import the part, right-click on the *Support* folder and select the **Import/Export** > Import File with Conversion command.

TOPSOLID				
🔣 🔛 🗊 😼 🍣 🕇	N 🕰 - 🚍			
🕂 🐋 Topsolid				
🛓 📲 References				
🎚 🖷 🕂 🗽 👔 Templates				
🗄 🖷 🖶 🌄 Favorites				
Suppor	Part			
E	Assembly			
	Machining			
58	Cut			
	Сору			
			L	
	Import / Export	- F	1	Export Package
7.3	Apply Automatic Part Numbering			Import Package
	Others		2	Import File with Conversion
	oties		1	Import File without Conversion

- Open the *TopSolidCam_SupportOnlyFinishPart.x_t* file (during installation, this file is installed in the "Programs Files\TOPSOLID\TopSolid 7.14\Samples" folder).
- In the dialog box that appears, select the following parameters.

Parasolid Import	-		×
Translator name: Parasolid			1
File name: TopSolidCam_S	SupportOnlyl	FinishPart.)	c_t
General Simplification and sewing T	emplates		
Document type for shapes: Part			~
Document type for assemblie: 🦻 Part			~
Translate attributes			
✓ × ?			

- Click on the 💙 icon to **confirm** the import.
- Using the F2 key, rename the imported file Support (part only).

<u>Note</u>: Right-clicking on a folder allows you to create a new entity in it, import one or more documents into it, or export, edit, or copy it.

Step 1 – Quick machining

In this section, we will explain how to quickly start machining the part that we just imported.



From the Project tree, right-click on the part document and select a *Promotion* Machining document.



The following dialog box appears.



You can select a predefined template. In our case, we will select **Blank Template** and click on the \checkmark icon to **confirm**.

The following dialog box appears, prompting you to choose some settings related to the machinable part.

- Choose a prismatic-shaped stock by selecting **Block**.
- Check the Eliminate the weak portions of stock box.

stock Offset	
stock Offset	
ock	
t	tock

- Click on 🗡 to **confirm**.
- Click on the Positioning 1 button to exit the positioning context.

The machining document is automatically created and placed in the same directory as its related part.

• Save the document by clicking on the \blacksquare icon or by using the Ctrl + S keyboard shortcut.

For clarity purposes and to help you become familiar with the Cam section of **TopSolid**, we will create a new folder. By doing this for each of the five methods that we are explaining, we will be able to come back more quickly to any of these methods later, if necessary.

• Right-click on the *Support* folder and select the **Folder** command. Rename the folder *Step 1*.

We will then move the machining document into the *Step 1* folder and rename it.

- Use Hold down the left mouse button and drag and drop the machining document into the Step 1 folder.
- Rename the machining document Step 1 Support.



• **b** Save and close all open documents.

<u>Note</u>: You can see that the part is enclosed in a prismatic stock and its margins have been defined by default. We will see later in the *Step 6 – Modifying a stock* section how to change this stock.

You can now start machining the part.

Step 2 – Inserting a part in a blank machining



This step produces the same result as Step 1, but introduces another feature of the software. In fact, we will create a blank machining document and then insert the machinable part.

Using this document would make it possible, for example, to use a template directly afterwards with a machine and fixed tools.

- In the *Support* folder, create a new **b** folder and rename it *Step 2*.
- As before, right-click on this folder and select the *Machining* command.
- Select a **blank template** and click on \checkmark to **confirm**.
- Rename the machining document Step 2 Support.



A machining document has been created. This document is blank for now. There are no machinable parts. We will now indicate to **TopSolid** that we want to machine the imported part.

To do this, we will place the part document in the machining document using the drag and drop technique. This technique is easily done by simply holding down the left mouse button while dragging the *Support (part only)* part document into the open machining document.



The following dialog box appears, prompting you to choose some settings related to the machinable part.

- Select the **NC part creation** option.
- Select a **block** prismatic-shaped stock.
- Check the Eliminate the weak portions of stock box.

Creation of M	achinable Part				×
NC part cre	Inclusion				
- Stock —					
		X			
Block	Cylinder	No stock	Offset		
Exact	MachineW				
✓ Eliminate th	e weak portion	s of stock			
Check the c	eometry of sto	ek			
	jeometry of sto	CK.			
		🖌 🖌	?		
		• • •	•		

- Click on V to confirm.
- Click on the Positioning 2

button to exit the positioning context.

The part is ready to be machined but it is positioned where you released the mouse button. This position might not be the one you require.

Example:



If you want to modify the position of the part, right-click on the part and select the 🍣 Edit NC Part Positioning command.

	N
	2D
	End Milling
<u></u>	Side Milling
	Drilling •
4	Roughing
	Vertical Roughing
	Others •
	Support 1 <566>
	Show Machined Part Setup Document
4	Edit NC Part Positioning

In the positioning context, select the **Frame on Frame** command from the **Assembly** tab. In the **Source** • frame field, select the Support (part only) frame from the drop-down list. In the Destination frame field, select Absolute Frame.



Positioning 2 button to confirm the positioning. Click on the

The part is now positioned in the center of the table.



• **Save** and close all open documents. You can now start machining the part.

Step 3 – Quick machining with setup



In this section, we will manually set up the machining, which was automatically performed in the previous steps.

There are two possible ways to create this machining setup, as seen in Step 1 (creating a setup document from an existing file) and Step 2 (creating an empty setup document and inserting the machinable part). Here, we will explain how to achieve the machining by following the method in Step 2.

- In the *Support* folder, create a new **folder** and rename it *Step 3*.
- Right-click on this folder and select the Document command.



• From the Advanced tab, select 🥶 Machined Part Setup, select Blank Template, then click on 🛩 to confirm.

🐑 New Document	– 🗆 X
New Document Type: Common Advanced Special Machined Part Setup Milling/Turning Method Drafting Bundle Workspaces Manager Drawing Machine Machine	Template: Project Templates My Templates Modèles standards Usinage - France Modèles standards - France Modèles standards Métal - France Modèles Standards Outillage - France Modèles Standards Usinage Fil - France Modèles Standard Templates - United States Standard Templates - United States Steel Standard Templates - United States
Project: TOPSOLID ~	Tooling Standard Templates - United States Wire Machining Standard Templates - United States Blank Template X 2

• Rename the machining setup document *Support 3 setup*.



A blank machining setup document has been created. To insert the part to set up, we will use the drag-and-drop technique discussed in Step 2.

Once the part has been inserted into the machining setup document, you can see that it does not include a stock.

- To edit the machining setup, right-click in the graphics area.
- Select the 🊧 Edit Part NC command.



You must now define the part and its stock.

• Click on the 🐸 Finishes, stocks and environment icon.

<u>Note</u>: **TopSolid** has automatically selected the finish and defined an enclosing stock that can be modified. You can define unique margins (identical margins in all directions) or customize the margins. Modifications can be assessed in real time in the 3D view.

We will choose 5mm margins in all the absolute frame's directions, except for the Z- margin that will be 60mm.

竊 Finishes, stocks and environment	X	Start Page	Support 3 setup*
🗳 Finishes	* ^	🗸 🖌	?
Finish:			
Support (part only) <144>			
			† 5 † 5
@ 54-44)			
Stock(s)	*		<u>† 5</u>
Stock Type Block			
			T 60
Dimensions	*		
Select a frame:	_		
Absolute Frame V	÷	ľ	
Single margin			
Size in X: Size in Y: Size in Z:			
👔 159,625171m 📳 110mm 📳 145mm			
Margin in X-: Margin in X+:			
🔮 5mm 🗳 5mm			
Margin in Y-: Margin in Y+:			
✓ 5mm			
Margin in Z-: Margin in Z+:			
60mm 5mm			
Reset margins Identical margins			

• Click on ؇ to confirm.

The part and its adjusted stock are now defined.

Your screen should now look like the following image:



Save the document.

There are two possible ways to create this machining, as seen in Step 1 (creating a machining document from the existing setup) and Step 2 (creating an empty machining document and inserting the existing setup).

• 😼 Save and close all open documents.

You can now start machining the part.

Step 4 – Machining with a selected machine and inserting a part or setup



This method is similar to the method in Step 2, with the exception that it includes a choice of machine. We will therefore only discuss how to include a NC machine.

- In the *Support* folder, create a new **folder** and rename it *Step 4*.
- Create a new 🚰 Machining document using a blank template, then click on 💙 to confirm.
- Rename this document *Step 4 Support*.

We now have a blank machining document in which we will insert a part.

- From the **Equipment** tab, click on the $\overline{\nabla}$ button to display the drop-down menu.
- From the Machine Tool menu, select the 🐝 Select a Machine command.



To find the machine more easily, you can filter the list of machines.

• In the Name or manufacturer field, enter mi.

A machine will appear only if the name or the manufacturer contains the text you entered. The search is not casesensitive and you may type in upper or lower case.

• Select the MIKRON HSM 600U 5-axis machine, then click on 💙 to confirm.

Select a Machine									×
	Filters Name or manufacturer: mi Machines source: All Machining Types: Milling Drilling Turning								~
	Name	Manufacturer	Drilling	Milling	Turning	Project	Compatibility	Comm	er ^
	D560	ROMI		\checkmark		2.Máquinas Globo	Compatible	Milling.	
	DISCOVERY 560	ROMI	\checkmark	\checkmark		TopSolid NC Machines	Compatible	Milling.	
The machine definition is	HSM 600U	MIKRON				TopSolid NC Machines	Compatible	Milling.	
valid.	NC Machine Milling (Head XYZ / Table A)	Missler Software	M	М	Π	TopSolid Machining	Compatible	Milling.	> ×
		🗸 🗶							

Once the machine is selected, you can insert a part or machining setup. To do this, refer to the *Step 2 – Inserting a part in a blank machining* section.

• Position the part so that it is located in the middle of the machine table as shown below.



• **I** Save and close all open documents.

You can now start machining the part.

Step 5 – Assembled part and stock, setup, then machining

In this section, we will use a stock and a part that have already been assembled. This method is particularly useful if the stock's geometry is variable or if the stock has already been machined. However, you have to be familiar with the **TopSolid'Design** assembly tool in order to use it.

We will first import the assembly in Parasolid format.

- In the *Support* folder, create a new **folder** and rename it *Step 5*.
- Right-click on this folder and select the Import/Export > 2 Import File with Conversion command.
- Open the *TopSolidCam_Support.x_t* file (during installation, this file is installed in the "Programs Files\TOPSOLID\TopSolid 7.14\Samples" folder).
- In the import dialog box, select the following parameters.

Parasolid Impo	ort			Х			
Translator na	me: Parasolid						
File name:	TopSolidCam_Support	.x_t					
General	Simplification and sewing Template	25					
Document	type for shapes: 🛛 🗗 Part			~			
Document type for assemblie: Assembly							
Translate attributes							
	∀ × ?						

• Click on 💙 to **confirm** the import.

• Rename the assembly document *Support*.

<u>Note</u>: When importing an assembly document, **TopSolid** imports the assembly and each component of the assembly separately. Consequently, you can use only the finish or only the stock for various modifications, for example.

- Right-click on the stock in the graphics area and select the 🤔 Attributes command.
- Adjust the transparency to 70% and click on V to confirm.
- **J** Save the assembly document.
- From the Project tree, right-click on the *Support* assembly document.
- Create a new Machined Part Setup document using a blank template. The following message appears asking you whether you want to disassemble the assembly.

🥜 Dis	sassembly	×
?	Do you want to disassemble the included assembly?	
	Yes No	:

- Click on **Yes**. In this way, each solid of the assembly will be displayed independently in the **Finish** list.
- Click on the Binishes, stocks and environment icon.
- In the dialog box that appears, click on **Stock Type** (the blue text) and select **User stock**.

鷆 Finishes, s	tocks and e	nvironment		_	_	
🥌 Finishes –						\$
Finish:						
Support 1 <1	57>					
Support 0 <1	62>					
🔍 Stock(s) –						-
Stock Type						
					~	
					FIT)	
			\sim		~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~	
Block	Cylinder	User stock	No stock	Inherited st	Offset	
Stocks -						
						^
Stock:						

• The **Finish** list contains the two solids, with **Support 1** being the part to be machined and **Support 0** being the stock. You therefore need to move **Support 0** to the **Stock** list. To do this, you simply have to move the mouse over **Support 0**, hold down the left mouse button and drag **Support 0** to the **Stock** list.

Your screen should look like the following image:

Finishes, stocks and environment	×
🗳 Finishes	* ^
Finish:	
Support 1 <157>	
- Stock(c)	
Stock(s)	*
Stock Type	_
	🏂 🔊 💧
Block Cylinder User stock No stock Inf	erited st Offset
- 👼 Stocks	*
Stock:	
Support 0 <162>	

• Click on 💙 to confirm.

<u>Note</u>: You can see that the selection areas are framed in a certain color and when a shape is selected for example, this shape will take on the color of the frame.

The machining setup document is complete. You can save and close it, and then start machining.

There are two possible ways to create this machining, as seen in Step 1 (creating a machining document from the existing setup) and Step 2 (creating an empty machining document and inserting the existing setup).

<u>Note</u>: We will use this machining setup in the machining tutorial.

You can now start machining the part.

Step 6 – Modifying a stock



When you scroll down the entire *TOPSOLID* project tree, you can see that machining setup documents have been automatically created for all the machining documents. This is so you can go back and modify the stock or the NC part origin, for example.

There are two ways to access the machining setup:

• Scroll down the tree structure and double-click on the automatically created machining setup documents.



• Or select a face of the part, then right-click in the graphics area and select the show Machined Part Setup Document command.



Once you have accessed the machining setup document, if you want to modify it, right-click in the graphics area and select the *between the setup to the setup to*



You now have access to the machining setup menus shown in Step 3.

2D Milling

Introduction

The purpose of this chapter is to create a machining program for the following part.



As you can see, to be machined, the part must be positioned on different orientations. To avoid several disassemblies, which can be detrimental to the desired manufacturing quality, we will use a 5-axis machine.

To help you become familiar with **TopSolid 7**, the machining of the part above is divided into several steps. For each step, a machining document is available in which operations have been completed. You can refer to these documents if you run into any difficulties. These machining documents are provided in a TopPkg file named *TopSolidCam_2DPartStepByStep.TopPkg* (this file is installed in the "Programs Files\TOPSOLID\TopSolid 7.14\Samples" folder). In order to follow the tutorial, we recommend that you import this file into a new project.

Example: For Step 6, which consists of performing the external machining of the part, a machining document named *Support_Step_5* is available, in which steps 1 to 5 are already carried out.

The part was designed in **TopSolid**. The designer used the "features" technology, which allows major automation of the machinings.

The overall programming approach is as follows:

- 1. Defining the stock part
- 2. Positioning and defining the part (as per TopSolid'Cam) and its program origin
- 3. Defining the operations with integrated stock management
- 4. Simulation
- 5. Editing the ISO file

Steps 1 and 2 are explained in the prerequisite tutorial (creating a machining setup document and machining document). We will start at Step 3, which consists of defining the machining operations.

As you have seen in this tutorial, several methods are available for creating a machining document. You can therefore start the machining by selecting the method of your choice. For this chapter, the selected method is method 5, which consists of machining an assembled stock and finish.

Machining

We are now in the machining context. **TopSolid 7** is simple to use and suggests two approaches for creating a machining.

• Subject – Verb:

Select the entities to be machined, and then select a machining command.

With this solution, you can select one or more faces with the mouse and indicate the machining on which to apply the faces.

• Verb – Subject:

Select a machining command and indicate the entities on which to apply this command.

These two methods will be used in this chapter.

Options

In general, you can adjust the machining options via the **Tools** > \square **Options** > \square **Machining** menu.

For example, in the **Tools** > ^{IIII} **Options** > ^{III} **CAM Options** > ^{III} **Simulation** command, you can choose between different modes of simulation once the machining operation has been confirmed.

🚻 Options		— 🗆 X
	~	Creation behavior
😽 Buildings		After operation creation:
CAM Options		O Do not simulate
🕀 CoroPlus ToolLibrary		Simulate with programmation mode
		 Simulate with current simulation mode
🔁 Hole analysis		Display
Machining Cloud		
¶ Tools		 Display the last few segments
		 Display full trajectory from beginning to current point
		 Always display the full trajectory
Machines		Enable jump on the trajectory
Method		Pause simulation before and after every rapid movement
NC Operations Manager		
Crigins		
Part settings		
Post-processors		
Side Milling		
WCS Ontions		
WIRE Options		
Drafting		
E Family		
Mold		
	~	Reset
		✓ × ?

Step 1 – Machining the upper section of the part



- Create a **New project** named 2D Machining by using a **blank template**.
- **** Import** the *TopSolidCam_2DPartStepByStep.TopPkg* package in your project.
- Open the *Support_Step_1* machining document.

We will first machine the upper face.

• Right-click on one of the two upper planar faces and select the 🕒 End Milling command.



A label such as the one below is displayed. Some values such as the maximal axial depth, which is equal to zero

because the tool has not been selected yet, may be different. They can be modified in the Settings option.

#1	
Time	00:00:00:000
Altitude	100mm
Stock to leave on floor	0,2mm
Stock to leave on wall	0,2mm
Stock to leave on wall island	0,3mm
Maximal axial depth	0mm
Final axial depth pass	0mm
Lead in point	
Tool Path Preview	Yes

Note: The **v** icon indicates that a facing operation will be performed. If you want to change the icon, you only have to double-click on it and select the appropriate operation (pocketing, open pocketing or expanded facing).

- Click on the **Click on the Content** icon. The blue asterisk indicates a required field.
- Double-click on the Face Mill Face Mill thumbnail and create a 63mm diameter face mill with a 10mm cutting height.



• Click on 💙 to **confirm** the tool.

The facing settings remain to be indicated.

- To do this, click on the Settings icon.
- Adjust the following settings:
 - Stock to leave on floor and Stock to leave on wall: Omm
 - Maximal axial depth: 5mm
 - Final axial depth pass: Omm
 - End milling strategy: Select Sweeping
 - Sweeping strategy: Select Zigzag cycle.
 - Check the **Minimum path number** box
 - Sweeping over length: 2mm

You should end up with the following result.

End Milling : Settings	×						
💪 Settings 🛍 Altitudes 🗼 Plu	inge 💐 Contouring integrated						
Islands facing High Sp	eed Machining Volumill						
Stock : 2mm Machined Stock + 0mm Stock Le	ft = 2mm						
Passes : 1 x 2mm = 2mm							
Take into account the stock shape							
🖆 Overlap	*						
Step over	External clearance distance						
25,2mm 40%	0,5mm						
Stocks to leave and steps	*						
Stock to leave on floor	Maximal axial depth						
0mm	5mm						
Stock to leave on wall	Final axial depth pass						
0mm	0mm						
Stock to leave on wall island	Final axial feed rate Machining						
0,3mm	= 1000mm/min						
Stock to leave on wall shift							
0mm							
Organization of strategies	*						
Order of the path	Crder by pockets						
Strategy	*						
Milling direction	Climb						
Z path stock fitting strategy	None None						
End milling strategy	Sweeping						
Clearance off stock							
Roll-in Approach Strict mode sweeping	X None						
Sweeping strategy	Zigzag cycle						
Minimum path number							
Sweeping contouring strategy	🧇 After sweeping						
Sweeping over length							
2mm							
Sweeping angle mode	V Automatic						
Sweeping end path strategy	Toffset						
Link strategy for free place	Work Arc						
Minimal length of the holes							
2mm							
Reposition clearance							
0,4mm							

Note: These settings are for guidance only and in no way resulting from a particular methods department. They are simply for verifying that the settings coincide with the obtained tool paths.

With these settings and this tool, the tool path preview should be as follows:



<u>Note</u>: **TopSolid** is configured to avoid unnecessary calculation to improve the calculation time. When modifying a machining setting (for example a stock to leave), the machining area and the tool path preview are not graphically

updated (it may take time). An icon 🕶 at the top right of the screen indicates whether or not the graphics area is updated. This icon has three states:

- 💞: The graphical update is manual and the graphics area is currently updated.
- **W**: The graphical update is manual and the graphics area is currently out of date. Clicking on this icon will update the graphics area (it may take time).
- 🗲: The graphical update is automatic. Each time a setting is modified, the machined area and the tool path are recalculated (it may take time).
- Click on 💙 to **confirm** the machining operation.

Simulating the machining

In this section, we will see how to resimulate the previously performed machining.

First, a very useful option allows you to remove material during machining. Then, to simulate the machining of one operation, you only have to click on the icon that indicates a facing operation, for example.



To simulate one or more operations, you can right-click on the selected operations and select the **Finulate** command.

To simulate all operations, right-click on 🥮 Part 1 and select the **Fimulate** command.

• 😼 Save the document.

Step 2 – Machining the left section of the part

• Open the *Support_Step_2* machining document.

<u>Note</u>: For this step, we will machine the outer section of the part using a radiused mill. The operation will be an open pocketing operation. The technique is the same as in the previous step.



- As before, right-click on the face to be machined shown in blue above.
- Select the Content of Content o

The following label is displayed.

#1	
Time	00:36:36:143
Altitude	36mm
Stock to leave on floor	0mm
Stock to leave on wall	0,3mm
Stock to leave on wall island	0,3mm
Maximal axial depth	0,9mm
Final axial depth pass	0,3mm
Lead in point	
Tool Path Preview	Yes

The < icon indicates that an open pocketing operation will be performed.



In the **Tool choice** option, double-click on the Radiused M... Radiused Mill thumbnail and select a 32mm diameter radiused mill, with a radius of 3mm and a total length of 150mm. From the Tool Assembly tab, adjust the output distance to 120mm.



- Click on 💙 to **confirm** the tool.
- Click on the Settings icon and adjust the following settings:
 - Stock to leave on floor: Omm
 - Stock to leave on wall: Omm
 - Maximal axial depth: 15mm
 - Final axial depth pass: Omm
 - End milling strategy: Select Sweeping
 - Sweeping strategy: Select Zigzag cycle raw
 - Sweeping contouring strategy: Select After sweeping
 - Sweeping over length: 5mm
 - Check the Minimum path number box

	Settings	<u>í</u>	Altitudes	*	Plunge		Co	ntouring integrated	
	Island	ds facir	ng	Hi	gh Speed Ma	chining		Volumill	
Sto	ck : 64mm M	lachine	d Stock +	0mm St	tock Left = 64	4mm			
Pas	ses : 4 x 12,8	mm +	1 x 12,8m	ım = 64ı	mm				
ź	Machining	profile	s options						;
\checkmark	Take into acc	ount t	he stock s	hape	🔽 Tal	ke into ac	count	the finish shape	
					🗌 Igr	nore botto	om rad	ius	
ź	Overlap								
Ste	ep over				Exterr	nal clearar	nce dis	stance	
10,	,4mm		40%		0,5mi	m			
la	Stocks to le	ave ar	nd steps						
		n floor			Mauir	nal avial e	lanth		;
0~	DCK to leave o	n nooi			15mm		iepin		
011 C1-					Einel	n in dent	h		
0~	DCK to leave o	n wali			Final o	axiai dept	n pass	•	
011 C1-			:					2	
0.2		n wan	Island		Final	axial feed	rate	Machining	
0,0			-1.164					= 1777mm/min	
0	ock to leave o	n wali	shirt						
UII	1111								
	Organizatio	n of st	rategies						;
Or	dar of the pai								
	der of the pa	th			0 📑	rder by p	ocket	s	
÷	Strategy	th			0 📑	rder by p	ocket	5	:
⊷ Mi	Strategy	n			<u></u> ∎ 0 	rder by p limb	ocket	ls	:
→¶ Mi Z p	Strategy — Iling direction	n ting str	ategy		== 0 == == == == ■ = = = = = = = = = = = = =	irder by p limb	ocket	ls	:
Mi Z p Ene	Strategy — Iling direction path stock fitt d milling stra	th n ting str tegy	ategy		0 <u>=</u> 0 	rder by p limb one weeping	ocket	IS	
→¶ Mil Z p Enc	Strategy — Iling direction bath stock fitt d milling stra Clearance off	th n ting str tegy f stock	ategy		0	rder by p limb lone weeping	ocket	15	
Mil Z p End Rol	Strategy Iling direction bath stock fitt d milling stra Clearance off Il-in Approac	th n ting str tegy f stock	ategy		€ 0	rder by p limb lone weeping lone	ocket	IS	:
Mii Z p End Ro	Strategy — Iling direction bath stock fitt d milling stra Clearance off Il-in Approac Strict mode s	n ting str tegy f stock :h sweepii	ategy		■ 0 ■ 0	rder by p limb lone weeping lone	ocket	5	:
Mi Z p End Ro Sw	Strategy Iling direction bath stock fitt d milling strate Clearance off Il-in Approac Strict mode s reeping strate	th ting str tegy f stock th sweepin	ategy		€ 0	rder by p limb one weeping lone	le raw	'S 	:
Mil Z p End Rol Sw	Strategy Iling direction bath stock fitt d milling stra Clearance off Il-in Approac Strict mode s reeping strate Minimum pa	th ting str tegy f stock th sweepin gy th nun	ategy ng		■ 0 ■ 1	rder by p limb lone weeping lone igzag cyc	ocket	is 	:
Mii Z p End Rol Sw Sw	Strategy Iling direction bath stock fitt d milling strat Clearance off Il-in Approac Strict mode s reeping strate Minimum pa reeping conto	th ting str tegy f stock th sweepin sy th nun puring	ategy ng nber strategy		■ 0 ■ 0	rder by p limb lone weeping lone igzag cyc	le raw	'S	-
Mil Z p End Rol Sw Sw Sw	Strategy Iling direction bath stock fitt d milling strat Clearance off Il-in Approac Strict mode s reeping strate Minimum pa reeping conto reeping over	th ting str tegy f stock sweepin gy th nun puring lenath	ategy ng nber strategy		■ 0 ■ 0 ■ 0 ● N	rder by p limb lone weeping lone igzag cyci	eping	is 	
Mil Z p End Rol Sw Sw Sw Sw Sw	Strategy Iling direction bath stock fitt d milling strate Clearance off Il-in Approac Strict mode s reeping strate Minimum pa reeping conto reeping over l 1000	th ting str tegy f stock th sweepin sweepin sy th nun puring length	ategy ng nber strategy		 ■■ 0 ● ● N ● S N ■ S N ■ Z ■ 	rder by p limb lone weeping lone igzag cyc fter swee	ocket	rs	
Mil Z p Enc Rol Sw Sw Sw Sw Sw	Strategy Iling direction bath stock fitt d milling strat Clearance off Il-in Approac Strict mode s eeping strate Minimum pa eeping conto eeping over l m eeping apple	th ting str tegy f stock th sweepin th nun puring length	ategy ng nber strategy		■ 0 ■ 0 ■ 0 ● N	rder by p limb lone weeping lone igzag cyci fter swee	le raw	is ,	-
Mil Z p End Rol Sw Sw Sw Sw Sw Sw	Strategy Iling direction bath stock fitt d milling strate Clearance off Il-in Approac Strict mode s reeping strate Minimum pa reeping conto reeping over l m reeping angle	th ting str tegy f stock th sweepin sweepin th nun puring length e mode	ategy ng nber strategy		■ 0 ■ 1	rder by p limb lone weeping lone igzag cyc fter swee utomatic	eping	r 	-
Mii Z p End Ro Sw Sw Sw Sw Sw Sw Sw Sw	Strategy Iling direction bath stock fitt d milling stra Clearance off Il-in Approac Strict mode s eeping strate Minimum pa eeping conto reeping over l m eeping angle eeping angle	th ting str tegy f stock th sweepin gy th nun puring length e mode path str	ategy ng nber strategy ategy		■ 0 ■ 0 ■ 0 ● N	rder by p limb lone weeping lone igzag cyci fter swee utomatic ffset	le raw	is	
Mil Z p End Rol Sw Sw Sw Sw Sw Sw Sw Lin	Strategy Iling direction bath stock fitt d milling strate Clearance off Il-in Approac Strict mode s reeping strate Minimum pa reeping over l m reeping angle reeping angle reeping end p	th ting str tegy f stock th sweepin th nun buring length a mode bath str r free p	ategy ng nber strategy ategy		■ 0 ■ 1	rder by p limb lone weeping lone igzag cyc fter swee utomatic ffset /ork Arc	eping	rs	-
Mil Z p Enc Rol Sw Sw Sw Sw Sw Sw Sw Sw Lin Mi	Strategy Iling direction path stock fitt d milling stra Clearance off Il-in Approac Strict mode s eeping strate Minimum pa reeping conto reeping over l m reeping angle reeping angle reeping end p sk strategy fo nimal length	th ting str tegy f stock th sweepin gy th nun ouring length a mode path str r free p of the	ategy ng nber strategy ategy alace holes		■ 0 ■ 1	rder by p limb lone weeping lone igzag cyc fter swee utomatic ffset /ork Arc	le raw	is 	-
Mil Z p End Rol Sw Sw Sw Sw Sw Sw Sw Sw Lin Mil 2m	Strategy Iling direction bath stock fitt d milling strate Clearance off Il-in Approac Strict mode s reeping strate Minimum pa reeping over l m reeping angle reeping angle reeping end p sk strategy fo nimal length m	th ting str tegy f stock th sweepin egy th nun buring length e mode bath str r free p of the	ategy ng nber strategy ategy alace holes		 □ □	rder by p limb lone weeping lone igzag cyc fter swee utomatic ffset /ork Arc	le raw	IS	

- Click on **V** to **confirm** the operation.
- **Save** the document.

Step 3 – Machining the right section of the part



• Open the *Support_Step_3* machining document.

As you can see, the left and right sections clearly match. To repeat this machining, we will add a geometry to the previous operation.

Note: We could have selected the two surfaces to be machined beforehand and perform the open pocketing operation.

- From the NC Operations tree, **Constant** the previously performed end milling operation.
- To add a geometry to an operation, click on the 🥏 Geometry icon.
- Add the right-hand surface by clicking on it.

The machining operation is repeated while keeping the same settings. This action also allows you to remove a geometry from an operation.

占 End Milling : Geometry		X Start Page F Support_Step_3*
~		A X X 2: Radiused Mill Extended Length D32 R3 L40 SD32
Part	Part 1	
Local stock		
·		
Geometry	(501)	
	(846)	
○ Optimisation		
	This is a second s	
Optimization method	None	
Optimization linear tolerance	Optimization angular tolerance	

Click on V to confirm the operation.

Note: So far, the tool path has been ordered by pockets. We can see here the advantage of such a method for this part with regard to the time saved. The machine will first machine all the left-hand layers and then the right-hand ones.

A strategy by altitudes could have been used if the part had thin walls.

In the end, the machining preview should be as follows:



Note: Instead of adding the face in the first operation, you can also request to display the tool paths \bigcirc and then drag and drop this machining onto the new face. You only have to hold down the **Ctrl** key while you perform the drag-and-drop operation.

• 😼 Save the document.

Step 4 – Machining the inclined planes



Machining the small inclined planes



- Open the *Support_Step_4* machining document.
- Right-click on one of the blue faces shown above and select the 😕 End Milling 2D machining command.
- In the W Tool choice option, check the face mill that was used for the first machining (63mm diameter).
 - Click on the 🠸 Settings icon and adjust the following settings:
 - Stock to leave on floor and Stock to leave on wall: 0mm
 - Maximal axial depth: 5mm
 - Final axial depth pass: Omm
 - Z path stock fitting strategy: Select Fit the stock
 - End milling strategy: Select Sweeping and the Zigzag cycle raw strategy
 - Check the **Minimum path number** box
 - Sweeping over length: 10mm
 - Sweeping angle mode: Select Value and Sweeping angle: 0°
| | Settings | دو | ltitudes | ł | Plunge | | Contouring integ | rated |
|-------|------------------|--------------|--------------|---------|-----------------|----------|---------------------|-------|
| | Islands | facing | Hiak | Sneer | - Machining | | Volumill | rateu |
| | k · 17 3925m | in Machin | ed Stock + 0 |)mm St | rock Left – 17 | 3925 | mm | |
| ass | es : 3 x 4.348 | 1mm + 1; | x 4.3481mm | n = 17. | 3925mm | ,55251 | | |
| ŝ | Machining p | rofiles opt | ions | | | | | \$ |
| Z I | ake into acco | ount the st | ock shape | | 🗸 Take into a | ccour | nt the finish shape | |
| é (| Overlap | | | | | | | \$ |
| Step | over | | | | External clear | ance d | listance | |
| 25,2 | ?mm | 40% | | | 0,5mm | | | |
| é. | Stocks to lea | ve and ste | eps | | | | | \$ |
| Stoc | ck to leave on | floor | | | Maximal axial | depth | ı | |
| 0mi | m | | | | 5mm | | | |
| Stoc | ck to leave on | wall | | | Final axial dep | oth pa | SS | |
| 0mr | m | | | | 0mm | | | |
| Stoc | ck to leave on | wall islan | d | | Final axial fee | d rate | F Machining | |
| 0,3r | nm | | | | | | = 1000mm/min | |
| Stoc | ck to leave on | wall shift | | | | | | |
| 0mr | m | | | | | | | |
| | Organization | of strate | gies | | | | | \$ |
| Ord | er of the path | , | | | 📕 Order bv | pocke | ets | |
| ٩. | | | | - 1 | | | | |
| * | strategy | | | | e | | | ~ |
| Mill | ing direction | | | - | Climb | | | |
| Z pa | ath stock fittir | ng strategy | / | 4 | Stock | | | |
| End | milling strate | egy | | • | 🤝 Sweepin | g | | |
| | learance off : | stock | | 1 | | | | |
| Roll | -in Approach | | | | Standard | | | |
| S | trict mode sw | veeping | | | - | | | |
| Swe | eping strateg | w | | i i | Zigzag cy | cle ra | w | |
| | Ainimum patł | h number | | - | | | | |
| | | | | | 🥯 Aftar aw | aanin | _ | |
| swe | eping contou | aning strate | -av | | And SW | cebiii | 9 | |
| Swe | eping over le | ngth | | | | | | _ |
| ion | | | | | Ph | | | |
| Swe | eping angle i | mode | | | Value | | | |
| Swe | eping angle | | | | | | | |
| 0° | | | | | | | | |
| Swe | eping end pa | th strategy | / | 1 | Offset | | | |
| Link | strategy for | free place | | - | Work Ard | C | | |
| Min | imal length o | of the holes | 5 | | | | | |
| 2mi | m | | | | | | | |
| Rep | osition cleara | ince | | | | | | |
| 0 4 r | nm | | | | | | | |

Note: As you can see, once the tool is selected, it automatically orients itself.

• Click on 💙 to **confirm** the operation.

Machining the large inclined planes



To machine the large inclined planes, you can reproduce the same operations as before or use the **Ctrl** + dragand-drop technique by dragging the first machining operation onto the new face in the graphics area.

To reproduce the machining on the other inclined plane, proceed as follows:

- From the NC Operations tree, left-click on the 💙 👬 3: Facing operation.
- Place the mouse cursor over the tool path that appears in the graphics area. The color turns red.
- Press the **Ctrl** key.
- While holding down the **Ctrl** key, left-click on the tool path and drag and drop it onto the new face.

The operation is then copied identically. You can edit some settings.

- Kedit the operation and click on the Settings icon.
- In the Link strategy for free place field, select the **5** Overlength link direct at working feed rate option.

The machining previews of the two inclined planes should be as follows:





- Click on V to confirm the operation.
- 🔛 Save the document.

Step 5 – Machining the center groove

• Open the *Support_Step_5* machining document.

For this step, we will machine a center groove. We will use a method that consists of selecting the geometry as late as possible.



- From the **2D/3D** tab, select the **C** End Milling 2D machining command.
- In the **Tool choice** option, select the **radiused mill** with a 32mm diameter and 3mm radius from the tool magazine.
- In the Geometry option, select the blue-colored plane as shown above.
- Click on the 🕨 icon to update the graphics area.

On the label in the graphics area, you can see that **TopSolid** has automatically recognized the open pocketing operation.



In the Settings option, enter zero stock to leave and keep all other default settings.

<u>Note</u>: This operation could have been included in Step 3. In fact, we could have selected this geometry during the first open pocketing operation. We could also have used the copy and paste method.

• Click on \checkmark to **confirm** the operation.

Step 6 – Machining the outer section

• Open the *Support_Step_6* machining document.

We will machine the outer section of the part by using a contouring cycle.



- Right-click on the blue-colored face as shown above.
- Select the **// Side Milling** command.
- In the **Tool choice** option, create a *32mm* diameter **side mill** with a total length of *120mm* and an output distance of *90mm*.



Click on V to confirm the tool.

- In the •
 - **Settings** option, adjust the following settings:
 - Stock to leave on floor: -1mm -
 - Stock to leave on wall: 0mm _
 - Maximal axial depth: 15mm -
 - Number of radial passes: 2 -
 - Final radial depth pass: 16mm -
 - Z path stock fitting strategy: Select Fit the stock -

Side Milling : Settings				x
	Jead in and out Altitudes			
Plunge	HSN	1	Angle Management	
Stock : 40mm Machined Sto	ock + -1mm Stock l	.eft = 41mm		
Passes : 2 x 13,6667mm + 1	1 x 13,6667mm = 4	1mm		
	tions	I Tala interne	and the Calabahan	*
I ake into account the st	ock snape		count the finish shape	
	lachine			
Stocks to leave and st	eps			*
Stock to leave on floor		Maximal axial	depth	
-1mm		15mm		
Stock to leave on wall		Final axial dep	th pass	
0mm		0mm		
Through machining				
Over depth in through mad	chining			
0,5mm				
💪 Radial passes				\$
Maximal radial depth		Number of rac	fial passes	
=8mm		2		
Final radial depth pass				
16mm				
🚊 Add spring pass				*
Final radial pass		🗌 On all axial	passes	
🗌 🔒 Use helical mode				*
🚽 Technology				\$
Milling direction		ず Climb		
Compensation method		Corrected	i	
Compensation diameter				
=32,05mm				
Use compensation code	:			
✓ Without extra fillet				
Corner rounding angle				
90°				
Z path stock fitting strateg	у	📥 Stock		
Reposition clearance				
0,4mm				
• Organization of strate	gies			\$
Radial passes order strateg	у	Crder by	offsets	
Order of the path		📑 Order by	pockets	

<u>Note</u>: You can select any face on the part's contour. The choice of face only determines the end and return points of the tool.

A negative stock to leave on floor is left in order to avoid having to redo the contour during a possible additional phase.

The machining preview should be as follows:



- Click on \checkmark to **confirm** the operation.
- 🔛 Save the document.

Step 7 – Machining the three steps



• Open the *Support_Step_7* machining document.

We will machine the three steps by using a multiple pocketing cycle.

To perform this machining, we will make a multiple selection by holding down the **Ctrl** key while selecting the three blue-colored faces.

- Hold down the **Ctrl** key and select the three blue-colored faces as shown above.
- Right-click in the graphics area and select the 😕 End Milling command.
- In the **Tool choice** option, create a 22mm diameter **radiused mill** with a radius of 3mm, a total length of 120mm and an output distance of 80mm.



• Click on 💙 to **confirm** the tool.

- In the Gettings option, adjust the following settings:
 - Stock to leave on floor and Stock to leave on wall: Omm
 - External clearance distance: Omm
 - Maximal axial depth: 15mm
 - Final axial depth pass: 0mm
 - Z path stock fitting strategy: Select Fit the stock
 - End milling strategy: Select Successive contouring

End I	Milling : Setti	ngs								x
عا	Settings	É	Altitude	s	*	Plunge			Contouring integrat	ted
-	Islands f	facing		High	n Speed	d Machining			Volumill	
Stoc	k:13mm Ma	chined S	tock + 0	mm S	tock Le	eft = 13mm				
Pass	es:1x13mn Machining pr	n = 13m rofiles o	m ntions							
	ake into acco	unt the	stock sha	ne		I Take into	acco	unt	the finish shape	~
	ake into acco	unt the	SLOCK SHE	ipe			ottom	unit urad	ius	
15	O						011011			
	ovenap									*
Step	over		ne/			External cle	aranc	e dis	tance	_
0,41						umm				
: <mark>ک</mark>	Stocks to lea	ve and s	teps —							*
Stoc	k to leave on	floor				Maximal ax	ial de	pth		
0mi	m					15mm				
Stoc	k to leave on:	wall				Final axial d	lepth	pass		
Om	m					Umm			÷	
Stoo	ck to leave on	wall isla	na			Final axial fe	eed ra	te	Machining	
U, Sr	nm skite lesve en	نباء الحس	£1.						= 2604mm/min	
0m	m	wan shi								
			!							
[•• '	organization	orstrat	egies							*
Ord	er of the path	1				Crder b	oy poo	cket	5	
• آھ	Strategy									\$
Mill	ing direction					🍶 Climb				
Z pa	ath stock fittir	ng strate	ду			Stock				
End	milling strate	egy				Succes	sive c	onte	ouring	
Poc	kets milling s	trategy				ø By inte	rnal.			
	imple geome	try optin	nization			Use the r	nixed	mill	ing direction	
	learance off	stock				Respect f	the st	ep o	ver	
Roll	-in Approach					🖒 Standa	rd			
Link	mode					🕎 S				
Link	feedrate					F Machin	ing			
						= 2604mm/i	min			
Rep	osition cleara	nce								
0,4r	nm									

- Click on 💙 to **confirm** the operation.
- 月 Save the document.

Step 8 – Performing the drillings



- Open the *Support_Step_8* machining document.
- Select the blue-colored cylindrical face as shown below.



• Right-click in the graphics area and select the **Drilling** > **I** Hole Machining command.

	2D		1	
کا	End Milling			
\mathbb{P}	Side Milling			
	Drilling	•	$\left\{ \mathbf{r}\right\}$	Hole Machining
4	Roughing			Apply Method
	Vertical Milling	•	-	
	Others	•		

The label provides some information, including the hole diameter (diameter of 10mm in our case).



Click on the Comparison of the search options, check the Same diameter box and uncheck the Same depth, Same altitude, Respect Through Hole, Same machining attribute and Same color boxes.

-52	Search options
	Same diameter
	Same depth
	Same altitude
	Respect through hole
	Same machining attribute
	Same color

- Click on the $\overline{\xi}$ icon to search for similar holes. We have five holes to drill.
- Do not include the partial cylinders in the analysis.
- Click on the Market Tool choice icon.
- Double-click on the **Twist Drill** thumbnail. By default, the tool diameter is the hole diameter. Adjust the total length of the drill to *120mm* and the output distance to *90mm*.
- **Confirm** the tool, then **confirm** the operation.

The drilling operation is achieved.

• Repeat the procedure to drill the two holes shown in blue below.



We will now machine the spot faced holes.

• Select the blue-colored cylindrical face as shown below.



- Right-click in the graphics area and select the **Drilling** > **I** Hole Machining command.
- In the Comparison of the same diameter box and uncheck the same depth, same altitude, Respect Through Hole, same machining attribute and same color boxes.

💐 Search options	_
Same diameter	٦
Same depth	
Same altitude	
Respect through hole	
Same machining attribute	
Same color	

- Click on the 😳 icon to search for similar holes. We have three holes to drill.
- On the label, in the **Kind of machining** field, double-click on the **U** icon and select **Spot Facing**.
- In the **Spot Face Mill** thumbnail. By default, the tool diameter is the hole diameter. Adjust the total length of the drill to *120mm* and the output distance to *90mm*.
- **Confirm** the tool, then **confirm** the operation.

The spot facing operation is complete.

• 😼 Save the document.

Step 9 – Performing the borings



• Open the *Support_Step_9* machining document.

We will machine the blue-colored bores as shown above. First, we will load the tools in the tool magazine.

- From the **Equipment** tab, select the **Tool Manager** command.
- Double-click on the **Twist Drill** thumbnail, then create a *39.2mm* diameter drill with a total length of *120mm* and an output distance of *90mm*. Click on ✓ to **confirm**.
- Double-click on the **Boring Bar** thumbnail, then create a 40mm diameter bar with a total length of 120mm and an output distance of 90mm. Click on ✓ to **confirm**.





The tools are added to the tool magazine.

- Right-click on one of the blue-colored cylinders as shown on the previous page.
- Select the **Drilling** > | | Hole Machining command.
- Select the 39.2mm diameter drill as the tool.

You can change the orientation of the drilling by double-clicking on the yellow arrow (circled in red below) located in the axis of the drilling in the graphics area.



The WCS solution is then automatically modified.

The two faces can be drilled either in a single machining operation, or in two machining operations.

 Repeat the operation by selecting the boring bar as the tool and adjusting the drilling type to Reaming in the Kind of machining field of the label.

Note: If the tool does not appear in the toolbar when selecting the tool, you only have to right-click and select

陇 Show all pockets.

As a standard, 98% of the nominal diameter should be bored.

• 😼 Save the document.

Step 10 – Alternative machinings



• Open the *Support_Step_1* machining document again.

We have previously seen how to completely machine a 2D part. Here, we will see a process that could reduce the amount of finishing operations. This roughing could potentially save you time (if using the same mill!).

- To do this, open the machining in its initial state (no machining operation).
- Right-click in the graphics area and select the 😫 Roughing command.



For this operation, we will machine the maximum material. The color planes display the machining bounds. You can manually move them or define them exactly.



- Click on the ORE Geometry icon.
- Go to the **Bounds** tab.
- Leave the default bounds, except for the Z minimum, which will be 18mm. To enter the Z minimum, first click on the black triangle icon, then click on the **Value** icon and enter *18mm*.



In the Store option, create a 32mm diameter side mill with a total length of 120mm and an output distance of 90mm.

2D Milling

• On the label, adjust the stock to leave on floor to 1mm, the stock to leave on wall to 0mm and the computing tolerance to 0.01mm.

Kind of machining	000
Time	00:00:00:000
Computing tolerance	0,01mm
Stock to leave on floor	1mm
Stock to leave on wall	0mm
Axial depth	7mm
End milling strategy	٩

• Click on \checkmark to **confirm** the roughing operation.

In top view, the machining tool paths should be as follows:



By default, the stock is not updated.

In order for the operation to update the stock, *edit* the roughing operation, click on the *Machine* Dialog Flags icon and select the *Update stock* option.

Roughing	
🖦 🖧 🛫 🤹 🙏	/ 🚅 👑 💕 🕄

<u>Note</u>: If you want to redo the machining by starting with a roughing operation, note that for the facing operations you must select multiple geometries.

Conclusion

At the end of the cycle, the part is practically machined and the following result is obtained.



The **Verify** tab allows you to check the machining and assess the tool paths of each tool. Each color corresponds to a different tool.

Some stocks to leave may not be machined, which is why you can use the **Comparison** tool to measure the distance between the finish and the fictitious stock. If the part is green, this distance is assumed to be zero. If it is closer to blue, not enough material was machined. Alternatively, if it is closer to red, too much material was machined.



You can see here that only the boring chamfers are not machined.

The purpose of this chapter is to introduce you to **TopSolid 7**'s Cam module and present its range of 2D features. By no means are the order of operations, choice of tools, or operation settings optimized or defined for industrial use.

3D Milling

Introduction

The purpose of this chapter is to create a machining program for the following part.



This is a sinking electrode for a perfume bottle mold.

To help you become familiar with **TopSolid 7**, the machining of the part above is divided into several steps. For each step, a machining document is available in which the operations have been completed. You can refer to these documents if you run into any difficulties. The machining steps are numbered from 1 to 4, so there will be four numbered machining documents. These machining documents are provided in a TopPkg file named *TopSolidCam_3DPartStepByStep.TopPkg* (during installation, this file is installed in the "Program Files\TOPSOLID\TopSolid 7.14\Samples" folder).

This tutorial is based on the assumption that the operator has experience with **TopSolid'Design** features and that they have already become familiar with the 2D features. It is therefore assumed that the user is familiar with the finish part, stock, machine, and tool concepts.

The overall programming approach is as follows:

- 1. Selecting the machine tool
- 2. Defining the stock part
- 3. Positioning and defining the part (as per TopSolid'Cam) and its program origin
- 4. Defining the work coordinate systems automatically (angular positions that must take the two machine tool's plates)
- 5. Defining the operations with integrated stock management
- 6. Simulation
- 7. Editing the ISO file

Given that this is a 3D tutorial, we will directly begin at Step 5. Therefore, the Cam part has already been created and the tool magazine is loaded with the required tools.

Machining

As you saw in the 2D tutorial, **TopSolid** suggests two approaches to perform a machining.

• Subject – Verb:

Select the entities to be machined, and then select a machining command.

With this solution, you can select one or more faces with the mouse and indicate the machining on which to apply the faces.

• Verb – Subject:

Select a machining command and indicate the entities on which to apply this command.

For 3D machinings, these two methods can be applied. Given that the entire part is machined, selecting a face opens the 3D machinings even though the entire part will be machined by default. You can also right-click in the graphics window to avoid having to select a face.

Step 1 – Roughing

- Create a **hew project** named *3D Machining* using a **blank template**.
- **** Import** the *TopSolidCam_3DPartStepByStep.TopPkg* package in your project.
- Open the *Nounours_Step_1* machining document.

During this step, we will completely rough the model. This operation consists of removing the maximum amount of material with a dimensioned roughing tool in order to remove the greatest amount of material with the least amount of cutting effort.



• Right-click in the graphics area and select the GRoughing command.



The color planes indicate the machining bounds, which means that the material will be machined within these limits.



Note: You can hide the color planes by disabling the 💜 Preview icon at the top right of the TopSolid window.

- **Preview** enabled: The color planes are visible as shown in the image above.
- Mereview disabled: The color planes are invisible as shown in the image below.



- In the **Tool choice** option, create a 20mm diameter **side mill** with a total length of 72mm and an output distance of 50mm.
- Click on the Settings icon and adjust the following settings:

Diverses	Alti	Collinia	Teal I Carr
Plunge and retract		Collisions	I ool definition
Computation setting	ngs		\$
Computing tolerance		Stock to lea	ive on wall
),05mm		0,5mm	
Machine everywhere	2	Stock to lea	ive on floor
Vinimal material left		0,5mm	
),1mm		Use Z Bu	iffer
Overlap			\$
Step over			
10mm		50%	
external clearance dista	nce		
2mm			
Organization of st	rategies		
			<u> </u>
Order of the path	y	📑 Order	S pockets
Order of the path		Ter Order	s by pockets
Order of the path		💼 Order	\$
Order of the path Strategy Milling direction End milling strategy		Climb	sive contouring
Order of the path Strategy Milling direction End milling strategy Pockets milling strateg	y	Climb © Succes	sive contouring
Order of the path Strategy Milling direction End milling strategy Pockets milling strateg	у	Climb Succes	sive contouring ernal.
Order of the path Strategy Milling direction End milling strategy Pockets milling strateg	у	Climb Climb Success Sw By Inte	sive contouring ernal. mixed milling dire the step over
Order of the path Strategy Milling direction End milling strategy Pockets milling strateg Roll-in Approach	y	Climb Climb Succes By inte Duse the Respect	sive contouring ernal. mixed milling dire the step over
Order of the path Strategy Milling direction End milling strategy Pockets milling strateg Roll-in Approach Link mode	у	Climb Climb Success By inte Use the Respect C Standa S	sive contouring rnal. mixed milling dire the step over rd
Order of the path Strategy Milling direction End milling strategy Pockets milling strateg Roll-in Approach Link mode	y	Climb Climb Succes By inte Use the Respect Standa S S	by pockets sive contouring rmal. mixed milling dire the step over rd
Order of the path Strategy Milling direction End milling strategy Pockets milling strateg Roll-in Approach Link mode Link feedrate	y	Climb Climb Success By inte Use the Respect C Standa S S Machin	by pockets sive contouring rnal. mixed milling dire the step over rd ning
Order of the path Strategy Milling direction End milling strategy Pockets milling strateg Roll-in Approach Link mode Link feedrate	y	Climb Climb Success By inte Use the Respect Standa S S Landa S S	by pockets sive contouring rnal. mixed milling dire the step over rd ning min
Order of the path Strategy Milling direction End milling strategy Pockets milling strateg Roll-in Approach Link mode Link feedrate Reposition clearance	y	Climb Climb Succes By inte Use the Respect Standa S S F Machin = 2865mm/	by pockets sive contouring rmal. mixed milling dire the step over rd ming min

Settings Altitudes Strategy Plunge and retract Collisions Tool definition Image: Machining altitudes Image: Axial depth Image: Axial depth Image: Smm Image: Axial depth Image: Axial depth Image: Smm Image: Axial depth Image: Axial depth Image: Smm Image: Axial depth Image: Axial depth Image: State of the	Roughing (Rough	ing)	: Settings	×		
Plunge and retract Collisions Tool definition Image: Machining altitudes Image: Axial depth Image: Axial depth Image: Scale of the second secon	Settings		Altitudes	Strategy		
 Machining altitudes Axial depth Smm Planar area machining Planar area step over 10mm Scallop optimization Manual selection of altitudes 	Plunge and retract		Collisions	Tool definition		
Imm □ = Scallop optimization □ = Manual selection of altitudes	Image: Second					
	10mm □ ≜ Scallop optimization ¥ □ ≜ Manual selection of altitudes ¥					

🚪 Roughing (Roughin	ig) : Settings	×
Settings	Altitudes	Strategy
Plunge and retract	Collisions	Tool definition
[®]		*
Retraction if necessar	у	
Optimized retraction		
Clearance altitude		
47mm		
- I		
Automatic		
Shift		
🔞 Inside Material —		*
Plunge strategy	Rampir	ng
Slope angle		
3°		
Minimal radius for slope	e plunge	
0mm		
Down feed rate	F Machir	ning
	= 2865mm/	min
🥩 Outside Material -		\$
Plunge strategy	🚽 Direct	
Down feed rate	F Rapid	
	= Rapid	
Safety distances		*
Safety distance		
2mm		

- In the **Settings** tab:
 - **Computing tolerance**: 0.05mm
 - Minimal material left: 0.1mm
 - Stock to leave on wall and Stock to leave on floor: 0.5mm
 - External clearance distance: 2mm
 - Step over: 10mm
- In the **Altitudes** tab:
 - Check the **Planar area machining** box.
 - Planar area step over: 10mm
- In the **Plunge and retract** tab:
 - Check the **Retraction if necessary** box
 - Adjust the inside material plunge strategy to Ramping and the angle to 3°

Note: It may take a while to update the stock, which is why it is not selected by default.

Roughing operations do not require an updated stock since the current stock is transparently calculated so that the remaining material to machine is known. The roughing operation analyzes all the previous active machining operations to know the actual shape of the stock at this point.

For 2D operations, if the stock is updated, the default start Z values will be correct. If not, they must be updated.

By default, the stock is not updated.

In order for the operation to update the stock, click on the W Machine Dialog Flags icon and select the
 Update stock option.



• Once all of these settings are defined, click on \checkmark to **confirm** the roughing operation.

The machining previews $\overline{\mathfrak{so}}$ should match the previews shown below.



The verification module quickly captures a realistic image of what will actually happen on the machine. It is important to confirm this step when the stock is not updated. You can view the state of the stock after this roughing.

- To use this module, simply select the operations to view from the NC Operations tree, right-click on them and select the Verify command.
- Click on the **Start animation mode** icon.

Note that you can modify the transparency by right-clicking on the part and moving the transparency cursor.



<u>Note</u>: The icon allows you to start the verification in **Turbo** mode. With this mode, response times are much faster.

Click on the icon to exit the verification module.

Step 2 – Re-roughing



• Open the *Nounours_Step_2* machining document.

The diameter of the previously used mill was relatively large with respect to the size of the part to be machined. It therefore left areas in which the remaining amount of material was too large to be removed only by the finishing tool. We will perform a re-roughing operation that consists of a new roughing operation performed with a smaller diameter tool.

As you saw at the start of this chapter, a roughing takes into account the current state of the stock (even though it is not up-to-date) in order to determine the areas to be machined. The re-roughing command is therefore the same as the roughing command.

It may be worthwhile to copy the roughing that has already been calculated so that you only have to modify a few settings (such as the tool used). This is easily done from the NC Operations tree.

- Select the first roughing operation.
- Simultaneously hold down the **Ctrl** and **C** keys.
- Simultaneously hold down the **Ctrl** and **V** keys in the location where you want to copy (in this case, at the end).

Your screen should look like the following image:

NC Operations 4 X
🕪 f 🙀 📃 🖬 🏰 🛍
(計
🕂 🕅 Programme 1
🖻 📲 Pièce 1
🖶 🚰 1: Side Mill D20 L35 SD20
🖶 🥍 WCS 1 [Origine 1 No rotation]
🔜 👬 👬 🔁 Roughing

A new operation is then created.

- Double-click on the new operation (operation 2) to edit it.
- Click on the I Tool choice icon, then create a 10mm diameter side mill with a total length of 72mm and an output distance of 40mm.

In the Gettings option, adjust the following settings.

Roughing (Roughing) : Settings 🛛 🛛 🗙					
Settings	Altit	udes	Strategy		
Plunge and retract	(Collisions	Tool definition		
Computation setti	ings		*		
Computing tolerance		Stock to leav	ve on wall		
0,05mm		0,5mm			
Machine everywher	e	Stock to leave on floor			
Minimal material left		0,5mm			
0,1mm		🗌 Use Z But	ffer		
🚊 Overlap			*		
Step over					
5mm		50%			
External clearance dist	ance		,		
2mm					
Organization of st	trategies				
	, <u> </u>	-	~		
Order of the path			by pockets		
🚽 Strategy			*		
Milling direction		of Climb			
End milling strategy		Success	sive contouring		
Pockets milling strates	9y	🐲 By inte	rnal.		
		Use the r	mixed milling dire		
		Respect t	he step over		
Roll-in Approach		🖒 Standar	rd		
Link mode		🖅 S			
Link feedrate		F Machin	ing		
		= 5730mm/r	nin		
Reposition clearance					
0,4mm					
Stay in closed pock	et mode				

🐴 Roughing (Roughing) : Settings 🛛 🛛 🗙				
Settings	Altitudes		Strategy	
Plunge and retract	Collisions		Tool definition	
Machining altitudes Axial depth				
C @ Planar area machining				-11
□			×	
☐ ➡ Manual selection of altitudes			¥	
Show altitudes of cuts				

- In the Settings tab:
 - Step over: 5mm
- In the Altitudes tab:
 - Axial depth: 1.5mm
 - Uncheck the Planar area machining box

<u>Note</u>: It is no longer necessary to machine the planar faces since it was already done for the first roughing operation.

• Click on 💙 to **confirm** the operation.



As before, you can check the result by using the verification module.

You can see in the image below that the finishing tool (shown in green) machined the affected areas.



Step 3 – Finishing

• Open the *Nounours_Step_3* machining document.

For this step, we will finish the machining by milling the shape using a 12mm diameter ball nose mill.



• Right-click in the graphics area and select the *Finishing* command. The following label is displayed.

Kind of machining	N
Time	00:00:00:000
Computing tolerance	0,01mm
Stock to leave on floor	0mm
Stock to leave on wall	0mm
Step over	0,5mm
Scallop height	0,0125mm
Sweeping angle	0°

• Double-click on the ⁽¹⁾ icon, which indicates the type of 3D finishing, then select a ⁽²⁾ **Constant step-over** finishing.

Kind of mact ᢤ 4		1
Time	00:00:00:000	
Computing tolerance	0,01mm	
Stock to leave on floor	0mm	
Stock to leave on wall	0mm	
Step over	0mm	
Scallop height	0mm	

The following label should be displayed:

Kind of machining	4
Time	00:00:00:000
Computing tolerance	0,01mm
Stock to leave on floor	0mm
Stock to leave on wall	0mm
Step over	0mm
Scallop height	0mm

• In the **Tool choice** option, create a *12mm* diameter **ball nose mill** with an output distance of *50mm*.



In the Settings option, adjust the following settings.

Finishing (Constant step-o	ver) : Settings X		
Settings	Strategy		
Plunge and retract	Collisions		
Geometry	*		
Computing tolerance	Stock to leave on wall		
0,01mm	0mm		
	Stock to leave on floor		
	0mm		
🗌 🧼 Use ball tool (enable 3d	-5x normal projection)		
Computing	•) *)		
Step over	Scallop height		
0,489694mm	0,01mm		
Max. dist. between points	Add a last offset at the end		
2mm			
3D curve offset		🕌 Finishing (Constant step-over)	: Settings
0mm		Settings	Strategy
Undercut management		Plunge and retract	Collisi
Slope management		Miscellaneous	
May close	Min sland	Trim the path at Z min.	
		Hsm	
50 		HSM spiral angle	
Tool path slope manageme	nt 🛛 🕹	10°	
🐇 Strategy	*	HSM on links	
Clockwise machining	Clockwise	HSM on path	
Start to end	+ ‡→ End to start	HSM smoothing intensity	
Propagation mode	Both inner and outer	0	
Stop propagation when touch	No stop		
Compensation	*	Minimize air cutting	
Commenting mode	No componention cont	On the shape until	
compensation mode	y no compensation sent	0mm	

\$

\$

40

\$

- In the **Settings** tab:
 - Scallop height and Computing tolerance: 0.01mm
 - Start to end: Select End to start
 - Max. dist. between points: 2mm
- In the Strategy tab:
 - HSM spiral angle: 10°
 - Uncheck the HSM on links box
 - Check the **HSM on path** box
- Click on V to confirm the finishing operation.

As before, tool paths can be reviewed which, for this operation, are as follows:



We will now trim the finishing pass above the upper plane. We could use the Z minimum, but for this example, we will see how we can also trim the machining with a sketch.



Click on the *f* icon at the top left of the screen to switch to design mode.

Note that you can also use the **Ctrl** + **W** default shortcut to switch between the machining and the design modes, and conversely.

- Right-click on the blue-colored plane as shown on the previous page and select the *Sketch* command.
- **Solution** Project the outer contour of the blue-colored surface.
- **Confirm** the sketch by clicking on the Sketch 1 button.
- Switch back to machining mode by clicking on the random in the system bar or using the Ctrl + W shortcut.
- Section Edit the finishing operation.
- Click on the 🥙 Geometry icon.
- In the **Bounds** tab, adjust the following settings:
 - Bounding mode: Select On
 - Make sure that the **Bounding curves management** box is checked.
 - Select the sketch as the bounding curve.



Click on V to confirm the finishing operation.

You should end up with the following result:



As a reminder, the verification controls the removed material. Therefore, you can see that the ball nose mill machined a large amount of material and thus finished the part in several areas (shown in blue below).



However, some material remains to be machined because the diameter of the finishing tool was not able to pass over the entire part.

You can request that the verification module clearly indicates the unmachined areas by comparing the current stock to the theoretical part to be machined. This also indicates whether a problem occurred.

Always in the verification context, right-click in the graphics area and select the *Isplay comparison* command, or click on the *Isplay icon* at the top left of the screen.

You should end up with the following result:



As you can see, the closer the color is to blue, the more the material remains to be machined. In this case, it is clear that there is a lot of material left to be machined. We will therefore perform a left material machining using a smaller radius.

Step 4 – Re-finishing

• Open the *Nounours_Step_4* machining document.

For this step, we will machine the areas that were not machined in the previous operation because the tool diameter was too large. We will therefore redo the finishing using a reference tool. The reference tool will be the one that was used in the previous step.



• Right-click in the graphics area and select the *Material left* command.



The following label is displayed.

Kind of machining	
Time	00:00:00:000
Computing tolerance	0,01mm
Stock to leave constant	0mm
Axial depth	0,6922mm
Scallop height	0,02mm
Reference tool diameter	10mm
Reference tool corner radius	5mm

• Double-click on the *icon,* which indicates a constant Z left material machining, then select **Radius machining**.

Kind of machinin	(‡ 🐼 🕢 🗳
Time	00:00:00:000
Computing tolerance	0,01mm
Stock to leave constant	0mm
Step over	0,6922mm
Scallop height	0,02mm
Reference tool diameter	10mm

- In the OT Tool choice option, create a 6mm diameter ball nose mill.
- In the Settings option, adjust the following settings:
 - Adjust the **computing tolerance** and the **scallop height** to *0.01mm*.
 - Since the reference tool in the previous step was a 12mm diameter ball nose mill, adjust the **reference tool diameter** to *12mm*.
 - Adjust the angular separation to 75°.

🕌 Material L	eft (Radius Machi	ning) : Settings	_	×
Settings	Plunge and retract Collisions				
🕲 Geometry				\$	n
Computing tolerance Stock to leave constant			ant		
0,01mm 0mm					
🗌 🕲 Use b	Use ball tool (enable 3d-5x normal projection)				zl
। ∂ 🧐 Computi	ing				3
Step over	-	Scal	lop height	^	·
0,346121mm	1	0,01	mm		١١
Max. dist. be	tween points	Min	imal path length	allowed	-
1mm		1m	n		١١
Maximal link	distance	Min	imal separation I	ength	-
=3mm		1m	m	-	ור
C 🧐 Referen	ce tool			~	레
Reference to	ol diameter	1		^	
12mm					
- @ Slope m:	anagement				_
Siope In	anagement			*	
Max. slope		Ang	ular separation		-
90		//5 //5			-
Horizontal p	rofiles machining	4	Climb mill		
Corner strate	зу	Ŋ,	Two by two linl	cs	
Vertical prof	iles machining me	Ş	Climb mill		
Vertical swee	eping mode	b	Spiral		
Compensati	ion			\$	E)
Compensatio	on mode	V	No compensati	on sent	

Click on V to confirm the material left operation. During calculation, information is displayed at the bottom left of the TopSolid window.

As in previous steps, we will verify the machining and tool paths. The stock is updated in machining operations 1 and 2, so we have to select machining operations 3 and 4 before launching the verification.



As you can see here, some material remains to be machined. You must therefore repeat the material left operation by reducing the diameter of the mill. For the parting surface, in order to keep the sharp angle, you can use a side mill to contour the shape (see 2D chapter).

The purpose of this chapter is to introduce you to **TopSolid 7**'s Cam module and present its range of 3D features. By no means are the order of operations, choice of tools, or operation settings optimized or defined for industrial use.
Turning

Introduction

The purpose of this chapter is to create a machining program for the following part.



As you can see, the part must be positioned on a turning machine in order to be machined.

Warning: If the default machine is not a turning machine, right-clicking on the design document and selecting the

Machining command will not be suitable.

- In this case, select the 🕮 **Options** command from the **Tools** tab.
- Expand the CAM Options menu, go to the Machines section and select the NC Machine Turning (XZ/C) machine.

111 Options					\times
🛓 🖳 😫 Assembly	0	NC Machine Milling (Head XYZ A)	TopSolid Machining		^
Bom	0	NC Machine Milling (Head XYZ B)	TopSolid Machining		
Buildings	0	NC Machine Milling (Head XYZ / Table	TopSolid Machining		
	0	NC Machine Milling (Head XYZ / Table	TopSolid Machining		
Cutting Conditions	0	NC Machine Milling (Head XYZ A / Tabl	TopSolid Machining		
	0	NC Machine Milling (Head XYZ A / Tabl	TopSolid Machining		
Machining Cloud	0	NC Machine Milling (Head XYZ B / Tabl	TopSolid Machining		
Tools	0	NC Machine Milling (Head XYZ)	TopSolid Machining		
Oser parameters Dialog configurations	0	NC Machine Milling (Head XYZ CA)	TopSolid Machining		
Display Options	0	NC Machine Milling (Head XYZ / Table	TopSolid Machining		
-12 Link movements	•	NC Machine Turning (XZ/C)	TopSolid Machining		
		-			~
Method				Reset	
NC Operations Manager					
		✓ × ?			

To help you become familiar with **TopSolid 7**, the machining of the part above is divided into several steps. For each step, a machining document is available in which the operations have been completed. You can refer to these documents if you run into any difficulties. The machining steps are numbered from 1 to 7, so there will be seven numbered machining documents.

These machining documents are provided in a TopPkg file named *TopSolidCam_ArbreCourt.TopPkg* which is installed in the "Program Files\TOPSOLID\TopSolid 7.14\Samples" folder. In order to follow the tutorial, we recommend that you import this file into a new project.

- Create a **W** new project named *Turning* using a blank template.
- **W** Import the *TopSolidCam_ArbreCourt.TopPkg* package in your project.

The part was designed in **TopSolid**. The designer used the "features" technology, which allows major automation of machinings.

The overall programming approach is as follows:

- 1. Defining the stock part
- 2. Positioning and defining the part (as per TopSolid'Cam) and its program origin
- 3. Defining the operations with integrated stock management
- 4. Simulation
- 5. Editing the ISO file

Steps 1 and 2 are explained in the prerequisite tutorial (creating a machining setup document and machining document). We will start at Step 3, which consists of defining the machining operations.

As you have previously seen, several methods are available for creating a machining document. You can therefore start the machining by selecting the method of your choice. For this chapter, method 5 will be selected, which consists of machining an assembled stock and finish.

In this chapter, we will perform various basic turning operations, including:

- Facing
- Turning
- Groove roughing
- Groove machining
- Threading
- Hole machining

Step 1 – Facing the end of the part

• From the Project tree, open the *Arbre_Court_1* machining document.

During this step, we will machine the right end of the part. We will plunge into the material with a facing tool oriented along the X axis.



- Select the extreme right face shown in blue below.
- Right-click in the graphics area and select the 💶 Roughing command.



Click on the Tool choice icon.

The following dialog box appears:



- Select an empty pocket in X+. If you select Z+, the tool will not be oriented properly.
- Double-click on the C-Rhombic ... C-Rhombic 80 thumbnail.
- In the **Template(s)** tab, double-click on **C80-Rhombic L External Turn**.



• Click on 💙 to **confirm** the tool.

The default machining direction is based on the first selected face. The default value will be $+90^{\circ}$ or -90° or 0° or 180° .



Since the pass depth is too large, you must modify its value.

- On the label, change the values as follows:
 - Stock to leave constant = 0.3mm
 - Pass depth = 2mm

Kind of machining	
Time	00:00:00:000
Stock to leave method	
Stock to leave constant	0,3mm
Pass depth	2mm
Pocket plunge	No
Machined zone	
Tool Path Preview	Yes

• Click on 💙 to **confirm** the operation.

If you want to simulate the machining operation, you only have to click on the soperation icon in the NC Operations tree, or right-click on the desired operation and select the Simulate command.

• 月 Save the document.

Step 2 – Turning

• From the Project tree, open the *Arbre_Court_2* machining document.

For this step, we will execute the same paraxial roughing operation as before, but this time the tool path will be oriented along the Z axis and not along the Y axis.

• Click on one of the blue-colored faces as shown below, then hold down the **Alt** key and click on the second face.



<u>Note</u>: The **Alt** key allows you to select all the faces between the two selected faces (included).

• Right-click in the graphics area and select the **Q** Roughing command.

The default machining direction is 180° and the previously chosen tool is still selected.

Kind of machining	
Time	00:00:00:00
Stock to leave method	
Stock to leave constant	0,3mm
Pass depth	2mm
Pocket plunge	No
Machined zone	
Tool Path Preview	Yes

• Click on 💙 to **confirm** the o>peration.

The part preview should be as follows:





Machined part

Tool path

Bave the document.

Step 3 – Groove roughing

• From the Project tree, open the *Arbre_Court_3* machining document.

For this step, we will machine the cone-shaped section of the shaft using the groove roughing technique, i.e. machine in the direction of the tool axis.

• Hold down the **Alt** key and select the two blue-colored faces as shown below.



- Right-click in the graphics area and select the 🥨 Roughing command.
- On the label, double-click on the 🔤 Kind of machining icon and select 🔛 Groove Roughing.

Kind of machining	
Time	00:00:00:00
Stock to leave method	Groove Roughing
Stock to leave constant	0,5mm
Auto	No
Groove roughing method	
RadialPass depth	0mm
Number of program points	1
Program point	
Machined zone	
Tool Path Preview	Yes

- In the **Tool Choice** option, select an empty pocket in **X+**.
- Double-click on the L-Rectangu... L-Rectangular thumbnail.

• In the **Template(s)** tab, double-click on **L-Rectangular L External Groove**.



• In the **Component(s)** tab, adjust the tool width to *3mm* and the radius to *0mm*.



- Click on 💙 to **confirm** the tool.
- On the label, change the values as follows:
 - Adjust the stock to leave constant to 0.3mm
 - Enable the Auto option: Yes
 - Adjust the **radial pass depth** to 2mm
 - Select 2 program points

<u>Note</u>: Double-clicking in the **Program point** or **Tool Offset XXXXX** box automatically opens the dialog box of the cutting conditions in the **Gauges** tab, which allows you to select the driven points (right and/or left).

Kind of machining	
Time	00:00:00:00
Stock to leave method	
Stock to leave constant	0,3mm
Auto	Yes
Groove roughing method	
RadialPass depth	2mm
Number of program points	2
Program point	Tool Offset 2 Left Auto Tool Offset 3 Right Auto
Machined zone	
Tool Path Preview	Yes

• In the Settings option, go to the Leveling tab and select a Total leveling method.

🗧 Roughing (Groove Roughing) : Settings 🛛 🛛 🗙			
Main	Strategy	Leveling	Stock to leave / limits
Leveling method Total			
Clearance distance			
0,2mm			
Max. step			
10mm			
Leveli	Leveling by successive contouring		
Scallop height maxi.			
0,75mm			

• Click on \checkmark to **confirm** the operation.

You should end up with the following result.



• 月 Save the document.

Step 4 – Groove machining

• From the Project tree, open the *Arbre_Court_4* machining document.

We will now machine the groove.



• Select the blue-colored face as shown below, then right-click and select the 😲 Groove command.



- Go to the 🙆 tool magazine.
- Select an empty pocket in X+.



- Double-click on the L-Rectangu... L-Rectangular thumbnail.
- In the **Template(s)** tab, double-click on **L-Rectangular L External Groove**.



• In the **Component(s)** tab, adjust the tool width to 2mm and the radius to 0mm.



- Click on 💙 to **confirm** the tool.
- In the Settings option, go to the Lead in tab and adjust the lead in over cut to 1mm.

Groove (Groove) : Settings 🛛 🛛 🗙			
Main Geometry	Lead in		
💊 Left ⊃ 🗹 Simil	ar left right		
Lead type	🚄 Direct		
Lead in over cut	Lead radius		
1mm	5mm		
Lead distance	Lead radius angle	Lead radius angle	
2mm	90°		
Lead angle			
45°			
Value of feed rate	Machining		
	= 0.3mm/rev		

- Click on 💙 to **confirm** the operation.
- **Save** the document.

• From the Project tree, open the *Arbre_Court_5* machining document.

For this step, we will remove the remaining stocks to leave resulting from the roughing operations.

• Hold down the **Alt** key and select the two blue-colored faces as shown below.



- Right-click in the graphics area and select the 🔅 Finishing command.
- Go to the ⁽¹⁾ tool magazine.
- Select an empty pocket in X+.



- Double-click on the V-Rhombic ... V-Rhombic 35 thumbnail.
- In the Template(s) tab, double-click on V35-Rhombic L External Turn.



- In the **Component(s)** tab, adjust the insert radius to 0.4mm.
- Click on 💙 to **confirm** the tool.
- On the label, enable the **pocket plunge** option.

Kind of machining	~
Time	00:00:00:000
Stock to leave method	
Compensation method	5
Pocket plunge	Yes
Tool Path Preview	Yes

• In the Settings option, go to the Main tab and adjust the pocket minimum width to 5mm.

🌼 Finishing (Contouring) : Settings	×
Main Stock to leave / breaking edges Lead in / lead o	ut
Opposite side axis work	
Manage tool angles	
☑ Tool path limitation	
Manage tool entry diameter	
	\$
Automatic feed rate variations	
Reduce factor	
2	
3	\$
Use compensation code	
Compensation method France	
Compensation radius	
=0,4mm	
S Machining type	\$
Contouring method Standard	
Association method for blend radius	
None	~
Clearance distance	
0mm	
🗹 🥸 Pocket plunge	\$
Forward clearance angle	
0°	
Backward clearance angle	
0°	
Pockets minimum width	
5mm	
Corner relief	*

- Click on \checkmark to **confirm** the operation.
 - **]** Save the document.

Step 6 – Threading

• From the Project tree, open the *Arbre_Court_6* machining document.

We will now thread the blue-colored face as shown below.



- Right-click on the blue-colored face as shown below and select the 🖤 Threading command.
- Go to the 🕼 tool magazine.
- Select an empty pocket in **X+**.



- Double-click on the ISO ISO thumbnail.
- In the Template(s) tab, double-click on ISO-Metric L-166-KF L External Thread.



- In the **Component(s)** tab, make sure that the step is 1mm.
- Click on ✓ to confirm the tool, and then the operation.
- 📙 Save the document.

Step 7 – Hole machining

• From the Project tree, open the *Arbre_Court_7* machining document.

For some turning operations, it may be interesting to view the inside of the part. Here, in particular, it is useful to create a sectional view for our drilling.

- To create the sectional view, select the 😽 Part Display command from the Turning tab.
- Make sure to check the **Enable turning display** box.



• Click on 💙 to **confirm** the display.

We will now perform the drilling operation.

 Select the blue-colored cylindrical surface as shown below and select the I Hole Machining command from the 2D/3D tab.



- Go to the 🚱 tool magazine.
- Select an empty pocket in **Z**+.

- Create a Ø15mm twist drill.
- Adjust the usable length to 50mm, the total length to 200mm and the output distance to 160mm.



- Click on 💙 to **confirm** the tool.
- On the label, adjust the geometry type to 🔽 Through stock and the rotating element to 🔿 Part.

Kind of machining	U
Time	00:00:00:000
Diameter	15mm
Machined depth	138,6301mm
Rotating element	8
Off center	0mm
Geometry type	
Through hole overdepth	2mm
Tool Path Preview	Yes

• Click on 💙 to **confirm** the operation.

You should end up with the following final result.



• 😼 Save the document.