

Mastercam® 2020

IMPERIAL | PROFESSIONAL COURSEWARE

HLE
DOWNLOAD
INSTRUCTIONS INSIDE



2020



MILL ESSENTIALS

Mastercam® 2020

MILL ESSENTIALS PROFESSIONAL COURSEWARE

To order more books:

Call 1-800-529-5517 or

Visit www.emastercam.com or

Contact your Mastercam dealer



Mastercam
CAD/CAM Systems

**Home Learning Edition
Demo Software
DOWNLOAD**

HOW TO DOWNLOAD:
Mastercam Home Learning Edition (HLE)
Demo Software

Visit www.eMastercam.com/files

Click on Mastercam Demo Software
from the categories on the right.

Proceed to download!

Mastercam 2020 Mill Essentials Professional Courseware

Copyright: 1998 - 2020 In-House Solutions Inc. All rights reserved

Software: Mastercam 2020

Date: August 23, 2019

ISBN: 978-1-77146-858-9

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, of 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 International copyright laws. All rights are reserved. This document may not, in whole or part, be copied, photographed, reproduced, translated or reduced to any electronic medium or machine readable form without prior consent, in writing, from In-House Solutions Inc.

Trademarks

Mastercam is a registered trademark of CNC Software, Inc.

Microsoft, the Microsoft logo, MS, and MS-DOS are registered trademarks of Microsoft Corporation;

N-See is a registered trademark of Microcompatibles, Inc.; Windows 7 and Windows 8 are registered trademarks of Microsoft Corporation.

Table Of Contents

Getting Started	13
Introduction “What Is “Mastercam“?”	14
The Mastercam® User Interface:	15
Setting The Grid	16
Home Tab Attributes And Organize Groups And The Mini Toolbar	16
Data Entry Shortcuts	19
The Right Mouse Click Menu	20
AutoCursor	21
Introduction To The Graphic Views, Planes & The Coordinate Systems	22
Mastercam® Parameter Dialog Boxes:	23
Mastercam® Work Flow:	24
Conventions Used In This Book:	28
Geometry Tools - Part 1	29
Introduction:	30
Instructor Demonstration Preview:	31
Step 1: Explore Mastercam Interface	32
Step 2: Create Point Position Command	35
Step 3: Create Line Endpoints & Parallel Commands	36
Step 4: Create Rectangle Commands	39
Step 5: Create Fillet Commands	41
Step 6: Trim Commands	43
Summary:	45
Exercise - Create Drawing #1	46
Optional Exercise - Create Drawing #2	51
Contour Toolpaths Part 1	53

Introduction	54
Drawing #3	55
Contour Toolpaths Information:	56
Instructor Demonstration Preview:	57
Step 1: Create And Locate Part Geometry	58
Step 2: Create The Job Setup	60
Step 3: Toolpath Selection	63
Step 4: Set The 2D Toolpath Parameters	66
Step 5: Verify The Toolpath Using Backplot	72
Step 6: Simulate The Toolpath In Verify	73
Step 7: Editing The Toolpath Parameters	76
Step 8: Post Process The Toolpaths	77
Step 9: Save Your Mastercam File	78
Summary:	78
Exercise #1	79
Exercise #2	80
Geometry Tools - Part 2	81
Introduction:	82
Instructor Demonstration Preview:	83
Step 1: Create Circles And Arcs Command	84
Step 2: Create Tangent Arcs	86
Step 3: Create Tangent Lines	88
Step 4: Create Chamfer Commands	89
Step 5: Break	92
Step 6: Duplicates	94
Step 7: Create Letters	96
Step 8: Other Geometry Creation Tools In The Wireframe TAB.	97
Summary:	97

Exercise #1	98
Step 9: Suggested Steps	98
Exercise #2- Drawing #4	102
Exercise #3- Drawing #10	103
Contour Toolpaths Part 2	105
Introduction	106
Instructor Demonstration	107
Part Drawing	108
2D Geometry Selection Reference Information:	109
Step 1: Create And Locate Part Geometry	109
Step 2: Create The Job Setup	111
Step 3: Facing Toolpath	113
Step 4: Set The Facing Parameters	113
Step 5: Machine The Open Contour With Multiple Passes	119
Step 6: Machine The Four Corners Using Contour Toolpath	125
Step 7: Contour The Outside Profile	130
Step 8: Chamfer The Top Of The Part	133
Step 9: Create Letters	136
Step 10: Engrave The Letters Using Contour Toolpath	138
Step 11: Post Process The Toolpaths To Generate The G-Code	141
Step 12: Save The Mastercam File	142
Summary:	143
Exercise #1	144
Exercise #2	145
Geometry Manipulation	147
Introduction:	148
Instructor Demonstration Preview:	149

Step 1: Transform Mirror Command	150
Step 2: Transform Translate Command	152
Step 3: Transform Offset Commands	154
Step 4: Transform Rotate Commands	156
Step 5: Analyze Commands	157
Step 6: Managing Attributes & Levels	159
Attributes	160
Step 7: Levels	161
Step 8: Chaining - Wireframe Mode	168
Step 9: Analyze Chains And Fix Their Problems	176
Step 10: Import A DWG File	180
Step 11: Transform Move To Origin	181
Summary:	181
Exercise #1 - Drawing #8	182
Exercise #2 - Drawing #15	183
Exercise #3	184
Step 12: Exercise 1 Suggested Steps	184
Pocket Toolpaths	191
Introduction	192
Instructor Demonstration	193
Part Drawing	194
Pocket Toolpaths Information	195
Pocket Toolpaths - Basic Steps Reference	196
Step 1: SAVE Job Setup For Future Jobs	196
Step 2: Create And Locate Part Geometry	200
Step 3: Set The Job Setup	202
Step 4: Facing Toolpath	203
Step 5: Machine The Closed Pocket	207

Step 6: Open Pocket Toolpath	213
Step 7: Remachine The Pockets	220
Step 8: Post Process The Toolpaths To Generate The G-Code	226
Step 9: Save The File	227
Summary:	227
Exercise #1	228
Exercise #2	229
Solids Geometry	231
Introduction:	232
Instructor Demonstration Preview:	233
Solid Terms	234
Step 1: Extrude Create Body Command	235
Step 2: Extrude Add Boss	236
Step 3: Solids Manager	237
(Optional) Create The Solid To Geometry1_Exercise2	240
Step 4: Extrude Cut Body Command	241
Step 5: Solid Hole	243
Step 6: Solid Fillet	247
Step 7: Solid Chamfer	249
Summary:	252
Exercise #1 -Drawing #8	253
Exercise #2 -Drawing #15	254
Exercise #3 -Drawing #10	255
Align To Plane & Align To Face	257
Introduction:	258
Instructor Demonstration Preview:	259
Step 1: Open The File	260

Step 2: Save The File	260
Step 3: Removing The Solid History	261
Step 4: Create A New Plane	263
Step 5: Align To Plane	265
Step 6: Open The File	269
Step 7: Save The File	269
Step 8: Align To Face Parallel	270
Step 9: Align To Face Coincident	275
Step 10: Merge A File	275
Drilling Toolpaths	279
Introduction	280
Instructor Demonstration:	281
Point/Circle Toolpaths Information	282
Step 1: Prepare And Orient Model For Machining	283
Step 2: Create The Job Setup	286
Step 3: Dynamic Mill Toolpath To Machine The Outside Shape	288
Step 4: Circle Mill The Large Center Hole	296
Step 5: Spot Drill, Drill And Tap The Four 1/2"-20 UNC Holes	302
Step 6: Drill The Holes	306
Step 7: Tapping The Four Holes	310
Step 8: Spot Drill The 3/8" Holes	314
Step 9: Drill 3/8" Holes	318
Step 10: Post Process The Toolpaths To Generate The G-Code	322
Step 11: Save Your File	322
Summary:	323
Exercise #1	324
Incremental Versus Absolute	325
Introduction	326

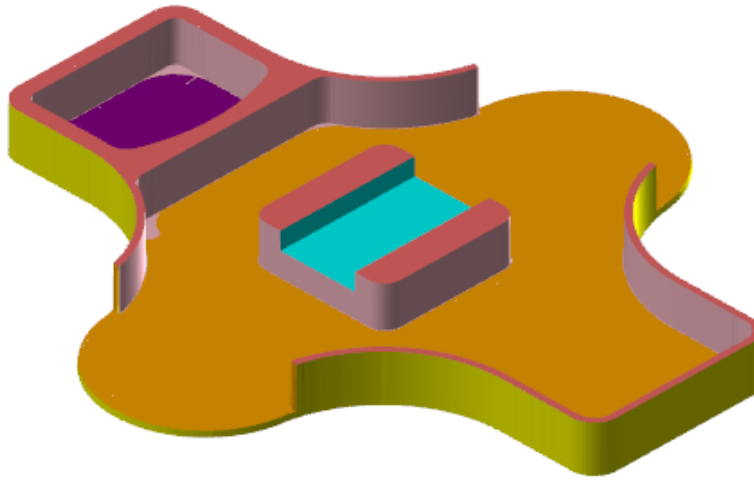
Instructor Demonstration	327
Step 1: Create The Job Setup	328
Step 2: Open Contour With Incremental Depth	330
Step 3: (Optional) Peel Mill Toolpath	338
Step 4: Machine The Slots Using Contour Ramp Toolpath	347
Step 5: Post Process The Toolpaths To Generate The G-Code	351
Step 6: Save The Mastercam File	351
Summary:	352
Exercise #1	352
2D HST Toolpaths - Part 1	353
Introduction	354
Instructor Demonstration:	355
Part Drawing	356
2D High Speed Toolpaths (HST) Information	356
Step 1: Prepare And Orient Model For Machining	357
Step 2: Create The Job Setup	357
Step 3: Dynamic Mill Toolpath To Machine The Outside Shape	360
Step 4: Dynamic Mill To Remove The Inside Material	368
Step 5: Dynamic Mill The Closed Pocket	375
Step 6: Peel Mill To Machine The Inside Center Boss	379
Step 7: Dynamic Rest Mill The Inside Corners	385
Step 8: Finish The Walls Using Contour	391
Step 9: Post Process The Toolpaths To Generate The G-Code	396
Step 10: Save Your File	397
Summary:	398
Exercise #1	399
Exercise #2	400

Multiple Setups	401
Introduction	402
Instructor Demonstration:	403
Step 1: Prepare And Orient The Model	404
Step 2: Create The Job Setup	407
Step 3: Dynamic Mill The Closed Pocket With The Island	409
Step 4: Machine The Outside Profile Using Dynamic Mill	415
Step 5: Remachine The Pocket Using Dynamic Rest Mill	420
Step 6: Finish The Pocket Floor And Walls Using Pocket Toolpath	423
Step 7: Dynamic Contour	429
Step 8: Spot Drill The Holes	434
Step 9: Drill 1/4" Holes	437
Step 10: Circle Mill The 1/2" Counterbore Holes	441
Step 11: Create A Stock Model	447
Step 12: Create A New Toolpath Group	449
Step 13: Change The Machining Plane To A New Face	449
Step 14: Dynamic Machine The Top With The Boss	453
Step 15: Dynamic Machine The Top Of The Boss	458
Step 16: Finish The Boss Walls Using Contour	463
Step 17: Chamfer The Holes Using Spot Drill	467
Step 18: Change The NC File Names	472
Step 19: Post Process The Toolpaths To Generate The G-Code	474
Step 20: Save Your File	475
Summary:	475
Exercise #1	476
2D HST Toolpaths - Part 2	477
Introduction	478
Instructor Demonstration:	479

Step 1: Prepare And Orient Model For Machining	480
Step 2: Create The Job Setup	483
Step 3: Face The Top Of The Part	485
Step 4: Set The Facing Parameters	485
Step 5: Dynamic Mill Machine Two Closed Pockets And Face The Island	488
Step 6: Create Edge Curves	494
Step 7: Dynamic Mill To Machine The Larger Open Pockets	496
Step 8: Dynamic Mill To Machine The Smaller Open Pockets	503
Step 9: Contour Toolpath To Finish The Inside Pocket Walls	510
Step 10: Contour Toolpath To Finish The Open Pocket Walls	515
Step 11: Circle Mill To Machine The Large Holes	520
Step 12: Spot Drill, Drill And Tap The Four 1/4"-20 UNC Holes	526
Step 13: Spot Drill And Drill The 3/8 " Holes	537
Step 14: Chamfer The Sharp Edges Using Model Chamfer	546
Step 15: Post The File	551
Step 16: Save Your File	551
Summary:	551
FBM Drill	553
Introduction	554
Instructor Demonstration:	555
Part Drawing	556
FBM Drill Toolpaths Information	557
Step 1: Open The Parasolid File	558
Step 2: Create The Job Setup	558
Step 3: Feature Based Machining Drill	560
Step 4: Post Process The Toolpaths To Generate The G-Code	573
Step 5: Save Your File.	574

Summary:	574
Lesson 14 - Exercise #1	575
Lesson 14 - Exercise #2	576
Multiple Setups - Part 2	577
Introduction	578
Instructor Demonstration:	579
Part Drawing	580
Step 1: Prepare And Orient The Model	581
Step 2: Create The Job Setup	581
Step 3: Contour The Full Radius Slot	583
Step 4: Use Toolpath Transform To Finish The Slots	587
Step 5: Machine The Pocket Using 2D HS Area Mill	589
Step 6: Export The Toolpath For Later Use	594
Step 7: Change The Machining Plane To A New Face	595
Step 8: Import The Operation To Front Plane	598
Step 9: Create The Letters	603
Step 10: Engrave The Letters Using Contour Toolpath	604
Step 11: Change The NC File Names	608
Step 12: Post Process The Toolpaths To Generate The G-Code	609
Step 13: Save Your File	610
Summary:	610
Exercise #1	611
2D Skills Test	613
Introduction	614
Skills Test #1	615
Skills Test #2	616
Appendix - Drawings	617
Quick Reference Card	649

2D HST Toolpaths - Part 1

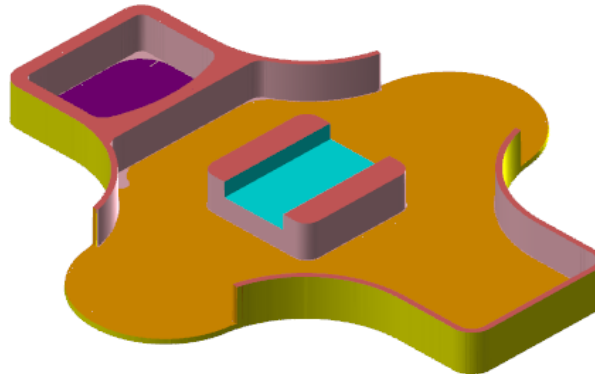


INTRODUCTION

Mastercam 2D high speed toolpaths are specially designed to produce the smoothest, most efficient tool motions, optimized for high speed and hard milling. These high speed toolpaths can help make the processes we have learned so far more efficient and automated, minimizing programming and cycle times.

Dynamic Mill Toolpath machines pockets, material that other toolpaths left behind, and standing bosses or cores using the entire flute length. The toolpath supports many powerful entry methods, including a custom entry method. Entry methods and micro lifts support custom feeds and speeds to optimize and generate safe tool motion.

The toolpath depends on the Machining strategy that you choose in the **Chain Options**. If the strategy chosen is **From outside**, the toolpath starts at the outmost chain and works its way in taking on the final shape of the part as it approaches the final pass. You can also machine pockets in which case the strategy selected is **Stay inside** which keeps the tool inside of the machining regions. The main difference between the **Dynamic Mill** and **Area Mill** is that the cutting method of the first one allows you to use the entire flute of the tool while with **Area Mill** small depth of cuts are recommended.



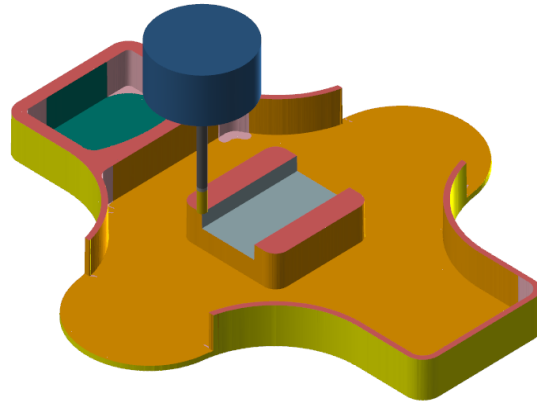
NEW CONCEPTS COVERED IN THIS TUTORIAL:

- ◆ Using imported solid model.
- ◆ 2D HST Geometry / boundary selection rules.
- ◆ Setting the 2D HST cut parameters.
- ◆ Dynamic Milling toolpaths.
- ◆ Area Milling toolpaths.
- ◆ Dynamic Rest Mill toolpaths.

INSTRUCTOR DEMONSTRATION:

Topics:

- ◆ Import a Parasolid with saved Coordinate system
- ◆ Job Setup
- ◆ Dynamic Mill From Outside Toolpath
- ◆ Dynamic Mill From Inside Toolpath
- ◆ Rest Mill Toolpaths
- ◆ Peel Mill Toolpath



NOTES:

PART DRAWING

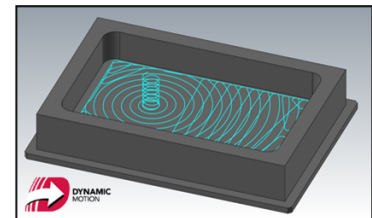
2D HIGH SPEED TOOLPATHS (HST) INFORMATION

2D High Speed Dynamic toolpaths utilize the entire flute length of their cutting tools to achieve efficiency in milling. They are designed to maximize material removal while minimizing tool wear.

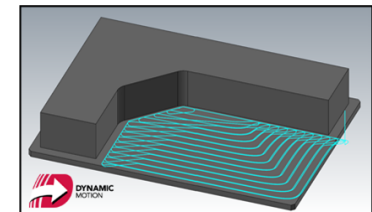
The dynamic mill toolpaths are all designed to simplify the programming of complex pocket and core shapes. Benefits include:

- ◆ Tool burial avoidance.
- ◆ Minimum heat buildup.
- ◆ Better chip evacuation.

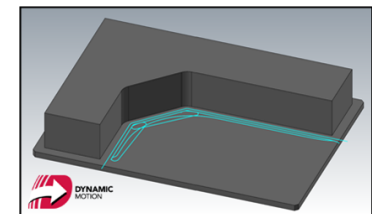
Dynamic Mill - Stay Inside Strategy machines pockets using one or more chains to drive the toolpath.



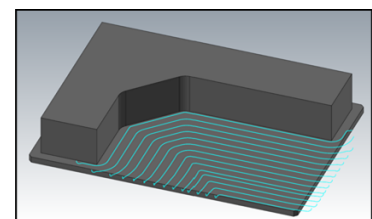
Dynamic Mill - From Outside Strategy machines open pockets or standing core shapes using the outmost chain as the stock boundary. The toolpath starts from the outside and works its way towards the inner boundary.



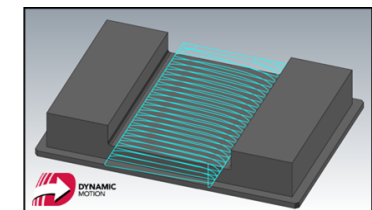
Dynamic Contour - Efficient milling of material off walls. Supports both closed or open chains.



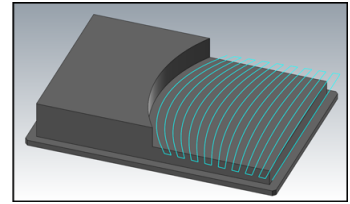
2D High Speed Area Mill toolpaths machine pockets using a smooth clean motion. Helical entries and tangent stepovers create efficient motion for tools. Cut parameters allow controlling smoothing to avoid sharp corners or direction changes.



2D HST Peel Mill toolpath generates an efficient climb mill cut between two selected contours or along a single contour. It uses a dynamic style motion with accelerated “back” feed moves when the tool is not engaged in material. For single chains, the user defines the width of cut, otherwise the width is defined by the area between the 2 boundary chains.



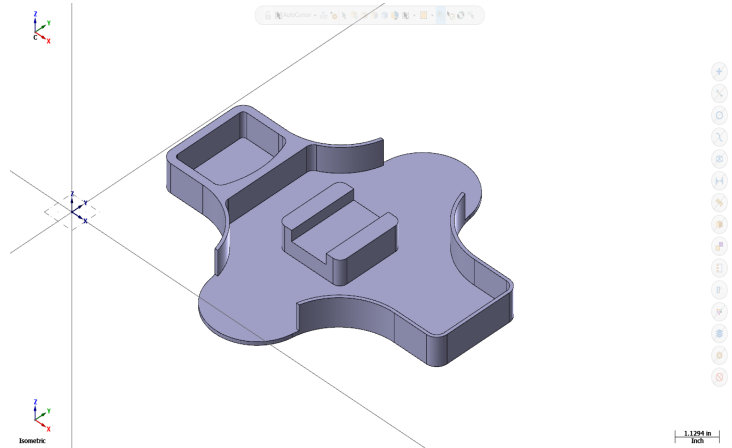
2D HST Blend Mill toolpaths morph smoothly between two open boundary chains. The toolpath supports the full depth of the cutting tool, utilizing more of the cutter's flute length resulting in less cycle time and wear.



STEP 1: PREPARE AND ORIENT MODEL FOR MACHINING

In this step you will choose the plane to machine from and position the part on the machine. Follow the same procedures to setup the machining plane as shown in the previous tutorials.

- ◆ Find and open the part “2DHST.x_t”.
- ◆ Change to the **Isometric** View and display the axes (**F9** and **Alt+F9**) to determine the model orientation.

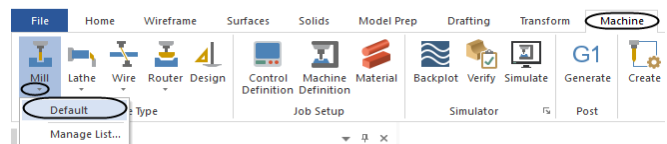


STEP 2: CREATE THE JOB SETUP

In this step you will supply Mastercam with information about the tool settings and stock size before starting to create the toolpaths.

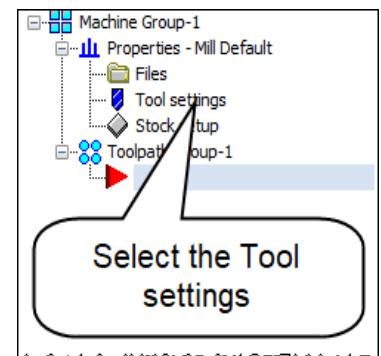
2.1 Load the Mill Default if needed

- ◆ From the **Machine** tab, select **Mill** and **Default**.



2.2 Set the Tool settings

- ◆ Click on the **Toolpaths** tab to open **Toolpath Manager**.
- ◆ Expand the **Properties** and select the **Tool settings** icon.



- ◆ Change the highlighted parameters as shown.

*Note: All of these settings should be pre-set from the saved **Operations defaults** we setup in the previous lesson.*

Machine Group Properties

Files Tool Settings Stock Setup

Default program number 1006

Feed Calculation

- ☒ From tool
- ☐ From material
- ☐ From defaults
- ☐ User defined

Spindle speed 5000.0

Feed rate 50.0

Retract rate 125.0

Plunge rate 25.0

☐ Adjust feed on arc move

Minimum arc feed 5.0

Toolpath Configuration

- ☒ Assign tool numbers sequentially
- ☒ Warn of duplicate tool numbers
- ☐ Use tool's step, peck, coolant
- ☐ Search tool library when entering a tool number

Advanced options

- ☒ Override defaults with modal values
- ☒ Clearance height
- ☒ Retract height
- ☒ Feed plane

Sequence number

Start 100.0

Increment 10.0

Material

ALUMINUM inch - 2024 Edit... Select...

- ◆ Select the **Stock Setup** tab.
- ◆ Click on the **Bounding Box** button.

Stock Plane

Top

Shape

- ☒ Rectangular
- ☐ Cylindrical
- ☐ Solid/Mesh
- ☐ File

Axis

- ☒ X
- ☐ Y
- ☐ Z

☒ Display

- ☒ Fit screen
- ☒ Wire frame
- ☐ Solid

Stock Origin

In view coordinates

X 0.0

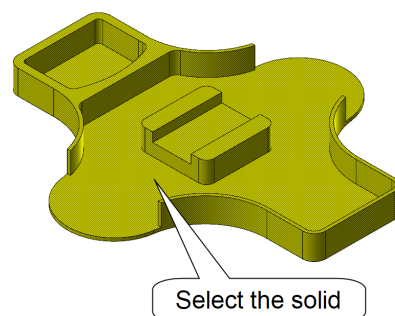
Y 0.0

Z 0.0

Select corners... Bounding box NCI extents

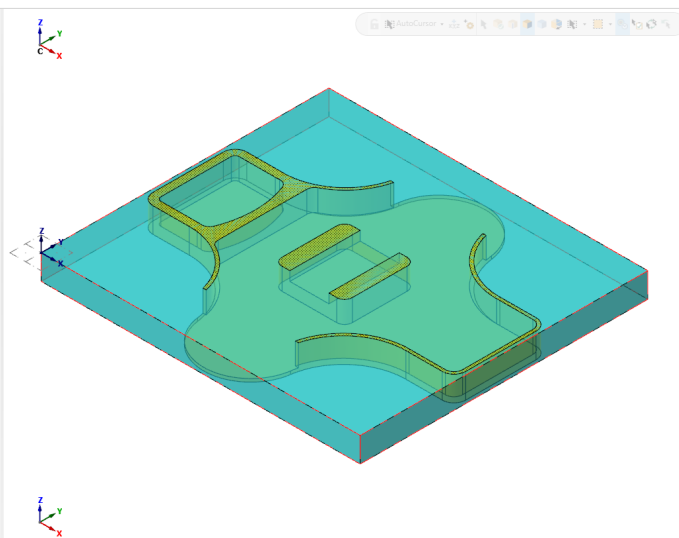
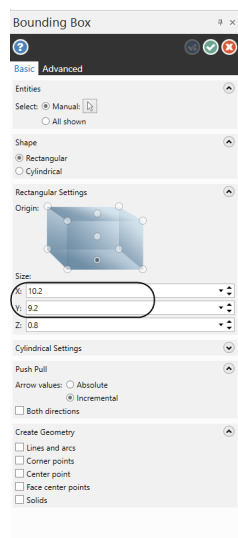
All Surfaces All Solids All Entities Unselect All

- ◆ Click on the solid to select it and press **Enter** to finish the selection.



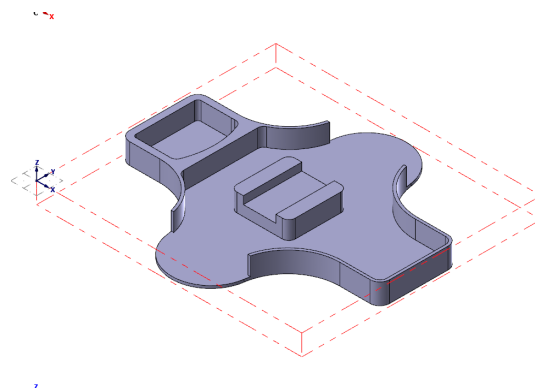
Note: Mastercam creates an image of the stock envelope. This can be saved as geometry and re-sized as needed. See Mastercam Help for more information.

- ◆ Set the **Bounding Box** parameters:
- ◆ **Shape** set to **Rectangular**.
- ◆ **Size X = 10.2000**.
- ◆ **Size Y = 9.2000**.
- ◆ **Size Z = 0.8**.
- ◆ Select the **OK** button to exit the **Bounding Box** panel.



Note: This will create the stock we will use as boundaries for our HST toolpaths. We expanded stock to match imported design intent and origin location. Users can also expand dynamically with Push Pull options

- ◆ Click the **OK** button to accept and close the **Machine Group Properties**.
- ◆ The stock should look as shown.



Note: The axes displays have been turned off.

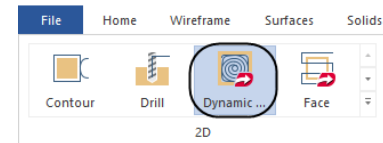
Note: You will generate several separate operations to machine the part in this lesson. These operations will include the latest technology from Mastercam. The operations include: Dynamic Mill - From Outside Dynamic Mill to remove inner material 2D HST Peel Mill and Dynamic Rest Mill.

STEP 3: DYNAMIC MILL TOOLPATH TO MACHINE THE OUTSIDE SHAPE

Dynamic Mill toolpath machines cores or pockets using the entire flute length. The toolpath supports many powerful entry methods, including a customized entry method. Entry methods and micro lifts support custom feeds and speeds to optimize and generate safe tool motion. The toolpath depends on the **Machining strategy** that you choose in the **Chain Options**. If the strategy chosen is **From outside**, the toolpath starts at the outmost chain and works its way in, taking on the final shape of the part as it approaches the final pass. You can also machine pockets, in which case the strategy selected is **Start inside**, which keeps the tool inside the machining regions.

Toolpaths

- ◆ From the **2D** group, select the **Dynamic Mill** icon.



2D HST CHAIN REFERENCE INFORMATION

For further information on settings, see the Mastercam Help file.

Machining regions: Allows selecting areas where the tool will remove material. If no machining region is selected, the area to be machined defaults to the stock size defined in Stock Setup.

Machining region Strategy:

Stay inside: keeps the toolpath inside the selected machining region.

From outside: Causes the toolpath to start from outside and work toward the machining region.

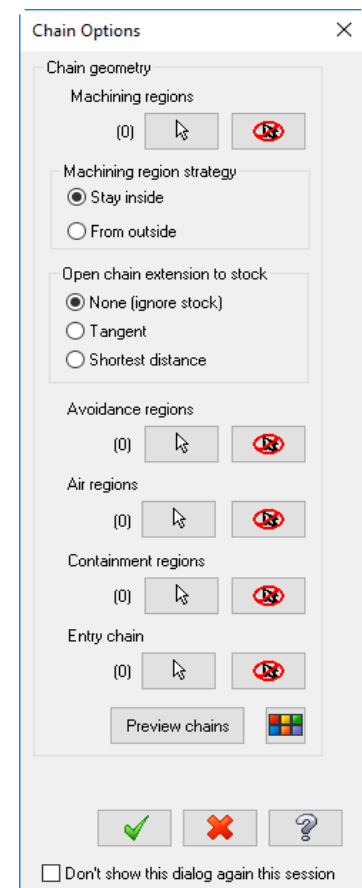
Open chain extension to stock: defines the option to cut open chains to the stock defined in stock setup.

Avoidance regions: defines areas to be avoided during machining.

Air regions: allows defining areas where there is no material so a tool can travel through it when machining.

Containment regions: defines an area where the tool cannot travel outside of.

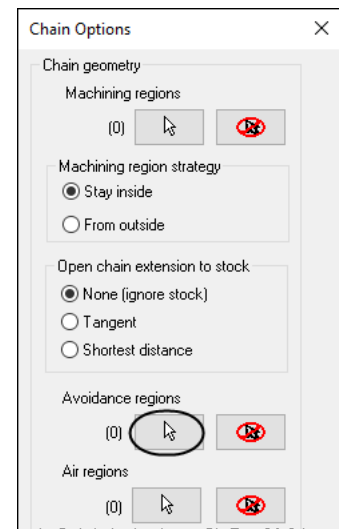
Entry chain: defines a chain of geometry or point where the tool enters the part to begin machining.



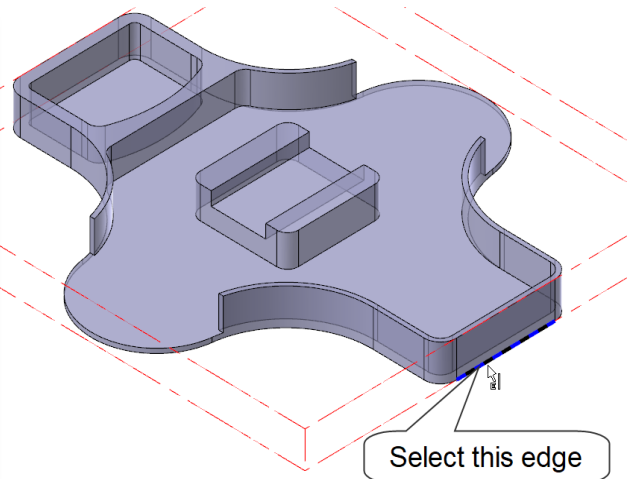
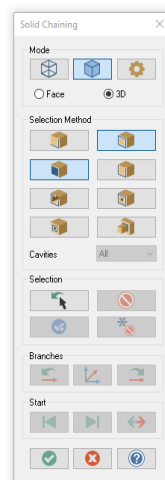
- ◆ The **Chain Options** appears as shown.
- ◆ For our toolpath, no **Machining region** needs to be selected.

*Note: When no geometry is selected, the area to be machined defaults to the **Stock Size** from the **Stock Setup** that we completed previously.*

- ◆ In the **Avoidance Regions**, click on the **Select** button.



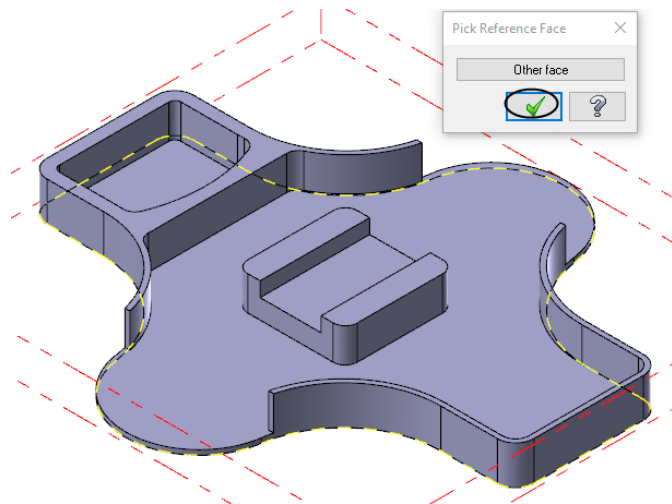
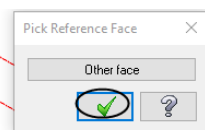
- ◆ In the **Chaining** dialog box leave the **Loop** button selected.
- ◆ Click on the edge of the bottom of the part to select the outside profile.



- ◆ In the **Pick Reference Face** dialog box, select the **OK** button if the bottom of the profile is selected as shown.

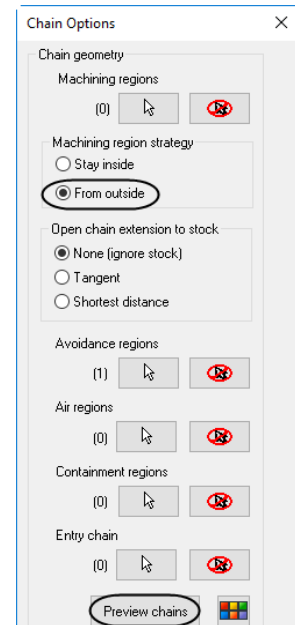
Otherwise, click on the **Other face** button.


- ◆ Select the **OK** button to exit **Chaining** dialog box.

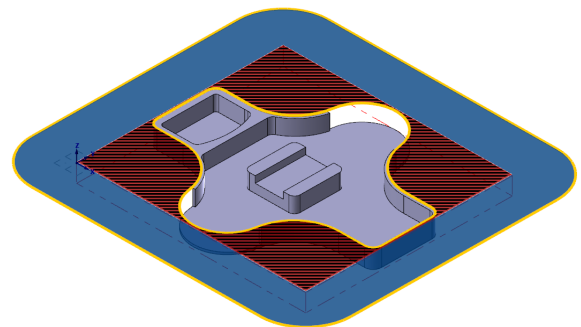


*Note: There will be no chains under **Machining Regions** and one under **Avoidance Regions**. With this type of toolpath, we do not need to set **Direction** since the machining occurs between the two boundaries selected.*

- ◆ Enable **From outside**.
- ◆ Click on the **Preview chains** button to check your selection.



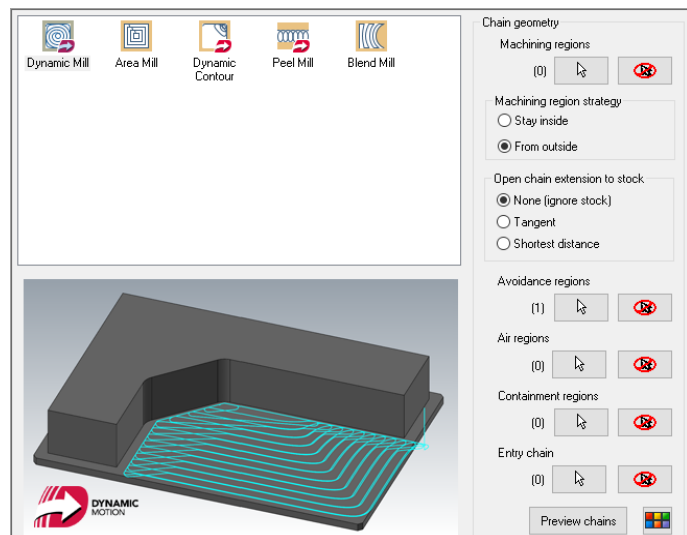
- ◆ Press **Esc** button when done.
- ◆ Click again on the **Preview chains** button to remove the display.
- ◆ Select the **OK** button to exit the **Chain Options** dialog box. 



- ◆ The **2D Toolpaths - Dynamic Mill** dialog box appears on the screen as shown.

*Note: The **Avoidance region** selected shows as **(1)**.*

Changes to any of the chaining areas can be modified from this dialog box.



*Note: In the following steps you will be setting the parameters to generate the **Dynamic Mill** toolpath.*

3.1 Tool

In the following steps you will choose a **1.0" Flat Endmill** from the tool library provided by Mastercam. You will use the **Filter** option to search through the tool library for a specific tool type which makes it easier and quicker to find the desired tool. The model drawing indicates that the smallest inside radius on the outer contour is 2.00 which can be machined with the **1.0" Flat Endmill**.

- ◆ Click on the **Select library tool** button and using the **Filter** options select the **1.0" Flat Endmill**.
- ◆ Rapid Retract enabled.
- ◆ Comment: "Use Dynamic mill to remove outside material".

#	Assembly Na...	Tool Name	Holder Name
1	--	1 INCH FLA...	--

Tool diameter: 1.0
 Corner radius: 0.0
 Tool name: 1 INCH FLAT ENDMILL
 Tool #: 1 Length offset: 1
 Head #: 0 Diameter offset: 1

☐ RCTF Spindle direction: CW
 Feed rate: 6.4176 Spindle speed: 534
 FPT: 0.003 SFM 139.7906
 Plunge rate: 6.4176 Retract rate: 6.4176
☐ Force tool change ☒ Rapid Retract

Comment
 Use Dynamic mill to remove outside material.

www.eMastercam.com

Sitewide Subscription

An eMastercam sitewide subscription will grant you instant access to all of our online resources, including our full library of eBooks and eCourses. Once subscribed, you will be able to watch all of our streaming eCourses in your browser or read any of our Mastercam eBooks using our Webviewer.

[www.eMastercam.com/
sitewide-subscription](http://www.eMastercam.com/sitewide-subscription)



Decades of experience. Thousands of post processors.

Choose In-House Solutions for all your post needs.



Implementing successful post processors for decades, the post department at In-House Solutions has earned a reputation for quality, resulting in one of the largest post departments in the world. We have an extensive library built using our new post engine, "IKE". We are continually developing and expanding our product to include new machines and controllers everyday, and if a post is not already available, we will develop a custom one for nearly any machine. We have great relationships with OEMs who provide technical information for both machines and controls, which in turn allows us to produce post processors that will generate edit-free code and run your machine efficiently.

Be sure to ask your Mastercam reseller for an In-House Solutions IKE post!

www.inhousesolutions.com/posts



Mastercam® @Courses



Included with each eCourse:
HLE demo software,
corresponding PDF eBook,
and personalized
final certificate.

LOGIN & LEARN!



Expand your Mastercam knowledge, anytime and at your own pace, with our selection of step-by-step online video eCourses.

Mastercam eCourses provide both novice and seasoned Mastercam users with the tools and information they need to excel.

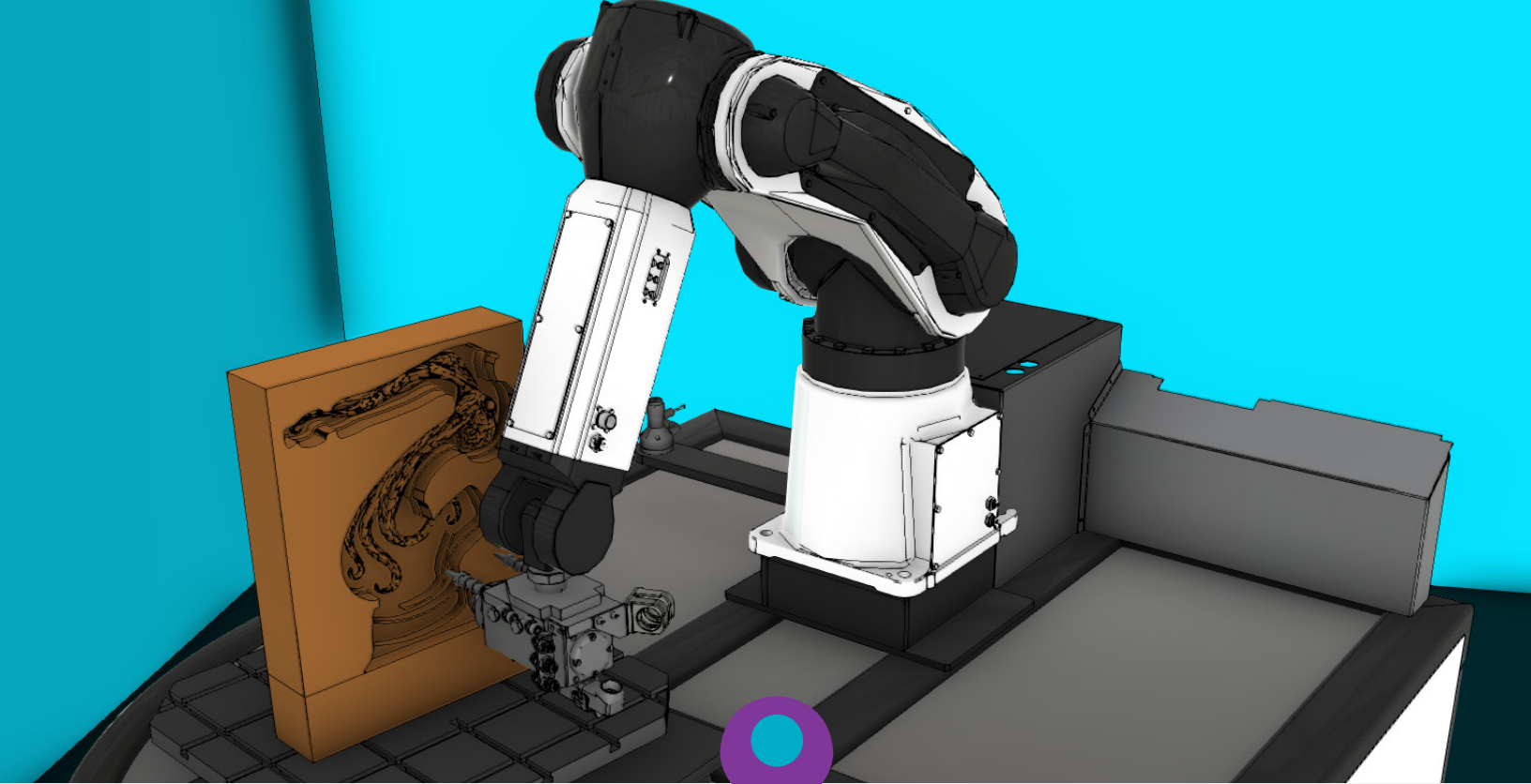
The “log in & learn” format of each eCourse allows users to set up their own online classroom, where each user’s experience is customized and tracked. Because the eCourses are not subscription-based and have no expiration date, users can log on and off at any time, and finish the program at their convenience without additional expense.

Approximate completion time ranges from 7.5 to 15 hours depending on the eCourse, making it possible for employees or students to gain new skills outside of work or studies.

Highlights:

- Online previews with table of contents including the time it takes to complete each session.
- Mastercam 2020 Home Learning Edition Demo Software download is included so you can follow along with our instructors.
- Corresponding eBook is provided.
- Quizzes follow each tutorial.
- Personalized certificate of completion for each successfully completed eCourse.
- **Instant, lifetime access**

www.eMastercam.com/eCourses



OCTOPUZ®

Robot Programming & Simulation Software

Program and generate robot code in a simple and easy way, regardless of application and brand. Spend less time programming and more time manufacturing!

Why choose OCTOPUZ?

- Multiple robots? No problem!
- Most robot brands supported
- Simple simulation building
- Extensive component library

- Automatic toolpath solving
- Application versatility
- CAM interface
- Complex kinematic systems

www.OCTOPUZ.com



TELL US WHAT YOU THINK

Our goal is to provide you the best Mastercam Training Solutions,
and we do that with your help.

Tell us what you like about our training solutions.

Let us know what you think can be improved.

Give us suggestions for future products.

www.eMastercam.com/feedback

We appreciate your feedback!

eMastercam.com
YOUR ONLINE MASTERCAM SOURCE



**DID YOU
LIKE THIS
BOOK?**

Training Tutorials
Instructor Materials
Handbooks
Professional Courseware
Pro. Certification Curriculum
Online Video eCourses
Site Licenses
Sitewide Subscription
Online Community

Discover other Mastercam Training Solutions at
www.eMastercam.com