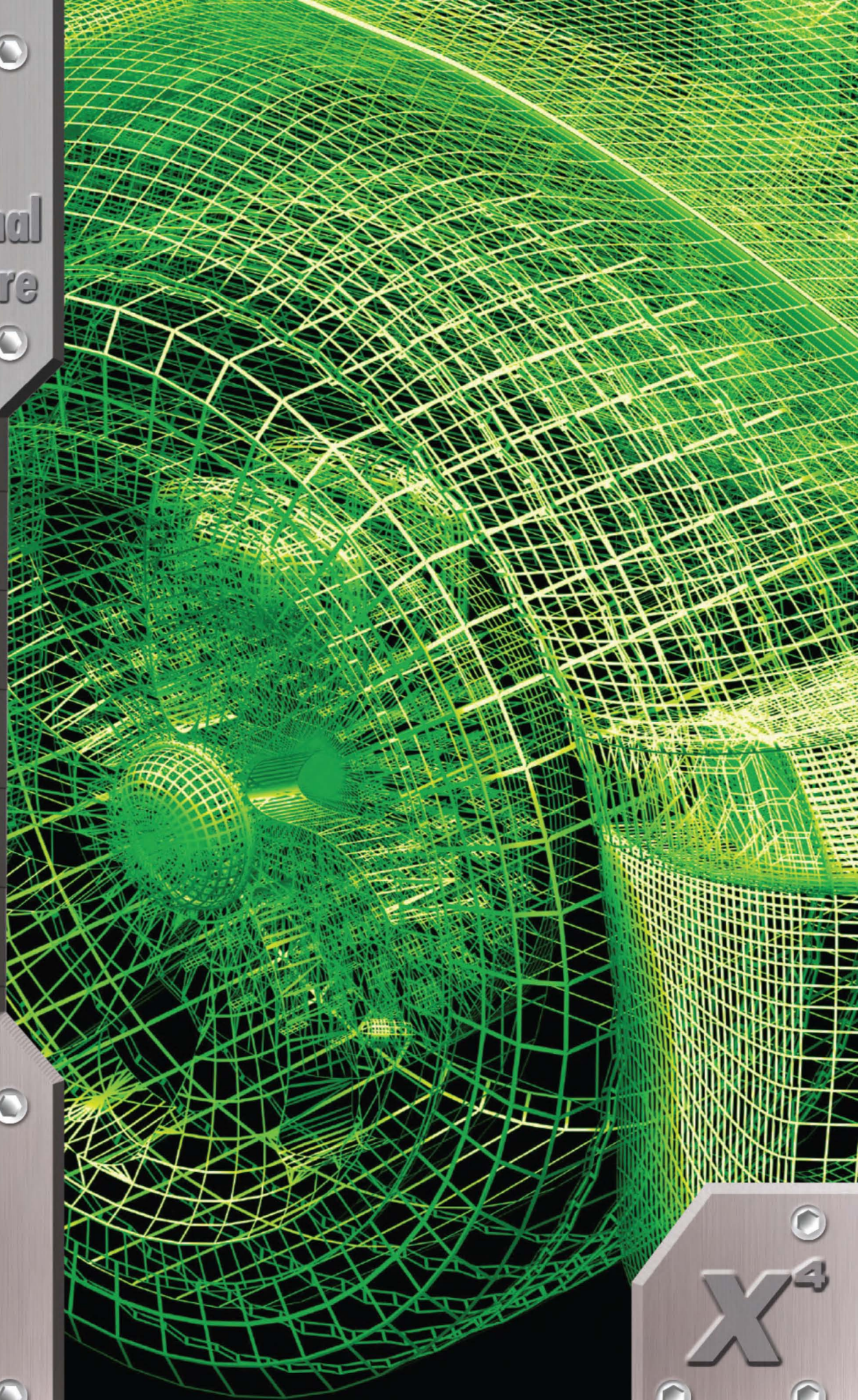


Mill
Level 3
Professional
Courseware

Mastercam®



X⁴



Professional Courseware
Mastercam X⁴ Mill Level 3

To order more books:
Call 1-800-529-5517 or
Visit www.inhousesolutions.com or
Contact your Mastercam Dealer

Mastercam® X⁴ Training Tutorials – Professional Courseware Mill Level 3

Date: June 10, 2009

Copyright © 1984 - 2009 In-House Solutions Inc. All rights reserved.

Author: Mariana Lendel

ISBN: 978-1-926566-31-3

Notice

In-House Solutions Inc. reserves the right to make improvements to this manual at any time and without notice.

Disclaimer of All Warranties and Liability

In-House Solutions Inc. makes no warranties, either express or implied, with respect to this manual or with respect to the software described in this manual, its quality, performance, merchantability, or fitness for any particular purpose. In-House Solutions Inc. manual is sold or licensed "as is." The entire risk as to its quality and performance is with the buyer. Should the manual prove defective following its purchase, the buyer (and not In-House Solutions Inc., its distributor, or its retailer) assumes the entire cost of all necessary servicing, repair, or correction, and any incidental or consequential damages. In no event will In-House Solutions Inc. be liable for direct, indirect, or consequential damages resulting from any defect in the manual, even if In-House Solutions Inc. has been advised of the possibility of such damages. Some jurisdictions do not allow the exclusion or limitation of implied warranties or liability for incidental or consequential damages, so the above limitation or exclusion may not apply to you.

Copyrights

This manual is protected under the copyright laws of Canada and the United States. All rights are reserved. This document may not, in whole or part, be copied, photocopied, reproduced, translated or reduced to any electronic medium or machine-readable form without prior consent, in writing, from In-House Solutions Inc.

Trademarks

Microsoft, the Microsoft logo, MS, and MS-DOS are registered trademarks of Microsoft Corporation; Mastercam Verify is created in conjunction with Sirius Systems Corporation; Windows 95, Windows NT; and Windows XP are registered trademarks of Microsoft Corporation.

Acknowledgements:

Special Thanks to:

Andrew Bleau for his help revising this book.

In-House Solutions team for their recommendations and input.

Sincerely,
Mariana Lendel

3D GEOMETRY CREATION CONTENT

1. ABOUT STANDARD PLANES (TOP, FRONT, SIDE, 3D/ISOMETRIC) AND CONSTRUCTION DEPTH.....	2
2. USER-DEFINED CONSTRUCTION PLANES /GRAPHIC VIEWS USING GEOMETRY AND ROTATE	4
3. USER-DEFINED CONSTRUCTION PLANES / GRAPHIC VIEWS USING NORMAL	5
4. CREATE DRAWING # 1.....	6
5. CREATE DRAWING # 2.....	7
6. CREATE DRAWING # 3.....	8
7. CREATE DRAWING # 4.....	9
8. CREATE DRAWING # 5.....	10
9. CREATE DRAWING # 6.....	13
10. CREATING CURVES AT THE EDGES AND AT THE INTERSECTION OF SURFACES.	15
11. CREATE DRAWING #7.....	16
SURFACE GEOMETRY CREATION	20
GEOMETRICAL SURFACES; REVOLVED, DRAFT, EXTRUDED & PRIMITIVES	20
12. SURFACE DRAWING #1. CREATE REVOLVED SURFACE	20
13. SURFACE DRAWING #8. REVIEW - CREATE REVOLVED SURFACE.....	22
14. SURFACE DRAWING #11. CREATE DRAFT SURFACE & FLAT BOUNDARY SURFACE	23
15. EXERCISES USING CREATE DRAFT AND EXTRUDED SURFACE COMMAND.....	25
16. EXERCISES USING PRIMITIVES.....	27
FREE FORM SURFACES; RULED, LOFT, NET, SWARF & FENCE.....	29
17. SURFACE DRAWING #6. CREATE RULED AND LOFT SURFACES	29
18. CREATE RULED SURFACES WITHOUT SYNCHRONIZATION AND WITH SYNCHRONIZATION	32
19. CREATE RULED/LOFT SURFACES USING WINDOW SELECTION.	33
20. SURFACE DRAWING #3.CREATE NET SURFACES & FLAT BOUNDARY SURFACES.	34
21. CREATE NET SURFACES USING WINDOW SELECTION.....	36
22. SURFACE DRAWING #15. REVIEW EXERCISE CREATE A NET SURFACE.	39
23. SURFACE DRAWING #14. REVIEW EXERCISE CREATE A NET SURFACE.	40
24. CREATE SWEEP SURFACES.	41
25. CREATE SWEEP SURFACES USING TRANSLATE AND ROTATE OPTIONS	44
26. SURFACE DRAWING #18.CREATE SWEEP SURFACES.	46
27. CREATE SWEEP SURFACES WITH SHARP EDGES	47
28. CREATE SWEEP SURFACES, CURVES AT THE SURFACE EDGES AND A FLAT BOUNDARY SURFACE	47
29. CREATE FENCE SURFACE.	54
DERIVED SURFACES; OFFSET, TRIM, FILLET, 2 SURFACE BLEND & 3 SURFACE BLEND & FILLET BLEND	55
30. SURFACE DRAWING #4. CREATE OFFSET SURFACE.....	55
31.CREATE SURFACE TRIM TO CURVES.....	56
32.CREATE SURFACE TRIM TO SURFACE.....	59
33.CREATE SURFACE TRIM TO PLANE; SURFACE FILLET TO SURFACE.	62
34.CREATE SURFACE FILLET TO SURFACE & SURFACE FILLET TO PLANE.	67
35. REVIEW EXERCISE.....	70
36.CREATE 3 FILLET BLEND SURFACES.....	71
37.CREATE 2 SURFACE BLEND.....	75
38.CREATE 3 SURFACE BLEND.....	76
39. REVIEW EXERCISE.....	78
DRAWINGS	

SURFACE TOOLPATH CREATION CONTENT

1. SURFACE ROUGH POCKET ON A CAVITY SHAPE.....	80
2. COMPARE THE SURFACE ROUGH POCKET WITH SURFACE ROUGH FLOWLINE.....	86
3. FINISH THE SURFACE USING SURFACE FINISH FLOWLINE.....	92
4. COMPARE THE SURFACE FINISH FLOWLINE WITH FINISH RADIAL.....	94
5. COMPARE THE FINISH FLOWLINE & FINISH RADIAL WITH SURFACE FINISH BLEND.....	97
6. SURFACE ROUGH POCKET ON A BOSS SHAPE	99
7. COMPARE SURFACE ROUGH POCKET WITH SURFACE ROUGH PARALLEL	103
8. COMPARE ROUGH POCKET & ROUGH PARALLEL & ROUGH CONTOURS	107
9. MACHINING THE BOSS SURFACE USING SURFACE FINISH SCALLOP	111
10. MACHINING THE FLAT SURFACE USING SURFACE ROUGH POCKET WITH FACING OPTION.....	115
11. COMPARE THE TOOLPATHS TO AN STL FILE	116
12. SURFACE FINISH LEFTOVER TOOLPATH	117
13. SURFACE ROUGH POCKET REVIEW	119
14. MACHINE THE LEAF USING SURFACE FINISH PENCIL WITH UNLIMITED PASSES.	121
15. REVIEW - MACHINING THE FLAT SURFACES	123
16. REVIEW - COMPARE THE TOOLPATHS TO AN STL FILE.....	124
17. REVIEW - SURFACE FINISH LEFTOVER TOOLPATH.....	125
18. CLEANING THE SHARP EDGES USING SURFACE FINISH PENCIL WITH LIMITED PASSES.....	127
19. SURFACE ROUGH PLUNGE TOOLPATH.....	128
20. REVIEW - SURFACE FINISH BLEND TOOLPATH WITH TRUE SPIRAL CUTTING METHOD.....	133
21. REVIEW - MACHINING THE FLAT SURFACES	135
22. REVIEW - SURFACE FINISH LEFTOVER TOOLPATH.....	136
23. SURFACE FINISH PROJECT TOOLPATH TO ENGRAVE THE LETTERS	138
24. SURFACE ROUGH POCKET REVIEW; SOLID SELECTION.	139
25. MACHINE THE REMAINIG AREAS USING SURFACE ROUGH RESTMILL.....	142
26. SURFACE FINISH CONTOUR.....	144
27. SURFACE FINISH SHALLOW.	145
28. SURFACE ROUGH RESTMILL FROM AN STL FILE.....	148
29. DEFINE A NEW PLANE AND MACHINE THE BOTTOM OF THE PART.....	151
30. HIGH SPEED CORE ROUGHING	154
31. HST HORIZONTAL FINISHING –THE FLAT SURFACE ONLY.	159
32. HST SCALLOP FINISHING –THE BOSS SHAPE ONLY.....	162
33. HST SCALLOP (REST PASSES) FINISHING –CREATING A FILLET.	164
34. SURFACE HIGH SPEED - CORE ROUGHING	166
35. HST HORIZONTAL FINISHING –THE FLAT SURFACES ONLY.	169
36. HST WATERLINE FINISHING.....	171
37.2 SURFACE HIGH SPEED - AREA CLEARANCE ROUGHING	173
38. HST SCALLOP FINISHING.	176
39. HST SPIRAL FINISHING.....	178
40. HST HORIZONTAL FINISHING –THE FLAT SURFACE ONLY.	180

HIGH SPEED SURFACE TOOLPATH CREATION

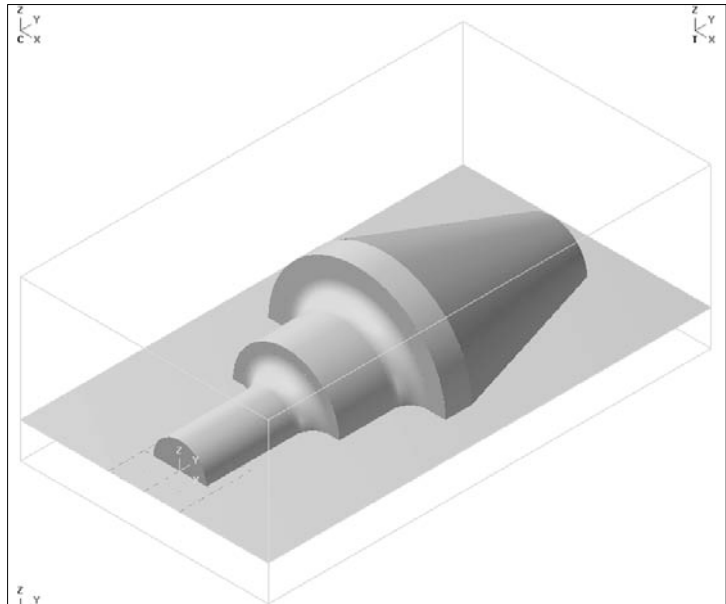
30. HIGH SPEED CORE ROUGHING

- ◆ Surface high speed toolpaths are a set of machining strategies that are specially designed to produce the smoothest, most efficient tool motions when machining surface models or solid faces.

Resources – Download the file from http://www.emastercam.com/files/x4_ptg-milllevel3.html

File

- **Open**
- Select “Hst-Core_Roughing.mcx”



30.1 Properties

- ◆ **Tool Settings** allows you to establish the material type of the part that will be cut. According to the material type and to the diameter of the tool Mastercam can calculate the feedrates and spindle speeds for each operation.
- Set the Program # 7
- Enable **Assign tool numbers sequentially** and all the **Advanced options**.
- Enable **Feed Calculation From tool**
- ◆ **Stock Setup** allows you to establish the stock size that will be used in Verify to simulate the machining process.
- Use Bounding box.

30.2 Surface High Speed- Rough Core

- ◆ **Core roughing toolpaths** are designed for machining models which can be approached from the outside. They minimize the need for helical ramp moves or full-width cutting. Core roughing toolpaths are generated from a set of surface profiles that describe the shape of your surfaces at different Z heights, plus a set of offset profiles that let you rough out stock as you approach the part from the outside. This toolpath can change the machining strategy within the same operation if your part has a mixture of bosses and cavities. In these cases, Mastercam will cut the cavities inside to out (like an area clearance cutting pass), and machine the bosses from the outside.

Toolpaths**➤ Surface High Speed...**

- Select all surfaces

Toolpath Type

- ✦ The **Toolpath Type** dialog box allows you to select the toolpath between rough and finish operations. It also allows you to change the drive/check surfaces that you selected and the
- ✦ Containment and the Approximate start point.

- Select **Roughing** and **Core Roughing**

Tool

- ✦ Allows you to select a tool, edit its properties, and enter feeds and speeds. You can insert a comment that will be output in the NC file after running the post processor.

- Select the 0.50" Bull Nose with the corner radius of 0.0625.

Holder

- ✦ Allows you to select the tool holder to be used in the operation. You can also; create a holder definition, load a holder from a library or edit the holder after it has been selected. You can check the holder for gouge.

- Select **Open library** and from CAT 40 select C4C4-0016

Cut parameters

- ✦ The Cut parameters dialog box allows you to establish the **Stepdown** for the cuts in Z; **XY stepover** to set the passes in XY plane; **Smoothing** to round the corners of the toolpath to maintain a higher feedrate. You can also establish the **Stock to leave** on **walls** and on **floors**. You can set how to control the tool's position around the boundary of the part.

Stepdown

- Set the Stepdown to **0.05** and enable Add cuts with a Min stepdown = **0.05** and Max profile stepover = **0.25**;

- ✦ Use Add cuts feature to insert additional cutting passes in the shallow areas.

XY stepover

- Set the % of dia = 50 this will automatically update the Min and Max distances

- ✦ If you are using toolpath smoothing, make sure that the minimum stepover is greater than the Offset Tolerance value and less than the radius of the tool shaft. The maximum stepover should be less than twice the minimum stepover.

Smoothing

- Set Max radius= 0.025

- **Max radius** limits the size of the arcs Mastercam will create to round the corners

- Profile tolerance = 0.005

Tool containment

- Enable **Center** in the **Tool containment**

- Tool containment allows you to select the closed contour inside of which the tool will machine the part.

- Stock to leave on walls = 0.05

- Stock to leave on floor = 0.05

- Note that the stock to leave on walls must be greater than or equal to the stock left on the floor. The only exception is for horizontal area finish passes.

Trochoidal motion leave it off

- **Minimize burial** enabled will have Mastercam automatically insert trochoidal loops in the toolpath in areas where the tool might be fully buried.

Entry helix

- Set **Radius** = 0.175
- Enable **Output 3D arc moves**
- **Steep/shallow** not enable

Linking Parameters

- Creates the links between the cutting passes. In general, you can think of linking moves as air moves when the tool is not in contact with the part

Retract method

- Enable **Minimum distance** to insert high-speed arcs to and from the retract height and maintains a minimum retract height above the surface for the fastest transitions.

Retracts

- Enter the size of the arc in the **Curl up** (arc to the retract height) = 0.15
- Enter the size of the arc **Curl down** (arc from the retract height) = 0.15.
- Part clearance = 0.15

Leads

- ✦ Controls how the tool moves onto and off of the part at the start and end of each cutting pass. These moves are applied to each pass no matter which cutting pass is selected.
- Linear entry/exit = 0.025
- Vertical arc entry = 0.075
- Vertical arc exit = 0.075

Fitting

- ✦ Controls how the entry and exit arcs will be fit into each pass.
- Set to **Minimize Trimming**
- ✦ The path of the retract will be as close to the surface as possible, maintaining a minimum distance from the surface to fit the arc.
- Set the **Max trimming distance** = 0.0825
- Select the **OK** button to exit

Smoothing

- ✦ **Profile tolerance** determines the maximum deviation between the smoothed and unsmoothed toolpaths on the outermost profile or cutting pass.
- Offset tolerance = 0.005625
- ✦ **Offset tolerance** is define similar with the Profile tolerance, but it is applied to all the inner passes.

Arc Filter/Tolerance

- ✦ **Total tolerance** displays the sum of the filter tolerance and cut tolerance. The dialog box that opens when you choose the **Total Tolerance** button depends on whether the 3D Advanced Toolpath Refinement feature is active for the Mastercam session.
- Total tolerance = 0.002
- ✦ **Refine Toolpath** is used to refine mastercam's surface and high speed toolpaths, reducing machining time and improving machined surface quality.
- ✦ To automatically define the total tolerance allocations and other settings, use the "wizard" slider controls in the "My preferences are" Section



- ✦ The **Tolerances Distribution** fields display the total tolerance you defined for the toolpath and the formula used. Use the sliders between the fields to allocate fixed percentages of cut, line/arc filtering, and smoothing tolerances in 5% increments. Or, enter the percentages directly into the fields.

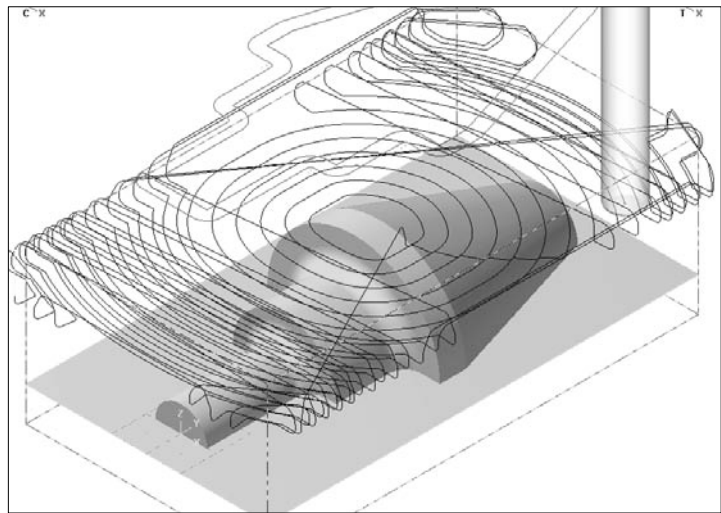
Tolerances Distribution

Cut tolerance 0.0002 10.0%	+	Line/Arc Filter tolerance 0.0004 20.0%	+	Smoothing tolerance 0.0014 70.0%	=	Total tolerance 0.002 100%
----------------------------------	---	--	---	--	---	----------------------------------


- ✦ If desired, use the **Line/Arc Filtering Settings** and **Smoothing Settings** to further refine the toolpath.

30.3 Backplot the toolpath.

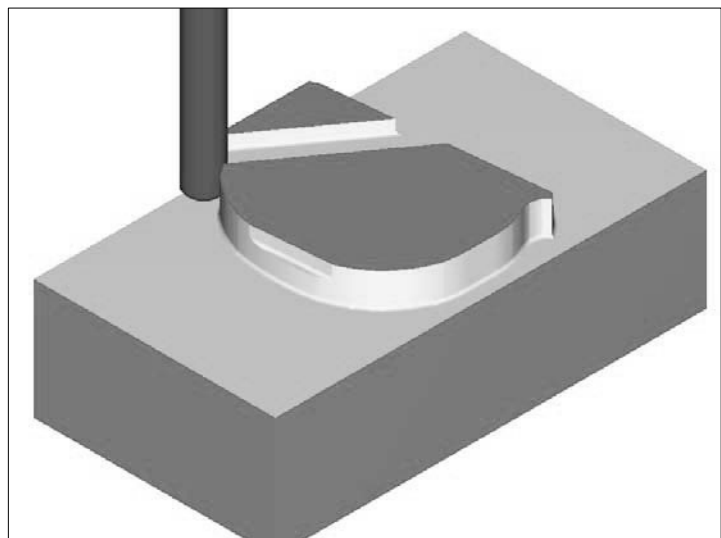
- ✦ Select **Play** 



30.4 Verify the toolpath.

- ✦ Select the **Machine** button to run Verify. 

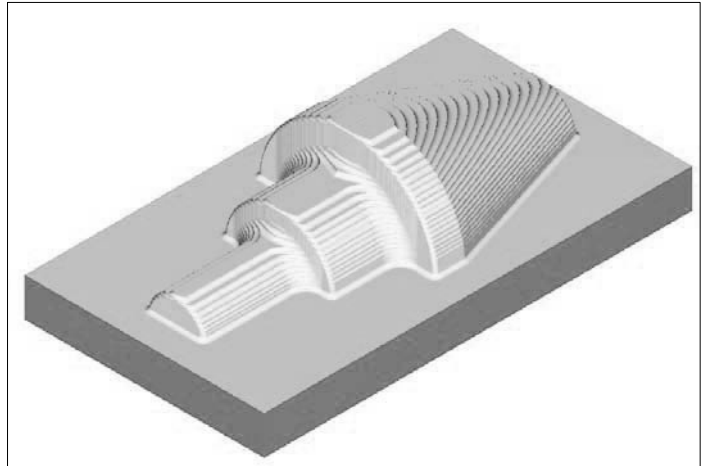
- ✦ Note that the first pass is too big. We set the depth cuts to 0.05.



30.5 Use the Steep/Shallow parameters to control the depth cuts from the stock.

- Enable **Use Z depths** and set the **Minimum depth** = 1.7 and **Maximum depth** = 0.0

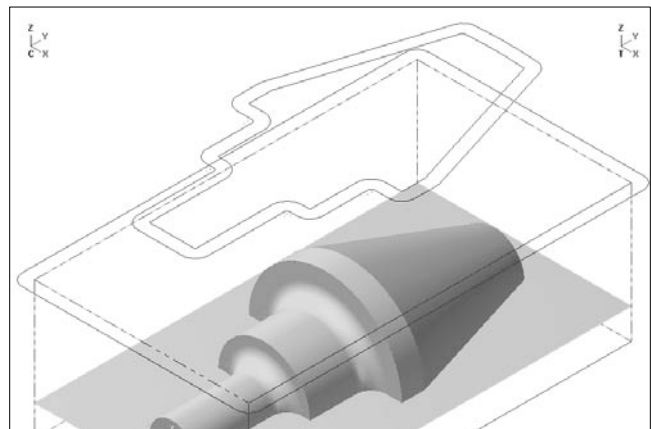
Regenerate, then Backplot and Verify the toolpath



31. HST HORIZONTAL FINISHING –THE FLAT SURFACE ONLY

31.1 Use the SilhouetteBoundary C-Hook to create a boundary around the shape.

- **Tips:** Set Z depth to 3 and make sure the construction plane is **Top**.
- Select all surfaces and unselect the bottom.
- Use **Xform offset contour** and offset outside, both the top rectangle (from the stock) and the boundary, with a 0.2 offset distance.



31.2 Surface High Speed Horizontal Area

- ✦ **High Speed Horizontal Area** toolpaths are used to machine the flat areas of the surface model. Mastercam analyzes the selected drive surfaces and automatically identifies the flat areas within each surface.
- Copy the operation in the **Operations Manger**
- Select the **Geometry** in the second Operation and add the two offset chains as containment boundaries.
- ✦ Note that the high speed toolpaths allows us to use as boundaries nested chains.
- Select the **Parameters** of the second Operation.

Toolpath Type

- Select **Finishing** and **Horizontal Area**

Tool

- Select the 0.375" Flat endmill.

Holder

- Select **Open library** and from CAT 40 select C4C4-0016
- Enable **Use holder for gouge checking**.

Cut parameters

- ✦ The **Cut parameters** dialog box allows you to establish the **Depth cuts**; the **XY Stepover** for the spacing between cutting passes; **Smoothing** to round the corners of the toolpath to maintain a higher feedrate. You can also establish the **Stock to leave** on **walls** and on **floors**. You can set how to control the tool's position around the boundary of the part.
- Set the **# of depth cuts** to 1.
- **XY Stepover**
 - Set the %of dia = 45.0
- **Smoothing**
 - Set Max radius= 0.01875
 - Tool containment set to **Inside** and **Offset distance** = 0.0 and enable **Add offset distance to tool radius**.
 - Make the Stock on walls/floors =0.

Trochoidal motion leave it to off.

- ✦ Select Trochoidal motion to have Mastercam automatically insert trochoidal loops in the toolpath in **areas where the tool might be fully buried**.

Transition

Mill Level 3

- Set the transition to **Entry helix** to insert arcs at the beginning and end of the ramp for the smoothest tool motion into and out of the move.
- Make the radius = 0.2

Linking Parameters

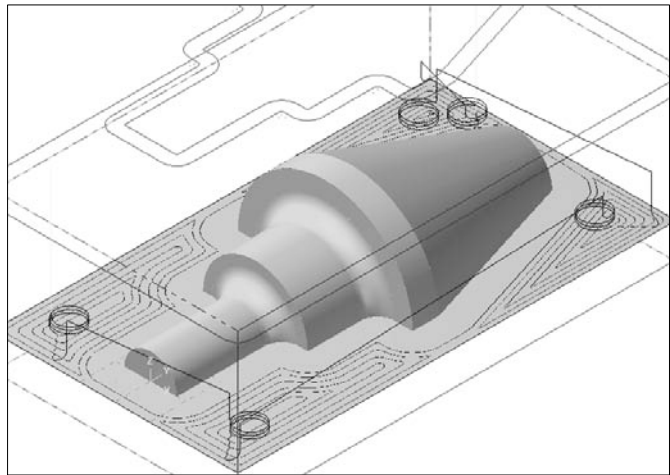
- Set the **Retract method** to **Minimum Vertical Retract**.
- Set the **Part clearance** to 0.075
- Linear entry/exit = 0.025
- Vertical arc entry = 0.075
- Vertical arc exit = 0.075
- Horizontal arc exit = 0.075
- Max ramp angle = 10
- Select the **OK** button to exit.

Arc Filter/Tolerance


- **Smoothing**
- Profile tolerance = 0.00375
- Offset tolerance = 0.0375

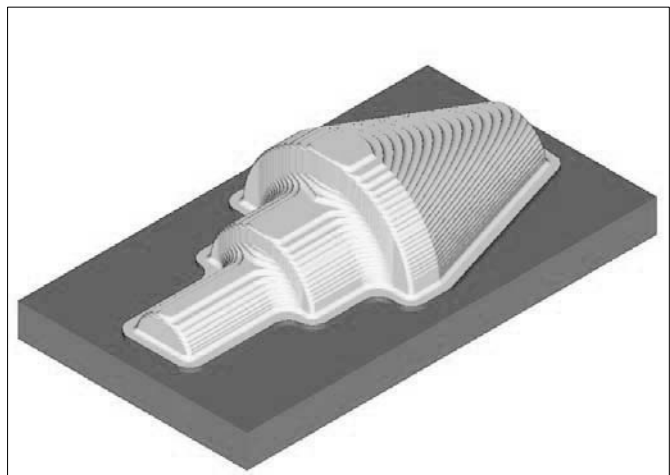
31.3 Backplot the toolpath.

- Select **Play** 



31.4 Verify the two toolpath.

- Select the **Machine** button to run **Verify**.
- 



32. HST SCALLOP FINISHING –THE BOSS SHAPE ONLY

- ✦ **High speed scallop** toolpaths differ from other finish toolpaths in that the stepover distance is a 3D value which is measured along the surface, instead of parallel to the tool plane. This ensures a consistent scallop height across the surface, regardless of the surface direction.

- Copy the operation in the Operations Manger
- Select the **Geometry** in the third Operation and rechain all and add the chain created with the C-Hook as the containment boundary.
- Select the **Parameters** of the third Operation.

Toolpath Type

- Select **Finishing** and **Scallop**

Tool

- Select the 0.50" Ball end.

Holder

- Select **Open library** and from CAT 40 select C4C4-0016
- Enable **Use holder for gouge checking**.

Cut parameters

- ✦ The **Cut parameters** dialog box allows you to establish the **Cutting method**; the **Stepover** for the spacing between cutting passes. You can also establish the **Stock to leave** on **walls** and on **floors**. You can set how to control the tool's position around the boundary of the part.

Cutting method

- Set the Cutting method to **One Way** to machine all the passes in a single default direction. Mastercam tries to maintain a **climb milling** orientation relative to the surface boundaries.
- Make sure that **Expand inside to cut** is not enable.

Stepover

- Set the stepover = 0.050
- Tool containment set to **Outside** and **Offset distance** = 0.3.
- Make the **Stock on walls/floors** =0.

Transition

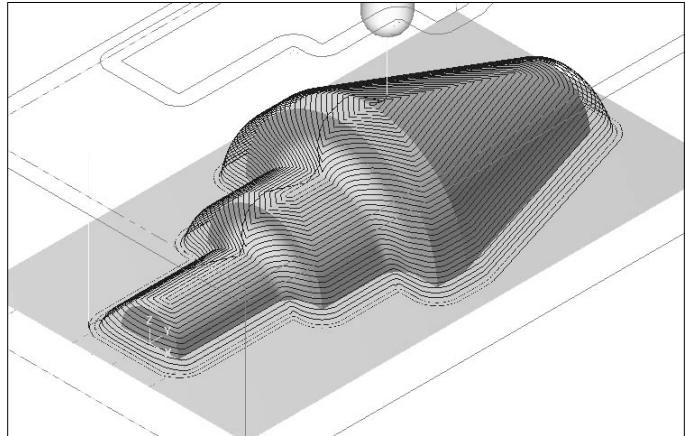
- Set the transition to **Tangential ramp** to insert arcs at the beginning and end of the ramp for the smoothest tool motion into and out of the move.


Linking Parameters

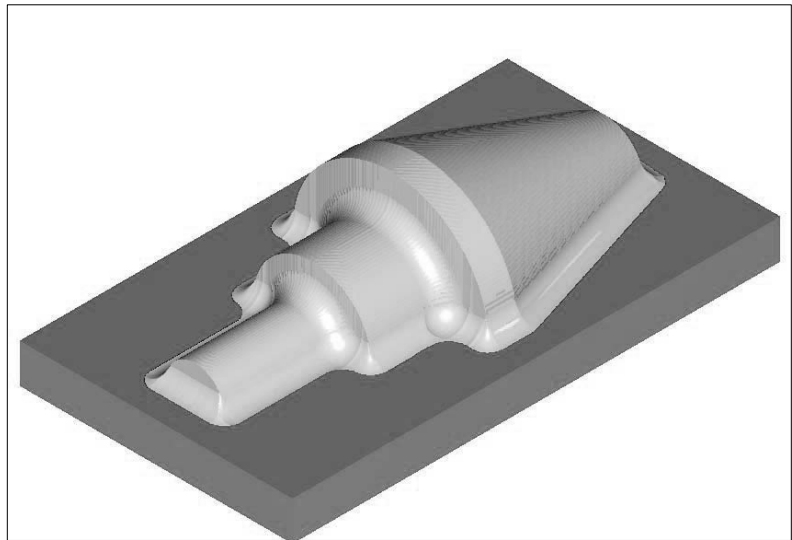
- Leave the parameters as set for the **Horizontal Area** cut.

Mill Level 3**32.1 Backplot the toolpath.**

➤ Select **Play** 

**32.2 Verify the toolpaths.**

➤ Select the **Machine** button to run Verify. 



33. HST SCALLOP (REST PASSES) FINISHING –CREATING A FILLET

- ✦ **High speed scallop rest passes** toolpaths calculate a stock model based on the dimensions of a roughing tool and only apply the toolpath to those areas. Use them for applications where you want a different cutting strategy than area clearance passes, and the remaining stock is light enough to be machined with a single cut.

- Copy the operation in the Operations Manger
- Select the **Geometry** in the fourth Operation and rechain all and add the chain created with the C-Hook as the containment boundary.
- Select the **Parameters** of the fourth Operation.

Toolpath Type

- Leave the **Finishing** and **Scallop**

Tool

- Select the 0.1250" Ball end.

Cut parameters

- ✦ The **Cut parameters** dialog box allows you to establish the **Cutting method**; the **Stepover** for the spacing between cutting passes. You can also establish the **Stock to leave** on **walls** and on **floors**. You can set how to control the tool's position around the boundary of the part.

Cutting method

- Set the Cutting method to **One Way** to machine all the passes in a single default direction. Mastercam tries to maintain a **climb milling** orientation relative to the surface boundaries.
- Make sure that **Expand inside to cut** is not enable.

Stepover

- Set the stepover = 0.0125
- Tool containment set to **Outside** and **Offset distance** = 0.3.
- Make the **Stock on walls/floors** =0.

Rest material

- Enable **Use rest material**
- Enable **Roughing Tool** and the **Diameter** = 0.5 and **Corner radius** = 0.25

Transition

- Set the transition to **Tangential ramp** to insert arcs at the beginning and end of the ramp for the smoothest tool motion into and out of the move.

Linking Parameters

- Leave the parameters as set for the **Horizontal Area** cut.

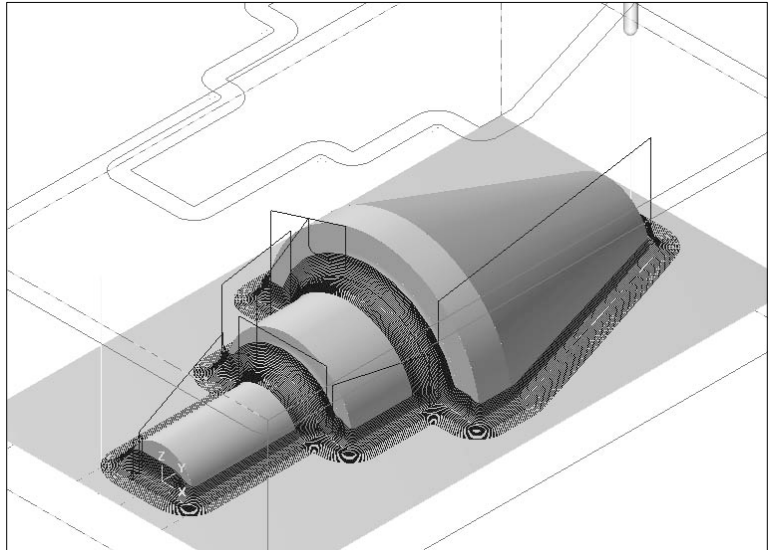
Mill Level 3**Arc Filter/Tolerance**

- Enable **Toolpath fillet** and set the **Radius** = 0.1875

- Toolpath fillet option tells Mastercam to insert an arc to the specified radius in the toolpath at sharp corners. The radius value that you enter here should be at least as large as the radius of the finish tool.
Note: The fillets are created as tool motions only. They are not saved as part of your surface model, and they have no effect on your part geometry.

32.1 Backplot the toolpath.

- Select **Play** 

**32.2 Verify the toolpaths.**

- Select the **Machine** button to run **Verify**. 