Tutorial 6: Chamfering



Introduction

This tutorial is intended to show an easy way to chamfer and smooth sharp corners by using 2 $\frac{1}{2}$ axis Chamfering operation. A tapered tool is suitable for this purpose.

We plan to chamfer the edges of the part using a 30 degree taper tool that has 0 radius at the tip.

The stepped instructions are accompanied by explanatory and introductory text. Reading this text will help you understand the tutorial methodology and provide information about additional options available.

Don't forget to save your work periodically! You may want to save the file under a different name so that the original file will be preserved.

Strategy to Machine the part

- Chamfering is performed using the 2 ¹/₂ axis Chamfer Machining Operation.
- The part would be machined using a single V-Groove bit.
- The wooden sheet will be held to the machine table or the spoil sheet on the table using double-sided tape.

Main Programming Steps

In creating programs for each setup, the following steps will be followed:

- Create the Stock geometry
- Set the Machine zero point with respect to the machine coordinates
- Create / Select the tool used for machining
- Set the feeds and speeds
- Set the clearance plane for the non-cutting transfer moves of the cutter
- Select the machining regions for containing the cutter to specific areas to cut
- Select the machining operations and set the parameters
- Generate the toolpath
- Simulate the toolpath.

You may have to repeat either all or part of these steps for subsequent operations.

Loading the Part Model

Part" refers to the geometry that represents the final manufactured product. Typical you would create this in Alibre Design. Use the Alibre Design menu bar or the Standard toolbar buttons to create, load and save part geometry.

- 1. Select File / Open Part, or click the Open Part icon from the Alibre Design standard toolbar.
- From the **Open** dialog box, select the **Chamfer.AD_PRT** file from the **Tutorials** folder in the Alibre CAM installation folder. (Default location C:\Program Files\MecSoft Corporation\Alibre CAM 2.0\Tutorials)

The imported part appears as shown below



Loading the Alibre CAM Browser

1. Select Alibre CAM from the menu bar and click Machining Operations Browser.





The Alibre CAM browser (MOPs and Cutting Tools) is now loaded and is docked over the Design Explorer. You can toggle between the MOPs browser and the Design Explorer from the tabs at the bottom of the window. It can be hidden by un-checking Browser on the Alibre CAM menu bar. To re-display the hidden Browser window, you can re-check **Browser** from the Alibre CAM menu entry. You can also resize it by dragging one of its sides.

Preparing the part for Machining

The Setup tab allows the user to specify Machine Setup, Select Post Processor, Stock Geometry, Machine Coordinate System (Machine Zero) & Preferences.

Setup Tab

 Go to the Alibre CAM MOps browser and click on the Setup tab. The loaded part has the stock model defined and includes a 2 ½ axis Facing Operation with a 0.5" Flat End Mill. The Machining Operation information is listed in the MOps browser as shown below.





Create Tools

To machine the chamfer, we will now create a 60-degree Taper Tool.

1. Go to the Alibre CAM Tools browser that is located below the MOps browser and select Create/Edit Tools. Select the Tool Type to Chamfer.





Set the tool name as ChamferMill1, Taper Angle = 30, Flute Length = 1, and Tool Length = 2. Under the Properties tab, set Tool Number = 2.

Create/Select Tool	
77773	
Tools In Library	Name ChamferMill1 1.6316 H-Holder Diameter -++ Holder Length Taper Angle Taper Angle Flute Length 0.4 D Flat Diameter 0 Comments Comments Comments Comments Comments
	Save as New Tool Save Edits to Tool Delete Tool OK Cancel Help

Note: Taper Angle represents the included angle for a taper tool. For example a 60-degree taper tool would have a included angle of 30 degrees. If you have a taper tool with a diameter select Chamfer Mill or Taper Mill under Create/Select Tool.

Setting Feeds and Speeds

You can assign Feeds & Speeds to a tool or you can load from a table. In this exercise, we will assign feeds and speeds to the tool.

- 3. Switch to the Feeds & Speeds tab inside the create/select tool dialog.
- 4. Use the following settings for feeds and speeds.

Properties F	eeds & Speed	ls
- Spindle Spe	ed 8000	RPM
- Feed Rates		
Plunge:	20	🚔 in/min
Approach:	20	🚔 in/min
Engage:	20	🚔 in/min
Cut:	30	🚔 in/min
Retract:	20	🚔 in/min
Departure:	20	🚔 in/min
Transfer Fe	edrate (Tf)	
💿 Use Ra	pid	
◯ Set	50	🤤 in/min

5. Click **Save as New Tool**. The tool is now created and listed under Tools in Library. Click OK to close the dialog.

Note: You can edit the tool properties and click Save Edits to Tool to save the changes. You can create additional tools by assigning a different name and specify the tool parameters.

Y 🔁 🛍 🛐 🖓
☐ Tools ⊕ ☐ FlatMill-0.5
······₩ ChamferMill1

The created tools are now listed under the Tools browser.

Create Machining Operations

1. Switch to the Create Operations tab in Alibre CAM Mops browser.

Chamfering

2. Select 2.5 Axis Mill and choose Chamfering.



This brings up the Chamfering Operations dialog. We will go over the steps for creating the toolpath.

Select Machining Features/Regions

- 3. Go to the **Machining Features/ Regions** tab and click remove all if any Machining regions are listed.
- 4. Click Select Containment Regions.
- 5. Switch to Design Explorer and select **Sketch3**.



Note: You can also select the sketch from Alibre Design View. When select containment region is selected, Alibre CAM turns on Sketch selection filter to allow the user to select sketches. Hold the Shift key down to select multiple sketches.



to complete the selection.

7. The selected region is now displayed under Machining Regions.





Selecting the Tool

Feeds & Speeds Tool Geometry Diameter Corner Radius Taper Tip Angle Tool Propertie Tool Name Tool # # of Flutes Cutcom Register	 Clearanc V 0 0 30 30 ChamferMill1 2
Tool Geometry Diameter Corner Radius Taper Tip Angle Tool Propertie Tool Name Tool # # of Flutes Cutcom Register	V 0 0 30 0 5 ChamferMill1 2
Diameter Corner Radius Taper Tip Angle Tool Propertie Tool Name Tool # # of Flutes Cutcom Register	0 0 30 0 S ChamferMill1 2
Corner Radius Taper Tip Angle Tool Propertie Tool Name Tool # # of Flutes Cutcom Register	0 30 0 S ChamferMill1 2
Taper Tip Angle Tool Propertie Tool Name Tool # # of Flutes Cutcom Register	30 0 ChamferMill1 2
Tip Angle Tool Propertie Tool Name Tool # # of Flutes Cutcom Register	0 S ChamferMill1 2
Tool Propertie Tool Name Tool # # of Flutes Cutcom Register	ChamferMill1
Tool Name Tool # # of Flutes Cutcom Register	ChamferMill1 2
Tool # # of Flutes Cutcom Register	2
# of Flutes Cutcom Register	
Cutcom Register	2
	0
Adjust Register	0
Z-Offset	0
Material	HSS
Coolant	None
Comments	
E Feeds & Speed	ds
Spindle Speed	8000
Feed Rate	30
Edit/Create/	Select Tool
Edireiedier	566667 001
Previe	w Tool
	Coolant Comments Feeds & Speed Spindle Speed Feed Rate Edit/Create/ Previe

8. Switch to the Tools tab inside the Chamfering operation.

9. Select ChamferMill1. ChamferMill1 is now selected as the active tool, and the Tool parameters are displayed to the right of the Tools window.

Set Feeds and Speeds

10. Click on the Feeds and Speeds tab.

Cut Parameters Machining Features Spindle Speed 8 Feed Rates Plunge (Pf)	Entry/Exi s/Regions	t A Tool RPM	dvanced Cut Pa Feeds & S	rameters Speeds	Sorting Clearance
Machining Features Spindle Speed 8 Feed Rates Plunge (Pf) 2	s/Regions	RPM	Feeds & S	opeeds	Clearance
Spindle Speed 8 8 Feed Rates Plunge (Pf) 2	3000	RPM			
Feed Rates Plunge (Pf) 2					
Plunge (Pf)	-				
	20 🚔	in/min	-	-	
Approach (Af) 2	20	in/min	Pf		
Engage (Ef) 2	20	in/min		Cf	Tf
Cut (Cf) 3	80	in/min	Af		
Retract (Rf) 2	20	in/min	Ef		Rf
Departure (Df)	20	in/min			
Transfer (Tf)					
💿 Use Rapid 🔄					
Set 5	0	in/min			
Load	From Tool				
Load Fr	rom Table				

11. Select Load From Tool. Alibre CAM will now get the feeds and speeds information that was set when the tool was defined.

Clearance Control

12. Switch to Clearance Tab.

Cut Parameters E	ntry/Exit	Advanced Cut Parame	ters Sorting
Machining Features/Regio	ns Tool	Feeds & Spee	ds Clearance
Clearance Plane Definition			Stock Max Z
○ Part Max Z + Dist	0.25		6
◯ Stock Max Z + Dist	0.25		11
⊙ Absolute Z Value दि	0.25 🚔 🔖		
Cut Transfer Method			
O Chin		Clearance	Plane
Skim Clearance (C)	0		
		Part Max Z	

13. Set the Clearance Plane Definition to Absolute Z Value = **0.25** and Cut Transfer Method to Clearance Plane.

Getting Started with Alibre CAM

Setting Cut Transfer to Clearance Plane would apply the Absolute Z value clearance between transfers when the tool moves from a machining region to another.

Specifying Cut Parameters

14. Switch to Cut Parameters tab.

Machining Features	s/Regions Too	Feeds & Speeds Clearance
Cut Parameters	Entry/Exit	Advanced Cut Parameters Sorting
- Global Paramete	ers	Region
Tolerance:	0.001	Toolpath
Stock:	0	Chord Tolerance
Compensation:	AUTO/NONE 🖌	→ ← Stock
Chamfer Parame	eters	
Chamfer Width:	0.125	
Tooltip Clearanc	xe: 0	
Max. Depth/Cut	. 0.025	Width
Right Use Outside. Outside.	/Inside for Closed Curv	res
	sing 3D Model	
- Stepover Contro	JI	Total Cut Width → H
Total Cut Wid	ith: 0	
Step/C	ut: 0	
<u>à</u>		Step & Per Cut

15. Set the Tolerance = **0.001**, Stock = **0**, Chamfer Parameters use Chamfer Width = **0.125**, Max Depth/Cut = **0.025** Cut Side = **Outside**. Click **Generate**. The Chamfering

toolpath is now generated and the Operation is listed under the Alibre CAM MOps browser.

Note: Toolpath display can be turned on/off by selecting Toolpath Visibility under the MOps browser.

Reorder a Machining Operation

If the chamfer operation is created above the 2 $\frac{1}{2}$ axis Facing operation, you can reorder the MOp using the steps below.

- 1. Minimize all MOps inside the Mop Set1.
- 2. Select the Chamfering MOp and drag it over the 2 $\frac{1}{2}$ axis Facing MOp. This would move the Chamfer MOp below the Facing MOp.



Chamfer toolpath is now generated and is displayed below.



Simulate Toolpath

Make sure to turn on Stock Visibility under the Simulate tab.



Chamfering Operation and click by to launch the Alibre CAM Stock Simulation window.

3. Click Simulate from the Stock Simulation window to run simulation.



Note: You can pause/stop, step, and skip the simulation using the controls available on the Simulation window. To control the simulation speed, select Simulation Settings from the Stock Simulation window.

3. Once the simulation is complete, you can close the Stock Simulation window and return to the Alibre CAM browser.

Post Processing

1. Select Machining Operations from the Create Operations tab and right click and select post process.



2. Specify the File Name as **Chamfering.nc** and click Save.

The post by default is set to Haas as specified under the Post processor setup. You can change the post processor by selecting a different one from the drop down menu in the list. The posted g code by default will be saved to the folder where the part file is located.

Lista Studio srl www.lista.it info@lista.it