

Mastercam 2019

IMPERIAL – HANDBOOK VOLUME 1



Demo Software Download Instructions Included

MILL 2D & SOLIDS

Mastercam® 2019

Handbook Volume 1

To order more books:

Call 1-800-529-5517 or

Visit www.emastercam.com or

Contact your Mastercam dealer

Mastercam 2019 Handbook Volume 1

Copyright: 1998 -2019 In-House Solutions Inc. All rights reserved

Software: Mastercam 2019

Author: In-House Solutions

ISBN: 978-1-77146-781-0

Revision Date: May 7, 2018

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, photocopied, 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; Windows 8 and Windows 10, are registered trademarks of Microsoft Corporation.

This document requires Mastercam Mill 3D.

Table of Contents

Legend	1
Chapters	
Introduction To Mastercam	3
Mastercam Workspace	39
CAD Drawing	71
Advanced CAD Drawing	151
Solid Modeling	211
Hole Making Toolpaths	263
Contour, Model Chamfer & Thread Mill Toolpaths	339
Pocket, Face, Circle & Slot Mill Toolpaths	397
2D High Speed Toolpaths & Feature Based Machining	439
Rotary Toolpaths	505
Change Recognition	533
Appendix A - Drill Charts	549
Appendix B - G & M Codes	555
Appendix C - Speed & Feeds	559

Legend

Conventions

Key words and Mastercam menu items are shown in bold the first time they are used. Columns on the outside edges of each page and note pages at the end of each chapter provide ample space for taking notes.

Useful tips, recommended settings, best practices, and detailed instruction on the most important features are included when possible.

Extra credit exercises are included on the student CD in PDF format. These will help build your skill to a higher level.

Terms

The following terms are used throughout this book.

- **Left Click** means to click once on the left mouse button.
- **Click** means the same as left click.
- **Right Click** means to click once on the right mouse button.
- **Scroll** means to roll the mouse scroll wheel, or move the scroll index in a list.
- **Options** are Mastercam functions selected from the main menu.
- **Enter** means to select the <Enter> key on your computer keyboard.
- **Press** means to press on a keyboard key.
- **Choose** means to select a menu option or button.
- **Open/Close** means to open or close a dialog or information box.
- **Dialog Box** is a window that opens to allow for the input of information and the setting of defaults.
- **Drop Down/Flyout Menus** are menus that expand down, left, right, or up, to reveal more menu lists.
- A **Function** is the same as a menu option or command.
- **Help** means the Mastercam help files loaded with your software.

7

Contour, Model Chamfer & Thread Mill

OBJECTIVES

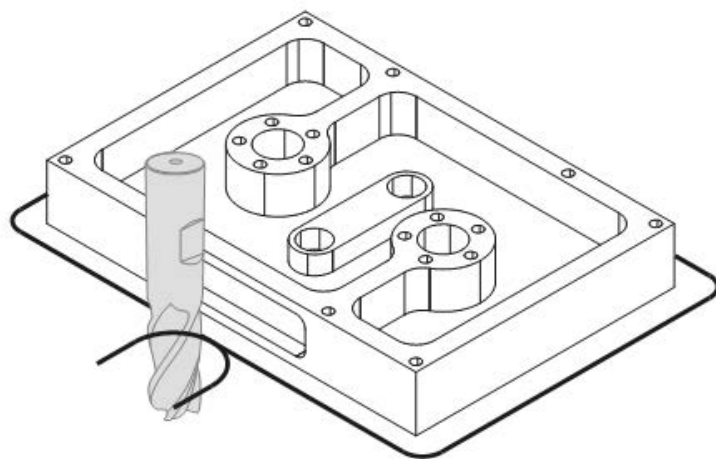
This chapter introduces contour toolpaths. Upon completion of this chapter, you should be able to do the following:

- Know the different tools used for contouring and general guidelines for tool selection
- Know how to calculate contour speeds and feeds
- Select geometry for contouring and know how to find and fix common chaining problems
- Know how to choose and create 2D, 3D, ramp, and remachining contour operations
- Understand climb, conventional, and slot milling and how to determine offset direction
- Fully understand computer, control, and wear compensation strategies, the advantages and limitations of each, and how to use them in practice
- Understand and properly set all contour parameters including speeds and feeds, cut depths, rapid heights, and more
- Know the most common lead in/out methods
- Understand issues related to machining splines, including chordal deviation and line/arc filtering
- Learn how Model Chamfer toolpath works
- Know how to create thread mill toolpaths

INTRODUCTION

Contour machining involves moving a tool around a profile of connected geometric entities, called a chain as shown in [Figure 1](#).

Figure 1

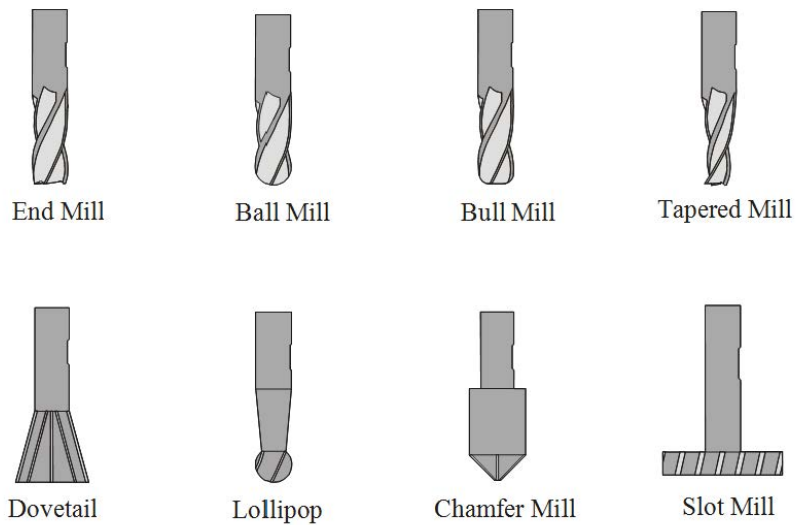


Besides roughing and finishing a part's walls, contour machining has many other applications. For example, they are used to machine O-ring grooves, deburr, and cut keyway slots.

Contour Tool Types

[Figure 2](#) shows a list of tools that can be used to machine a contour.

Figure 2



Tool	Purpose
End Mill	Used for roughing and finishing straight walls.
Ball Mill	Spherical end, used extensively for surface machining.
Bull Mill	An end mill with a corner radius. Used to put a small radius between the wall and floor of a part.
Tapered Mill	An end mill with a taper, or "draft". Used to put a slight taper on walls, typically 1-5 degrees.
Dovetail	Used to cut dovetail slots.
Lollipop	A spherical mill with a smaller shank. Allows better clearance and undercuts. Used extensively in 5-axis machining and aerospace applications.
Chamfer Mill	Similar to a highly tapered end mill. Used to form a chamfer on the edges of parts or for deburring.
Slot Mill	Also called a keyset cutter. Used for undercut slots such as O-ring grooves.

Contour Tool Selection

There are so many variables related to contour tool selection that it is difficult to establish definite rules. Your own personal experience, the advice of other experts in your shop organization, and data from tool catalogs and sales representatives can be very helpful.

It is useful to have some basic rules for guiding your decisions. For example, try to select the largest practical tool diameter. That is usually the biggest tool that will fit within the smallest internal corner radius.

If the profile includes very small inside arcs, consider using a large tool first, then cleaning up these small corners using Contour Remachining with a smaller tool.

Contours usually provide ample room for the chips to be ejected from the work area (chip clearance). Thus, three- or four-flute end mills are practical for many contour machining operations. These not only allow higher rates of material removal than two flute-mills, but also are more rigid and produce less vibration.

Keep the flute and overall length of the cutter as short as is practical. The flute length of the cutter divided by the diameter should be less than about 2.5-3.0. For example, a 1/4" diameter tool should be no longer than about 3/4. If the ratio of the tool you use is higher, expect to reduce speeds and feeds to reduce tool vibration (chatter).

Tool stepover, step down, and the amount of finish pass vary depending on the tool, material, and application. A rule of thumb is to use values equal to the tool radius in aluminum, less in harder materials.

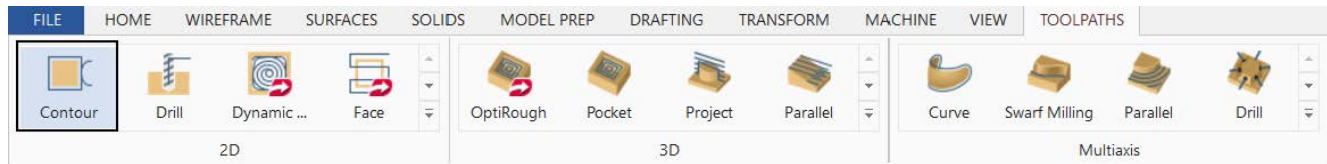
If the contour must be held to tight tolerances or requires a very good surface finish, a finish pass is used. Finish passes typically remove between 0.015" to 0.05" of stock, depending on the material and application.

High Speed Steel end mills have good utility and are relatively inexpensive. Solid Carbide cutters have much longer life and better material removal rates.

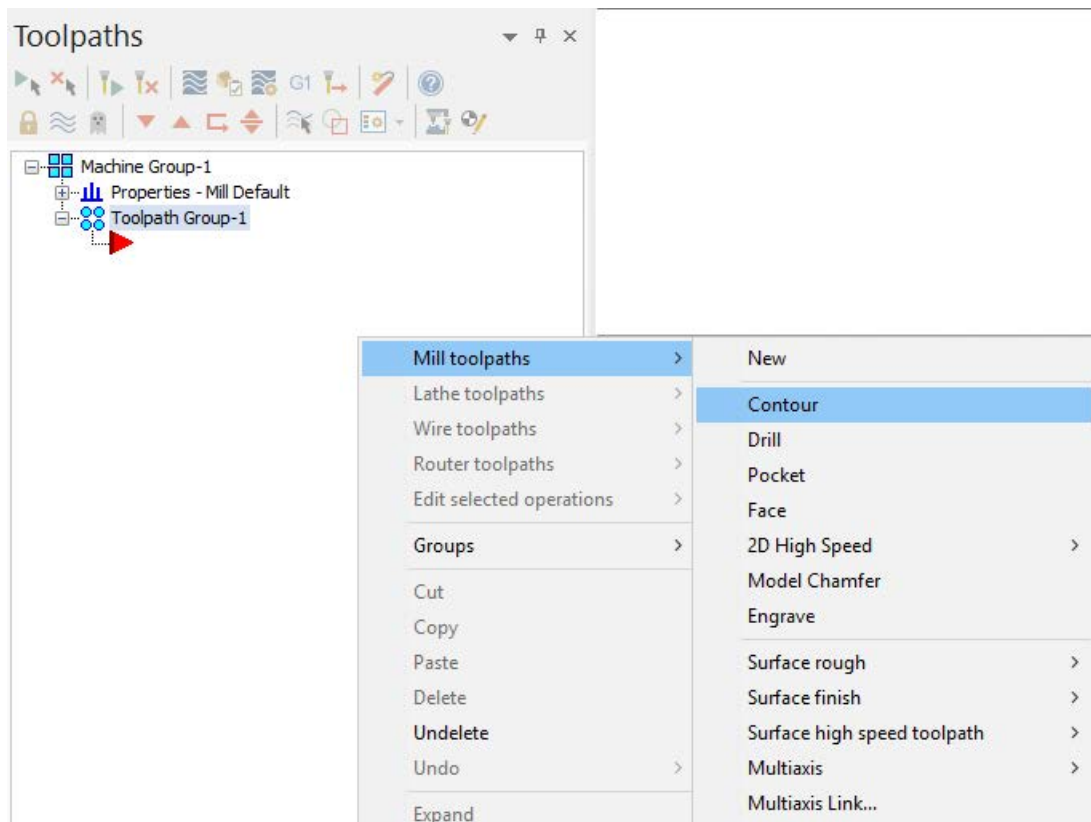
Creating Contour Toolpaths

Select contour toolpaths using any of these methods:

- From the **TOOLPATHS** tab, select the **Contour** icon from the **2D** group.



- From the **Toolpaths Manager**, right mouse click and select **Mill Toolpaths**, then **Contour**.

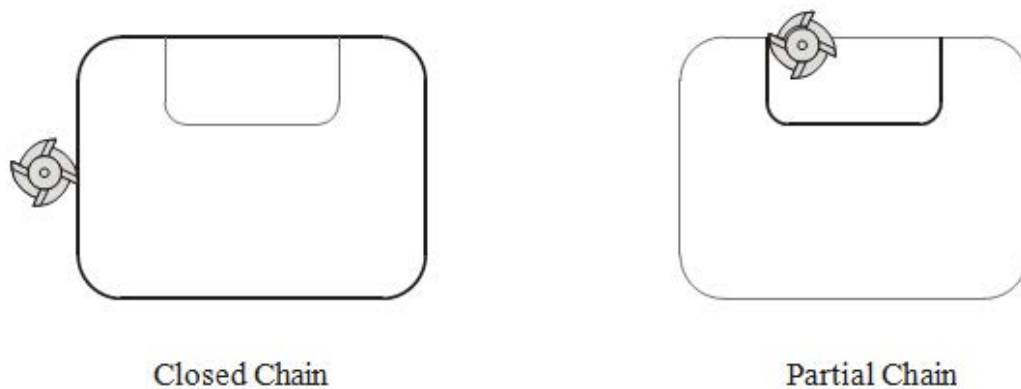


Toolpaths	Purpose
Contour	Drives a tool around a 2D or 3D profile. Both multiple depths of cut and XY passes can be specified.

Chaining

Contour toolpaths involve moving a tool around a profile of entities, called a chain. A chain can consist of wireframe geometry, or the edges and faces of solid bodies. In most cases, this profile is closed. Sometimes the chain is not closed, and is referred to as a partial chain as shown in [Figure 3](#).

Figure 3

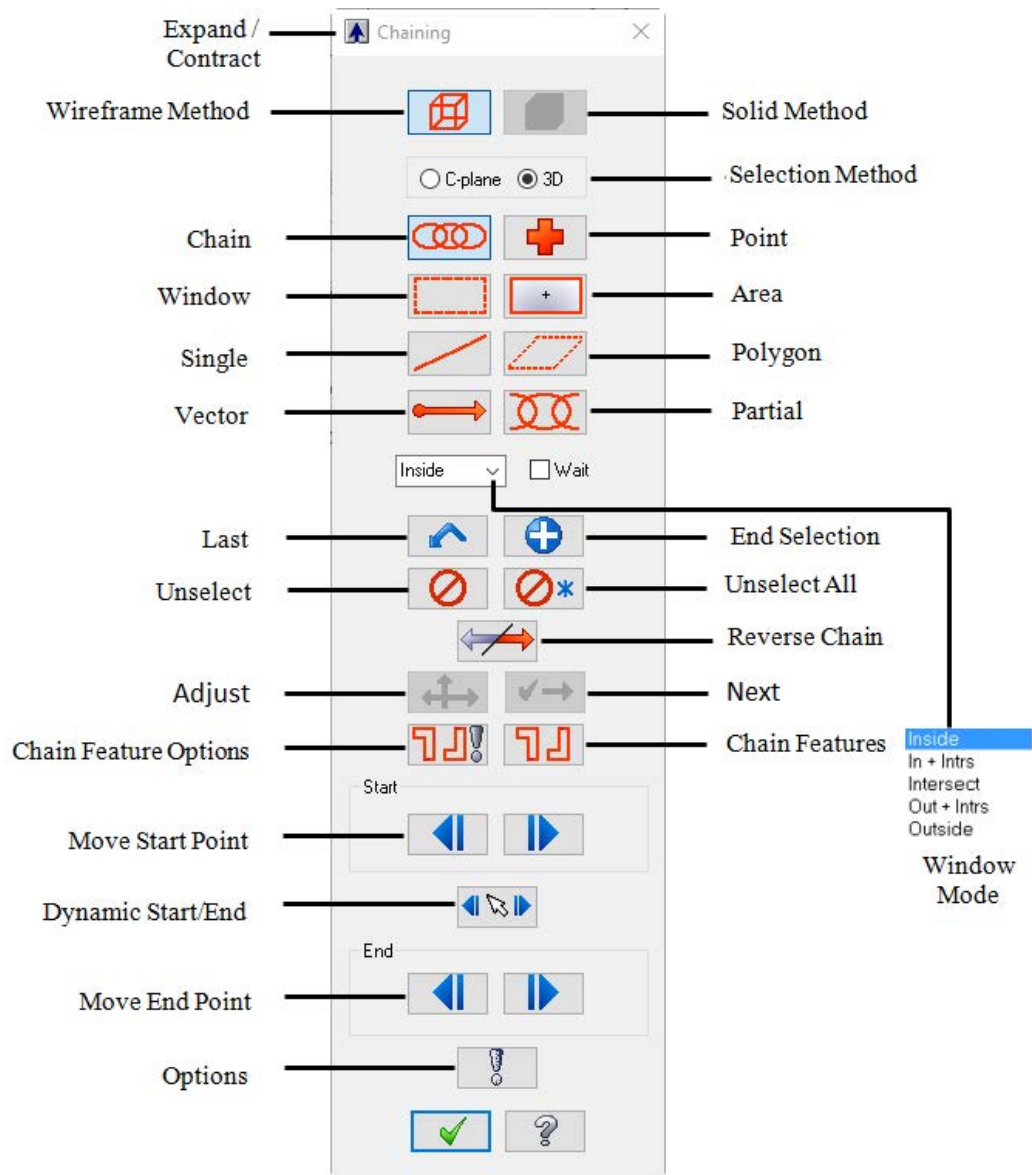


The process of selecting the profile of entities is called chaining. Chaining does several things:

- It selects and sequences the geometry used to control the machining path.
- Sets the machining direction.
- If the chain is closed, it sets the start point for the toolpath.

After selecting a contour function, Mastercam launches the chaining dialog box as shown in [Figure 4](#), which is used to select toolpath geometry.

Figure 4



Item	Definition
Wireframe	Sets chaining to select wireframe geometry, i.e. lines, arcs, points and splines.
Solids	Sets chaining to select the edges and faces of solid bodies. This option is only selectable with solids visible on the screen.
C-Plane	Sets chaining so only entities on the same Cplane are selectable. Use this mode when selecting 3D wireframe geometry so the tool does not pause at every intersection with extrusion lines.
3D	Do not restrict chaining to entities on the same plane.
Chain	To select a closed loop of entities.
Window	To select entities for chaining using a rectangular fence.
Single	To select only the entity picked.
Vector	To select entities crossing a sketched line.
Point	To select a point. Points are used for several purposes, including setting a contour's start point or a location for the tool to move through to/from contours.
Area	To select entities by picking within a closed area.
Polygon	To select entities for chaining using a polygon fence.
Partial Chain	A section of a continuous profile.
Window Mode	Determines how window select works.
Last	To reselect the chain from the previous operation.
End Chain	To end the chaining process for this loop.
Reverse	To change the direction of a chain.
Adjust	To guide the chaining when a branch point has been reached and allows you to select a different direction or entity.
Next	To guide the chaining when a branch point has been reached and allows you to continue in the direction of the red arrow.
Unselect	To unselect an entity from the chain.
Chain Feature Option	Allows you to set the way Mastercam recognizes related chains.
Chain Feature	Instructs Mastercam to add chains to the chain manager.
Move Start Point	To move the start of chain. For contouring, tool entry point is based on the start of the first entity selected. Moving the start point changes where the tool enters/exits a closed contour.
Dynamic	Sets the start/end point by dragging the cursor along a profile.

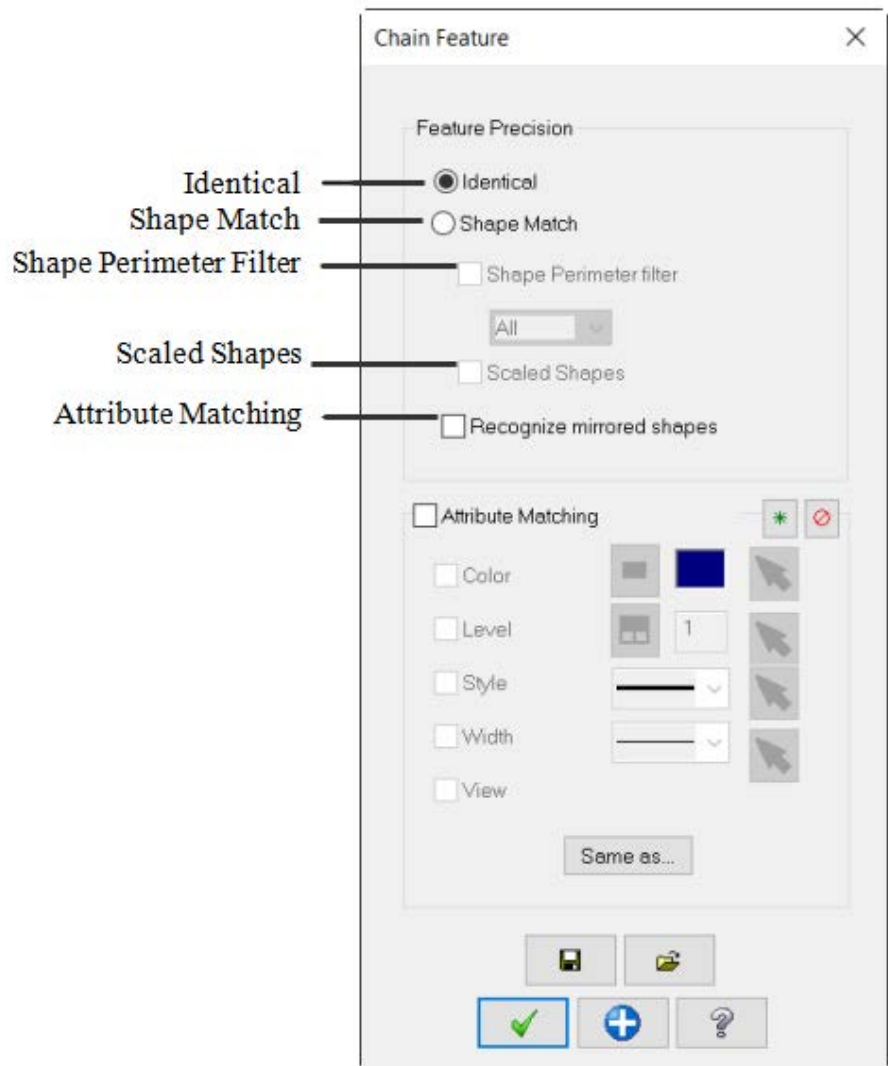
Item	Definition
Move End Point	Active for partial chains. To change the end point.
Options	Chaining options provide extensive control over how chaining works.

Chain Feature Options

Chaining Feature Options gives you more control over chaining. You can choose identical features meaning they must have the same number of entities, with the same measurements. There is also an option to select chains which are similar in shape. The chains selected will have the same features and arrangement but will differ by size.

[Figure 5](#) shows the **Chain Feature** dialog box.

Figure 5



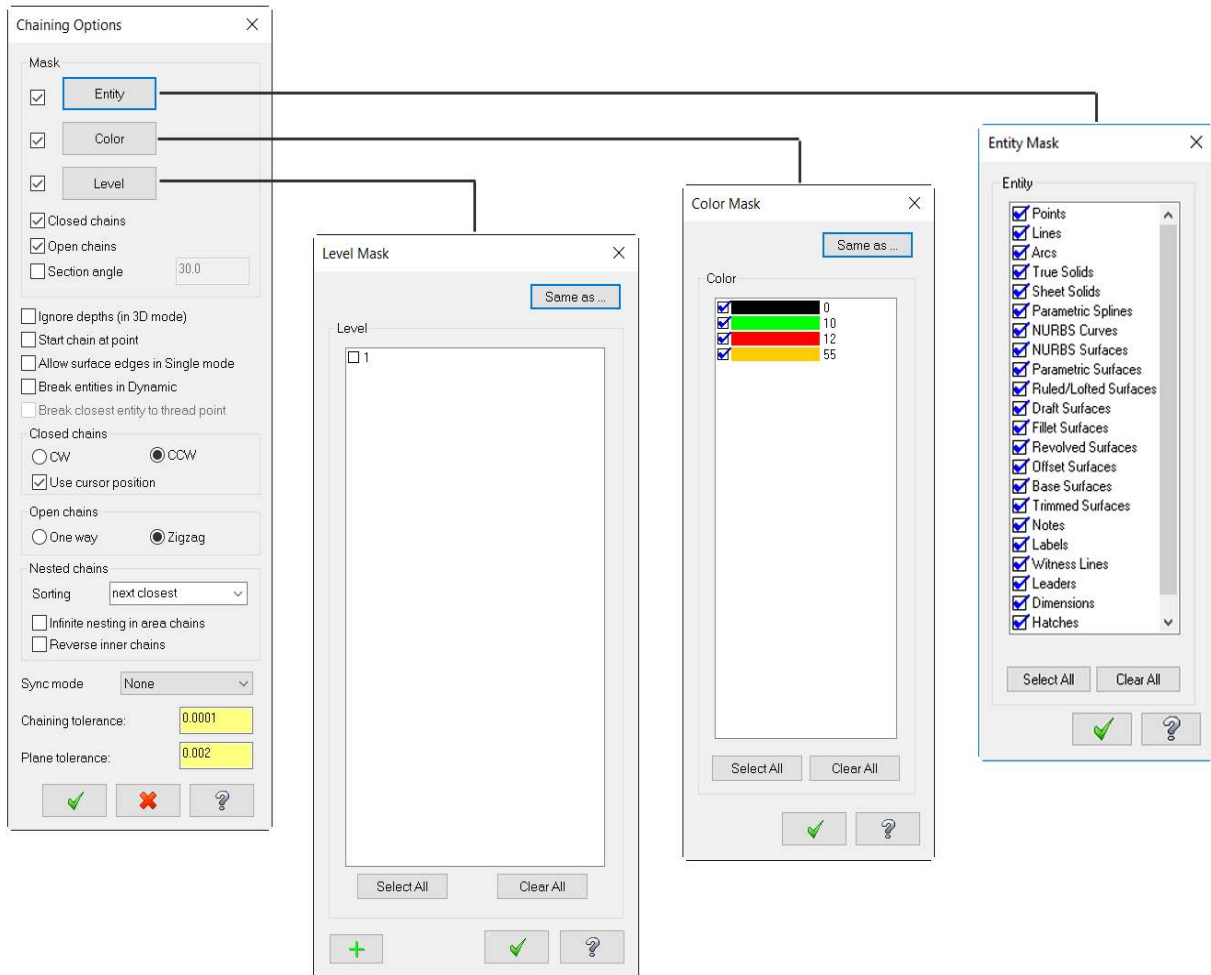
Item	Definition
Feature Precision	Controls the geometric recognition of the chains.
Identical	Looks for features which are the exact geometrical match.
Shape Match	Finds features that are similar in shape. Shapes must have the same features and arrangement but differ by size.
Shape Perimeter Filter	Evaluates the length of the chain perimeter to provide an additional selection filter.
Scaled Shapes	Limits the chain selection to those that are scaled versions of the Original. The chains must contain the same number of entities in the proportions.
Attribute Matching	Lets you fine tune the selection of chains.
Color	Restricts the chain that will be selected based on color.
Level	Restricts the chains that will be selected based on level.
Style	Restricts the chain that will be selected based on line style.
Width	Restricts the chains that will be selected based on the line width.
View	Restricts arcs that will be selected based on the view of the first arc encountered in the chain selection.
Same As	Returns to the graphics screen to select a piece of geometry.
Save	Saves the Chain Feature option settings to a file with a chain extension. All options will be saved as they are currently set in the dialog box.
Load	Loads the previous saved chain file.

Chaining Options

Chaining has many options. **Mask options** are very useful for setting Mastercam to consider only certain entities for chaining as shown in [Figure 6](#).

Sync options are used almost exclusively for Wire EDM or Multi-axis toolpath programming.

Figure 6

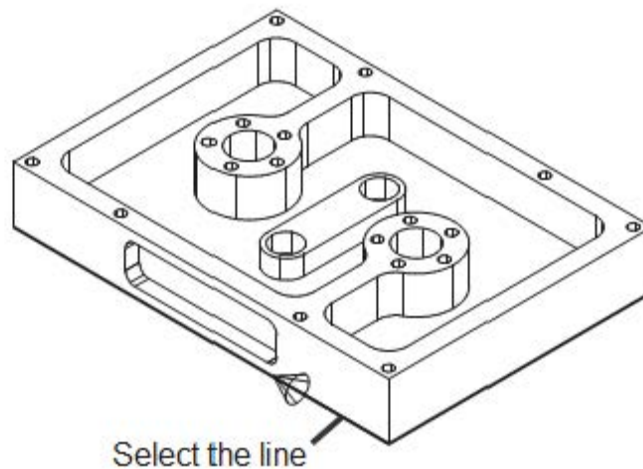


Item	Definition
Mask	This sets which entities, color, or level are selectable by chaining. Items without checkmarks beside them will be ignored when chaining.
Ignore Depths	Chaining entities that are connected in the plane, but which may have different Z-values.
Start Chain At Point	Assumes you have placed a point somewhere along each chain, and forces this point to be the contour start. This is used mostly in Wire EDM programming.
Closed Chains	Sets a mask that selects only closed chains.
Open Chains	Sets a mask that selects only open chains.
Closed Chain Direction	Sets default direction for closed chains. Use cursor position means chaining will start near the start of the first entity picked.
Open Chains	One way prevents chaining from reversing when it gets to the end of the open chain.
Nested Chains	Closed boundaries that lie inside others are called nested chains, and can be selected with a single click of the mouse using the Area option. These settings determine how the sequence and direction of these nested chains are set.
Break Entities In Dynamic	When in Dynamic mode, will break an entity into two at that point.
Section Angle	Chaining will pause if the angle between entities changes more than the specified degrees.
Sync Mode	This is used almost exclusively for Wire EDM and Multi-axis programming, but has applications in some surface creation too. Sync means to match control points on one path so that they correspond with points on another.
Chaining Tolerance	Sets the maximum gap that can exist between entities for chaining. It is recommended you leave this tolerance as it is and repair any breaks in the contour using trim and other functions geometry creation and modification to produce a perfectly closed loop.
Plane Tolerance	Determines the maximum distance an entity can be from the plane and still be chained.

Chaining Example

The following illustration shows how to machine the OD contour of the part, Exercise 5-1, Servo Housing. To machine the outside profile starting at the front of the part, proceed as follows.

- **Step 1:** Select **Toolpaths, Contour Toolpath**.
- **Step 2:** In the **Chaining** dialog box, turn on **C-Plane** and make sure the wireframe option is selected. Since this model contains both **Solid** and **Wireframe** geometry, you could use either setting, but work with the **Wireframe** option for now.
- **Step 3:** Select the line where indicated. The entire profile should change color and the single green pointer indicates the direction of cut.



- **Step 4:** Use the **Reverse** or **Move Start Point** functions if needed to make the arrow appear as shown above.
- **Step 5:** Select **Accept**.

Chaining Problems

It is not uncommon for wireframe drawings to contain errors that cause chaining problems. For example, two lines have been drawn where only one is needed. Very small gaps between entities may not even be visible at zoom extents.

Good drawing practices, including the use of the AutoCursor and trim functions, will avoid most problems. But even the most careful person makes mistakes. Therefore, it is a good practice to check the integrity of wireframe chains before starting to create toolpaths.

The most common chaining errors are caused by the following problems:

- Small Breaks
- Duplicate Entities
- Overlapping Entities

Small breaks can be avoided by using the **AutoCursor** to ensure entities connect.

One way to find small breaks is using **Screen, Display Entity Endpoints**. This function temporarily places a point at the end of each entity. Small gaps are often obvious as they appear as points that are noticeably brighter than others.

Zooming in on these areas may reveal two points close together, one at the end of each entity as shown in [Figure 7](#).

The best way to fix these gaps is to trim the two entities.

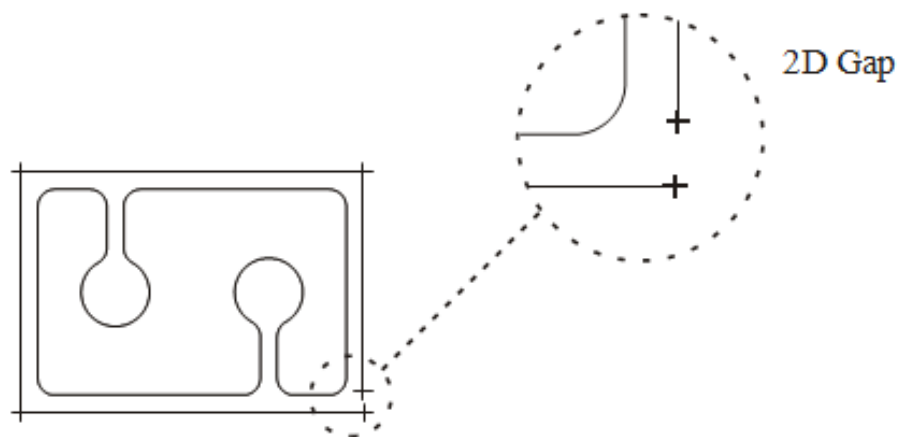
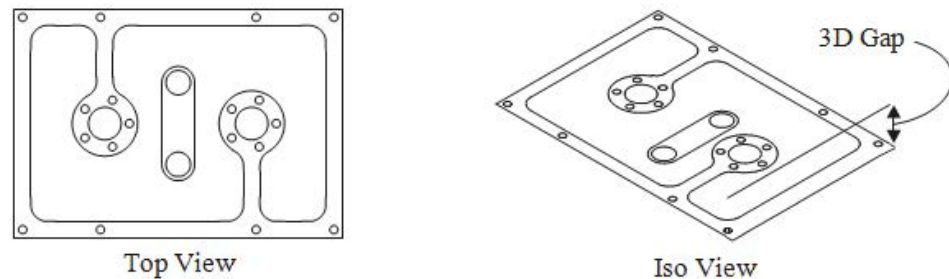


Figure 7

Display Entity Endpoints displays points until you select **Enter** on the keyboard.

Another type of break occurs when entities in the chain do not lie at the same depth. When viewed from the top, the chains appear to be continuous. Changing to an ISO view reveals one entity at a different Z-value than the others as shown in [Figure 8](#).

Figure 8



Delete Duplicates

Chaining sees intersections with duplicate entities like any branch point and does not know which entity to chain next.

Use **Delete duplicates** to find and delete duplicate geometric entities. Only true duplicates are deleted. For example, two arcs must share the same endpoints, center, radius, and plane to be considered duplicates.

Because of the way splines are defined, two splines that follow the same path may not be true duplicates, since they may have different node points. Don't rely on Delete duplicates to detect duplicate splines.

Overlapping entities share only one endpoint with another entity, and can be very difficult to detect.

Consider, for example, the profile shown in [Figure 9](#) on the next page.

It consists of four lines connecting points P1, P2, P3, and P4, and an extra line, between the points P3 and P5. This line shares an endpoint with Line 2 at P3. The other end of this line is not shared with any other entity.

When Mastercam tries to chain this profile, chaining stops at P3. Mastercam reports the message, "Branch point reached. Select next branch." Mastercam does not know if you want to follow the path along Line-3, or along the Extra Line.

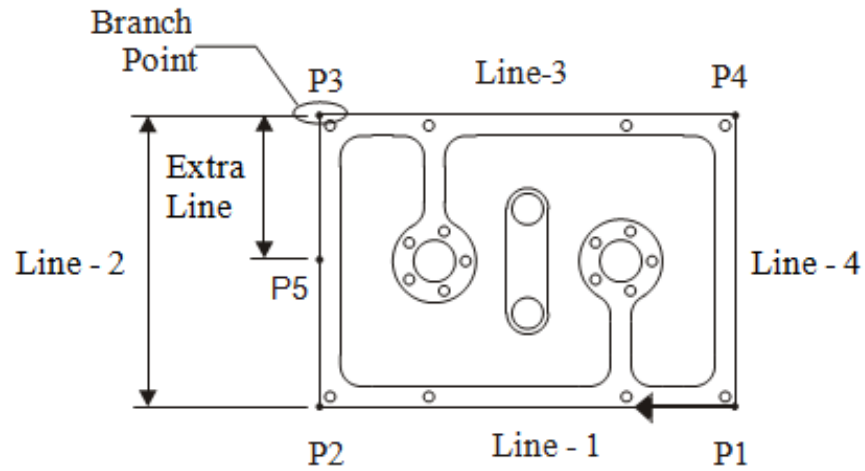
The first indication that something is wrong is that chaining does not start and end where you expected. Normally, chaining starts at the beginning of the selected entity. Shared endpoints cause chaining to start at the shared endpoint, regardless of where you clicked with the mouse.

The second indication is the Branch point reached message. Once you get used to how Mastercam works, you will notice when something does not work as you expected. Rather than ignoring these indications, take the time to look closer to find out why things are different and fix them.

It is usually best to find and fix overlapping entities and other geometry problems before attempting to create toolpaths. A watertight model makes toolpath creation much easier.

Using Display Entity Endpoints helps find overlapping entities. Look for points where you would not expect them; for example, in the middle of a straight line.

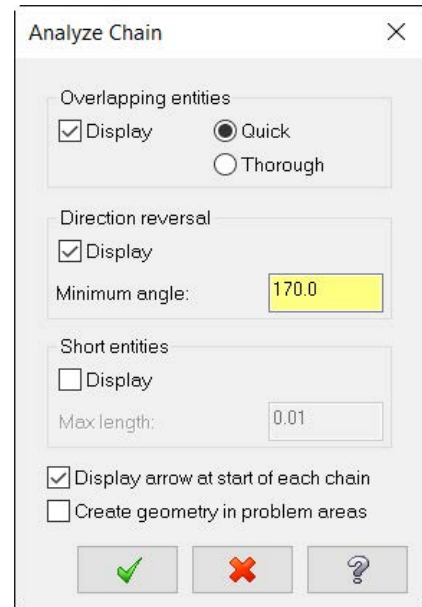
Figure 9



Analyze Chain

Analyze Chain can help find and diagnose chaining problems. [Figure 10](#) shows the **Analyze Chain** dialog box.

Figure 10

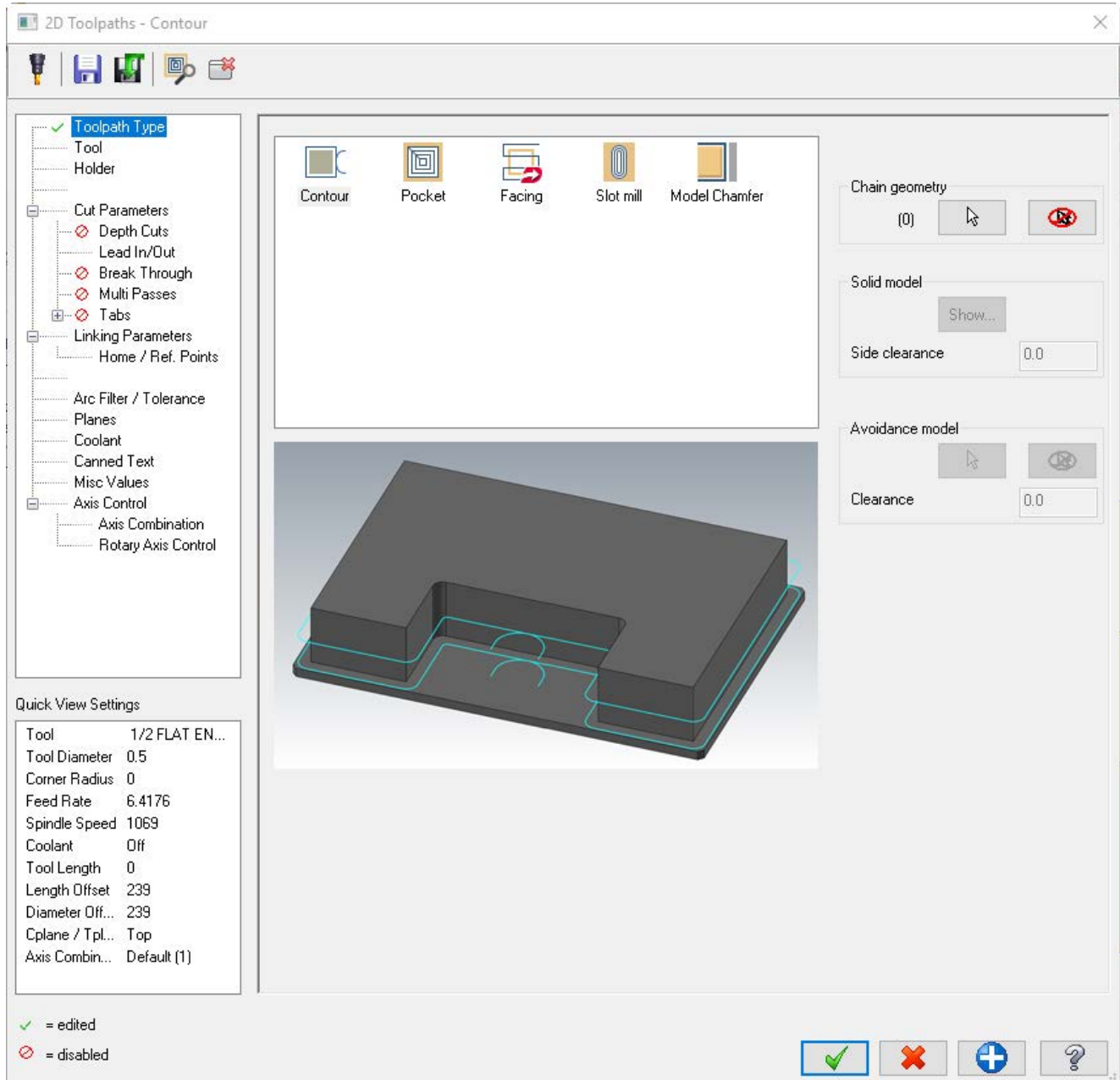


Item	Definition
Overlapping Entities	Finds and marks overlapping entities, which are the most difficult chaining problems to locate manually. Try the Quick method first and use Thorough if the overlap is not located.
Direction Reversal	Checks for entities that reverse direction suddenly. Minimum angle sets the angle that this function will detect "fish tailed" entities, or entities that change direction abruptly.
Short Entities	Finds wireframe entities that are less than the specified distance.
Display Arrow At Start Of Each Chain	Marks errors by displaying an arrow. These arrows disappear when this function is exited.
Create Geometry In Problem Areas	Creates a small arc at each problem area. These arcs remain after this function is exited.

Toolpath Type

After chaining the geometry the contour **2D Toolpaths - Contour** dialog box will appear as shown in [Figure 11](#). Down to the left side of the screen are the toolpath parameters. On this screen we can choose our toolpath type. We may also remove the chains we currently have or add more chains to the one we already have if we click on the select button in the chain geometry area.

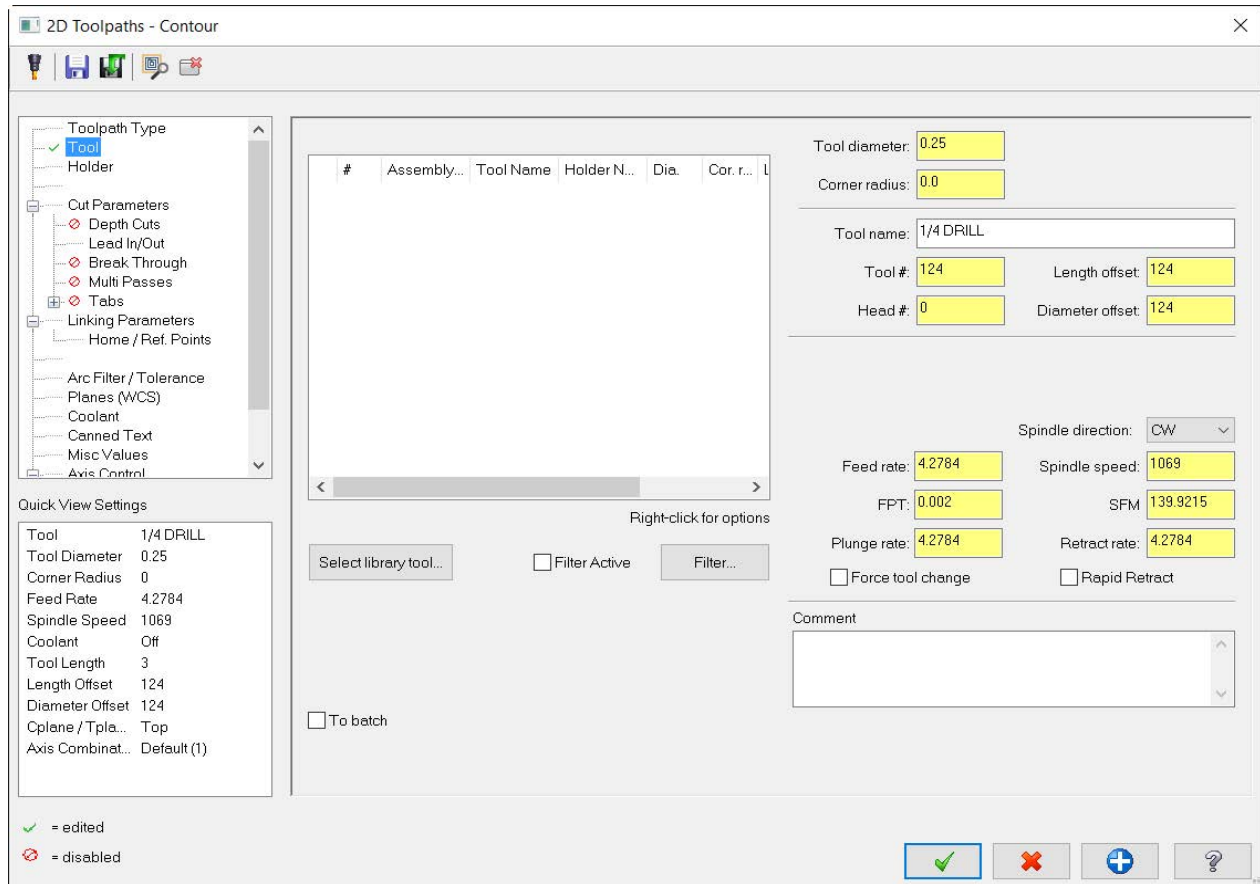
Figure 11



Tool

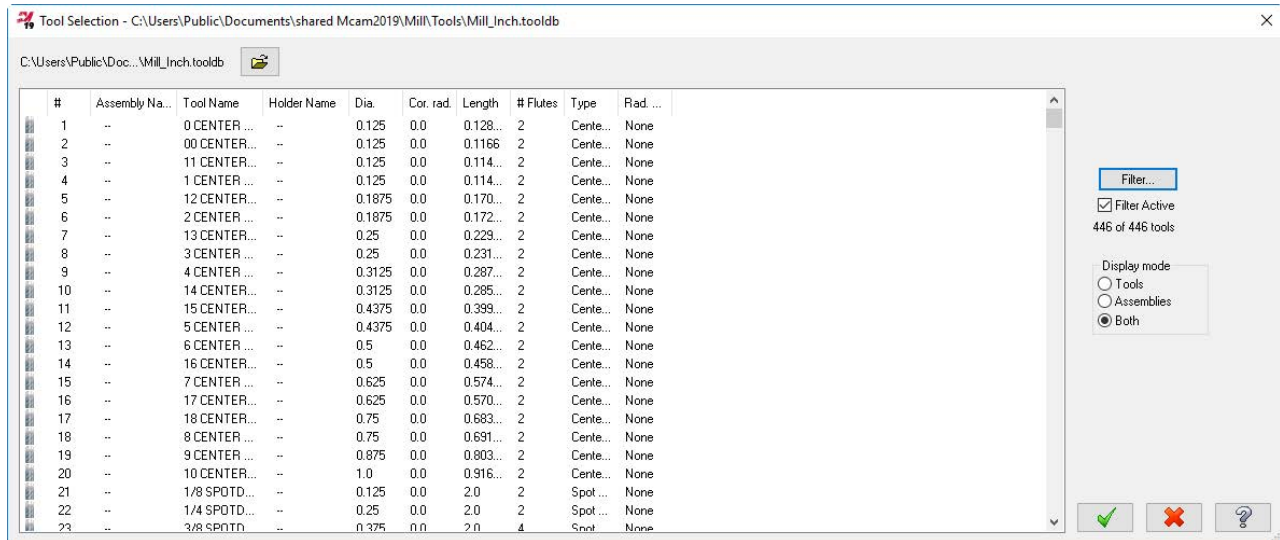
Tool allows us to choose a tool for the contour operation as shown in [Figure 12](#). Here we can also set the Feed rate and Spindle speed. Those can now be set using FPT or SFM if those numbers are available from your tooling supplier. Select a mill tool from the library, set tool parameters, and then select Holder.

Figure 12



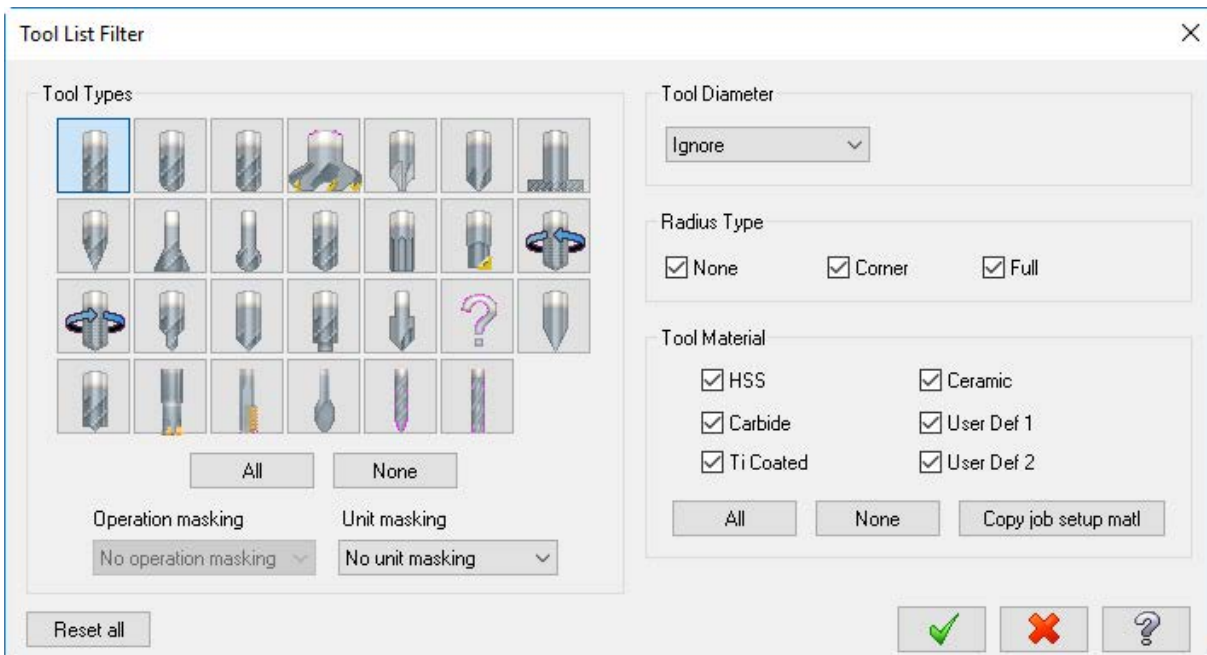
Select Library Tool

Mastercam has an extensive tool library. See all tools in the library by clicking on the **Select library tool** button.



To view only selected types of tools, proceed as follows:

- **Step 1:** Choose the **Select library tool** button on the **Tool Selection** page and then a tool selection page will appear. Choose the **Filter** button.
- **Step 2:** Select the none button and then click on the icon of the tools you want to display. In this case, the Flat End Mill icon in the far upper-left corner.

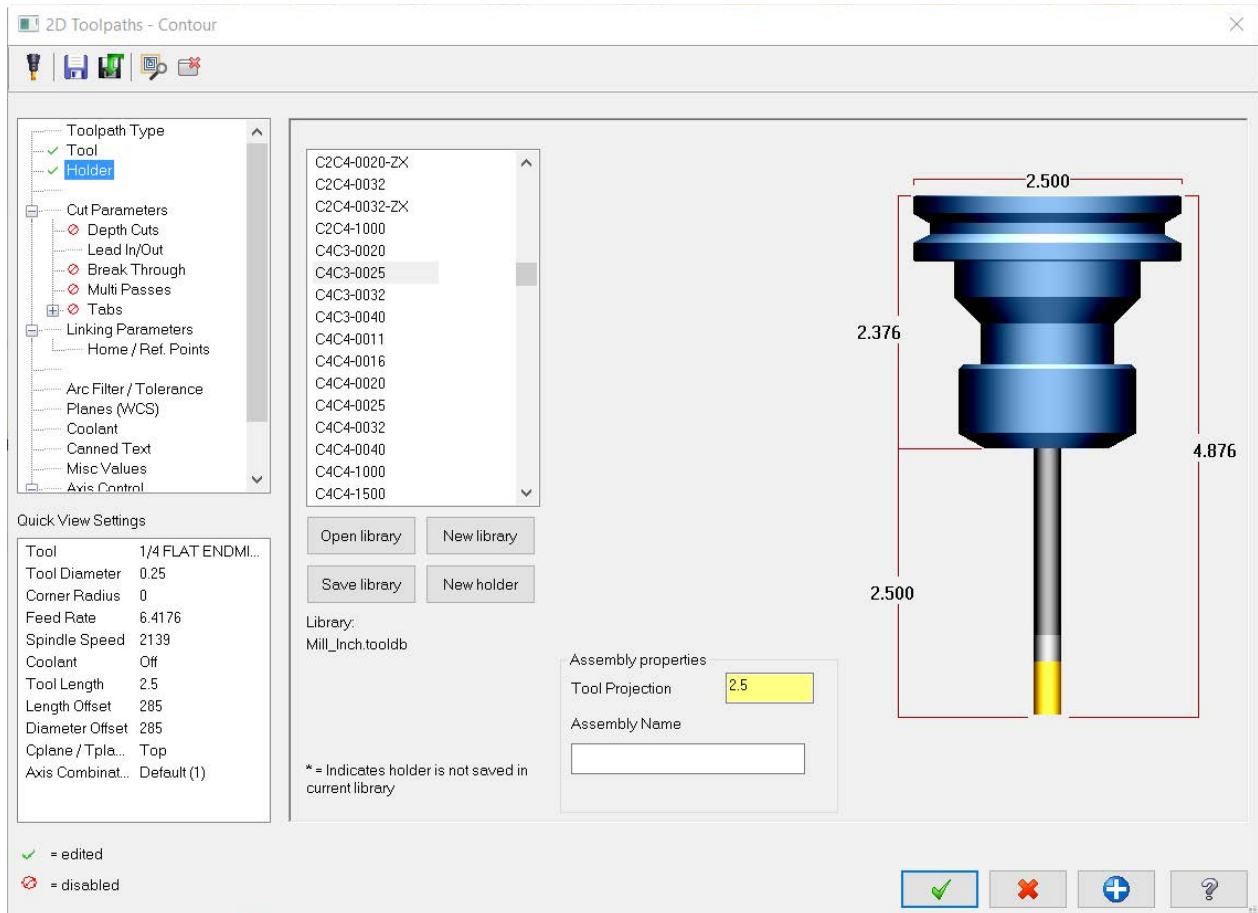


- **Step 3:** Select the **OK** button. When you close the dialog box, only the Flat End Mill will be displayed in the **Tool Selection** page.

Holder

Holder lets you select the holder to be used in the operation. You can create, load, and edit a holder if needed, as shown in [Figure 13](#). When you select a holder, Mastercam displays a preview of its geometry. When you enable the option Use holder for gouge checking, this activates a feature which will ensure your holder does not come in contact with the part.

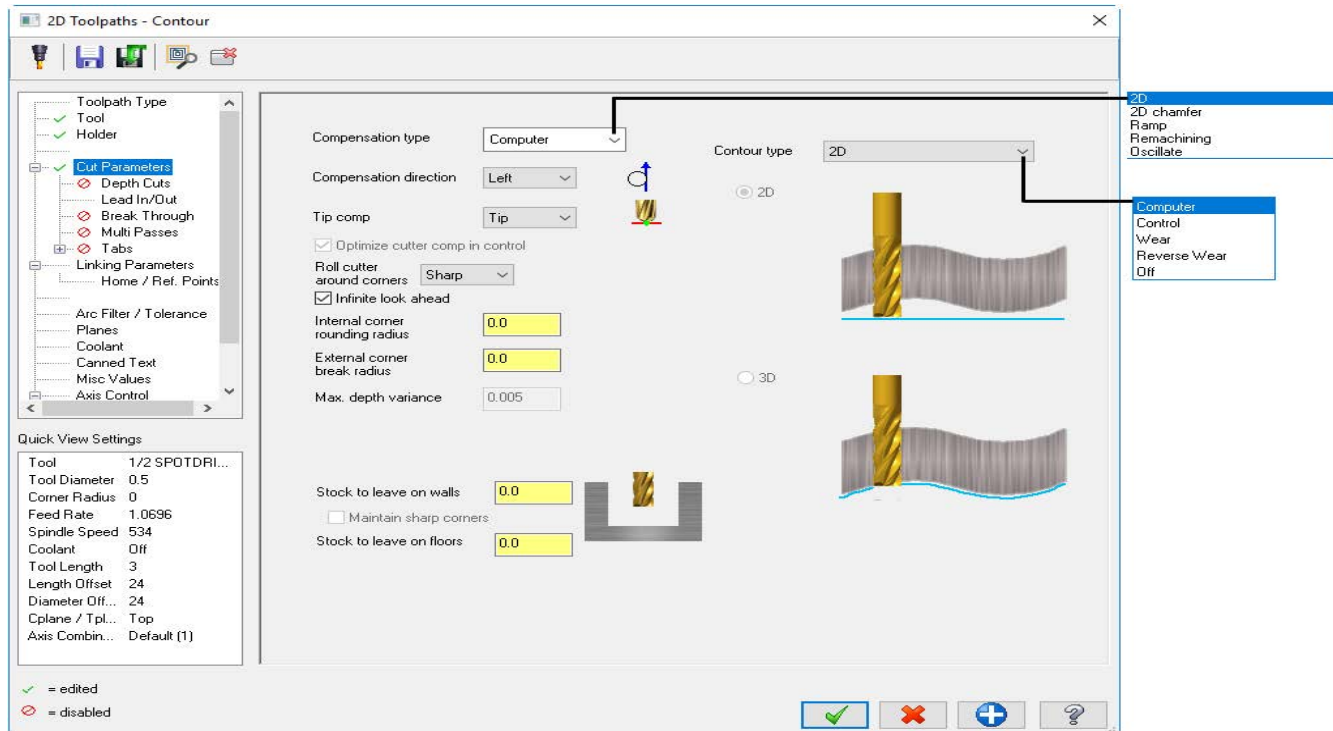
Figure 13



Contour Cut Parameters

Cut parameters control how the tool approaches, machines, and departs the contour toolpath as shown in [Figure 14](#). It also controls how cutter compensation is output, stock to leave, and many other contour machining parameters. Your screen should look like it does below.

Figure 14



Symbols

Finish Cuts 426

Numerics

2D Contour 361

2D/3D Drawing Mode 178

2D/3D Mode 44

3D Cartesian Coordinate System 172

3D Contour 362

A

Absolute coordinates 75

Absolute/Incremental 296

Air regions 442

All Mastercam X Files 28

Analyze Chain 354

Arc 82

Arc Filter/Tolerance 388

Arc Options 100

Arc Ribbon Bar 101

Arc/Filter Tolerance 453

Arcs 99

Area Mill - Cut Parameters 455

Area Mill - Transitions 457

Area Mill - Trochoidal Motion 456

Area Mill Toolpath 454

AutoCursor Cues 95

Automated Hole Processing 336

Avoidance Regions 442

Axes Designation 506

Axis Substitution 518

Axis Substitution Toolpaths 528

B

Backplot 322

Backplot Example 326

Backplot Home Tab 323

Backplot View Tab 326

Base Feature 213, 216

Bisecting lines 86

Blend Mill - Cut Parameters 464

Blend Mill - Lead In/Out 466

Blend Mill Toolpath 464

Boolean 213

Boolean Add 214

Bore 266

Bore In-Out (G85) 310

Boss 212

Bounding Box 119, 125

Brick Solid 213

C

CAD Drawing 71

CAD/CAM 3

Canned Text 427

Cartesian Coordinate System 73

Center Drill 266

Centerline Vs Of Center Machining 519

Chain Feature Options 346

Chaining 342

Chaining Example 350

Chaining Options 348

Chaining Problems 351

Chamfer 108, 213

Chamfer Contour 363

Chamfer Ribbon Bar 109

Change Recognition 533, 538

Change Recognition Step By Step 544

Changing Toolpaths 320

Changing Views 50

Check All Tracked Files 537

Check Current File 535

Circle Mill Toolpath 431

Clear All Solid Face and Feature Color 241

Climb vs Conventional Machining 369

Color 44, 47

Communicate 334

Compensation Direction 371

Configuration 66

Configure AutoCursor Dialog 96

Constant Radius 227

Construction Depth (Z

) 175

Construction Depth Example 176

Construction Plane (Cplane) 173

Containment regions 442

Contour Break Through 383

Contour Cut Parameters 359

Contour Depth Cuts 377

Contour Lead In/Out 379

Contour Linking Parameters 386

Contour Multi Passes 384

Contour Remachining 367

Contour Tool Selection 341

Contour Tool Types 340

Contour Toolpath 342

Contour Toolpaths 339

Coolant 453

Copy WCS 277

Counterbore	266	Dynamic Contour Toolpath	458
Counterbore (G82)	303	Dynamic Mill - Break Through	451
Countersink	266	Dynamic Mill - Cut Parameters	445
CPlane 2 Lines Example	183	Dynamic Mill - Depth Cuts	447
Cplane Rotate	186	Dynamic Mill - Entry Motion	448
Create a New Toolbar	60	Dynamic Mill - Linking Parameters	452
Create New Folder	28	Dynamic Mill - Rest Material	450
Create Points	114	Dynamic Mill Toolpath	444
Creating Arcs	104	Dynamic Planes	182
Creating Axis Substitution Toolpaths	522	Dynamic Xform	152
Creating Contour Toolpaths	342	E	
Creating Fillets	107	Edge	212
Creating Lines	90	Editor	332
Creating Planes	180	Ellipse	119
Creating Pocket Toolpaths	400	English/Metric	165
Customize Drop Downs/Context Menu RMB ..	62	Enter Coordinate Values and AutoCursor Toolbar	93
Customize Drop Downs/RMB	62	Entering Numeric Data	97
Customizing Toolbars	53	Entry Chain	442
Cut	212	Extrude	214, 217
Cut Parameters - Facing	475	Extrude Base Feature	221
Cut Parameters - Restmil	478	Extrude Dialog Box	218
Cut Parameters - Roughing	477	Extrude Direction	213
Cutter Compensation	370	F	
Cutter Compensation in Computer	372	Face	212
Cutter Compensation in Control	373	Face Toolpath	428
D		Facing Pockets	405
Datum	78	Facing Tools	472
Delete	127	FBM Additional Settings	485, 501
Delete Duplicates	352	FBM Circle Mill Parameters	482
Delete duplicates	352	FBM Deep Drilling	492
Depth Cut Order	426	FBM Drill	487
Disjoint Body	213, 225	FBM Drill Circle Mill Parameters	495, 496
Draft	213	FBM Drill Hole Detection	489
Draft Faces	215, 241	FBM Drill Hole Mapping	491
Drafting	82, 126	FBM Drill Linking Parameters	500
Drawing Using Cplane	177	FBM Drill Setup	487
Drill	266	FBM Drill Tools	499
Drill Cut Parameters	292	FBM Features	486, 502
Drill Cycles	298	FBM Helix Bore Parameters	484, 498
Drill Linking Parameters	294	FBM Hole Milling	495
Drill Location Selection	286	FBM Hole Milling Parameters	480
Drill Point Manager	321	FBM Linking Parameters	485
Drill Tip Compensation	295	FBM Mill	469
Drill Tool Selection	267	FBM Pre Drilling	494
Drilling Selection	284	FBM Setup	469
Drop Down Menu	41	FBM Spot Drilling	493
Dynamic Contour - Contour Wall	459	FBM Wall Finish	479
Dynamic Contour - Cut Parameters	458	Feature	213

Feature Based Machining	467
Feature Detection	471
File Compression	36
File Name	28
File Save	29
File Tracking	534
Fillet	106, 213
Fillet Ribbon Bar	106
Find Features	241
Finish Step	426
Finish Tools	474
Fit Screen	49
Function Prompt	41

G

G-Code	14, 15, 19
General Selection Toolbar	128
Geometry Nesting	152
Getting Help	21
Graphic Area	41
Group/Results	156
Groups	45, 54

H

Helix	119
HMC Setups	509
Holder Page	291
Hole-Axis	242

I

Incremental coordinates	75
Index Toolpaths	510
Island Facing Pockets	408
Island taper angle	426

K

Keep Tool Down	426
Key Mapping	64
Keyboard command	41
Keyboard Commands	52

L

Lathe Radius Lathe Diameter	182
Layout	215
Letter Options	123
Letters	119, 122
Level	45
Line	82
Line Ribbon Bars	88
Line Style	44
Line Width	44, 47
Lines	84

List Options	28
Loft	213, 214, 241
Look In	28

M

Machine Group Properties - Files	280
Machine Group Properties - Stock Setup	282
Machine Group Properties - Tool Settings	281
Machine type	268
Machining regions	441
Machining Splines	387
Mastercam Code Expert	332
Mastercam Coordinate Systems	80, 172
Mastercam File	14, 15, 19
Mastercam File Functions	23
Mastercam Geometry	81
Mastercam Programming Process	264
Mastercam Workspace	40
Max Rough Step	426
Mirror	152, 159
Mitered Corners	227
Model Prep	241
More Point Function	114
Mouse Cursor	41
Mouse View Commands	51
Move	246
Move To Origin	196
Move to Origin	152
Multiple Machining Region	441

N

Named Views	49
Native File Types	34
Native Files	33
Neutral File Translations	31
New	24

O

Offset	166
Offset Contour	167
Offset Contour Example	170
Open	24
Open chain extension to stock	442
Open External	24
Open Pockets	413
Origin	41
Overflow	227
Overlapping entities	352

P

Parallel lines	86
----------------------	----

Pattern	214	Recognition	213
Peck Drill (G83)	306	Rectangle	119
Peel Mill - Cut Parameters	461	Rectangle Options	121
Peel Mill Toolpath	460	Rectangle Ribbon Bar	121
Perpendicular lines	85	Rectangles	120
Plane Indexing	510, 511	Rectangular Array	152, 195
Plane Manager	273	Red X	28
Plane Selection	43	Remove Faces	242
Planes = Gview	182	Retrieving Numeric Data	98
Planes = WCS	182	Revolve	213, 214, 241
Planes By Geometry	182	Right Hand Rule	190, 507
Pocket Chaining	401	Right Mouse Button	28
Pocket Cut Parameters	402	RMB	41
Pocket Cutting Methods	416	Roll	152
Pocket Depth Cuts	426	Rotary Feed Rates	520
Pocket Entry Motion	418	Rotate	152, 162
Pocket Finishing	424	Rotate Direction	163
Pocket Helical Entry	422	Rotate Toolpath	514
Pocket Lead In/Out	425	Rough End Mill	399
Pocket Ramp Entry	420	Roughing Tools	473
Pocket Remachining	410	S	
Pocket Roughing Parameters	415	Save	24
Pocket Tool Selection	398	Save As	24
Pocket Toolpath	398	Save Some	24
Pocket Tools	398	Scale	152, 164
Pocket Types	403	Select Library Tool	357
Pockets with Islands	404	Select Only	135
Point	82, 109	Selection Override Options	130
Point Options	111	Sheet Solid	213
Point Ribbon Bar	112	Shell	213
Point Style	44, 47	Simple Drill (G81)	300
Polar Coordinates	76	Simulate the toolpath in Verify	328
Polygon	119	Slot Mill Toolpath	434
Post Process	331	Solid Add Boss	224
Post Processo	14, 15, 19	Solid Body	212
Pre-Defined WCS	271	Solid Chamfer	229
Print	24	Solid Chamfer Dialog Boxes	230
Print Preview	24	Solid Chamfer Edge Example	231
Project	152, 193	Solid Constant Radius Fillet Dialog Box	227
Propagate Along Tangencies	227	Solid Cut	224
Push-Pull	241, 242	Solid Distance And Angle Chamfer	241
Push-Pull Example	244	Solid Face-Face Fillet	241
R		Solid Fillet	241
Radius	227	Solid Fillet Floor Example	228
Ramp Contours	364	Solid One-Distance Chamfer	215
Rapid Height G98/G99	312	Solid Options	213
Reading CAD Data	30	Solid Shell	215
Reamer	266	Solid Terms	212

Solid Two-Distance Chamfer	241
Solid-Draft	220
Solids	81
Solids Fillet	226
Solids Manager	232
Solids Menu	214
Spiral	119
Spline	82
Spot Drill	266
Standard Pockets	403
Starting Mastercam	20
Stretch	152
Subprogram	426
Surfaces	81
Sweep	213, 214, 241
System Defined Cplanes	174
T	
Tangent lines	85
Tap	266
Taper angle	426
Tapered Wall	426
Tapping (G84)	308
Thicken	215, 241
Thin Wall	213
Toggle Operations Manager	54
Tool Page	289
Toolpath Controls	317
Toolpath Display Colors	543
Toolpath Properties	279
Toolpath Type Page	288
Toolpaths Manager	314
Toolpaths/Solids Manager	41
Tracking Options	536
Transform Commands	152
Transform Toolpath	514
Translate	152, 153
Translate 3D	152, 191
Translate 3D Example	192

Translation Problems	35
Tree View List	287
Trim	215, 241
Trim/Break	115
Trim/Break Options	115
Trim/Break Ribbon Bar	116
Trimming	117
U	
Undo/Redo	98
Use Island Depths	426
User Defined Cplanes	180
User-Defined WCS	272, 276
Using WCS	274
Using Wear Compensation	376
V	
View Summary Report	542
View Toolbars	49
Viewport Axes	41
Views	49
VMC Setups	508
W	
Watertight	213
Wear Compensation	375
Window Control	41
Wireframe	81
Wireframe Geometry	82
Wireframe Geometry Options	83
Work Coordinate System (WCS)	270
X	
Xform Roll	524
Xform Unroll	526
Z	
Zip2 Go	37
Zoom Selected	49
Zoom Window	49

Did you like this book?

Discover other Mastercam Training Solutions at
www.eMastercam.com

eMastercam has the largest Mastercam community on the web with over 56,000 members and over a million forum posts. While there, be sure to join the conversation!



**Training
Solutions
Offered**

Training Tutorials • Instructor Materials • Professional Courseware
Handbooks • Video Training • Online Video eCourses
Sitewide Subscription • Site Licenses • Online Community

Mastercam® @Courses

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



GETTING STARTED



MILL ESSENTIALS



MILL ADVANCED



LATHE



MULTIAXIS ESSENTIALS



MASTERCAM FOR SOLIDWORKS



LOG IN & LEARN TODAY!

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

- NEW – FREE Getting Started eCourse
- Online previews with table of contents including the time it takes to complete each session.
- Mastercam 2019 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**

Only \$149 each!



www.eMastercam.com/ecourses

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.

\$39.99
/1 month

\$129.99
/4 months
(save \$29.97)

BEST VALUE:

\$359.99
/12 months
(save \$119.89)



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

Post processors that are requested worldwide.



We speak your machine's language.

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 that is continually developed and expanded to include current machines and controllers. 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.

www.inhousesolutions.com/posts



Mastercam 2019



Demo Software Download Instructions Included

FREE eBooks

Download today
www.eMastercam.com/freebooks

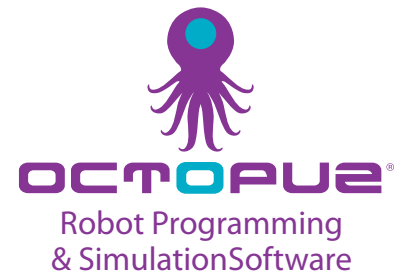
DOES YOUR SCHOOL HAVE ROBOTS? YOU SHOULD CHECK OUT OCTOPUZ.

As easy to teach as it is to learn, OCTOPUZ is an offline robot programming and simulation software used world-wide in industry. Prepare your students for the real world of robotics, with OCTOPUZ they will gain the necessary skills to not only be hired, but to excel.



DESIGN. PROGRAM. LEARN.

www.OCTOPUZ.com/for-education



With OCTOPUZ you can:

1

Create your
custom cell.

2

Program your
application.

3

Verify using
simulation.

4

Generate and
post code to ALL your
robot brands.

Why use OCTOPUZ?



Get the most out of lab
time, program offline and
run online on the robot.



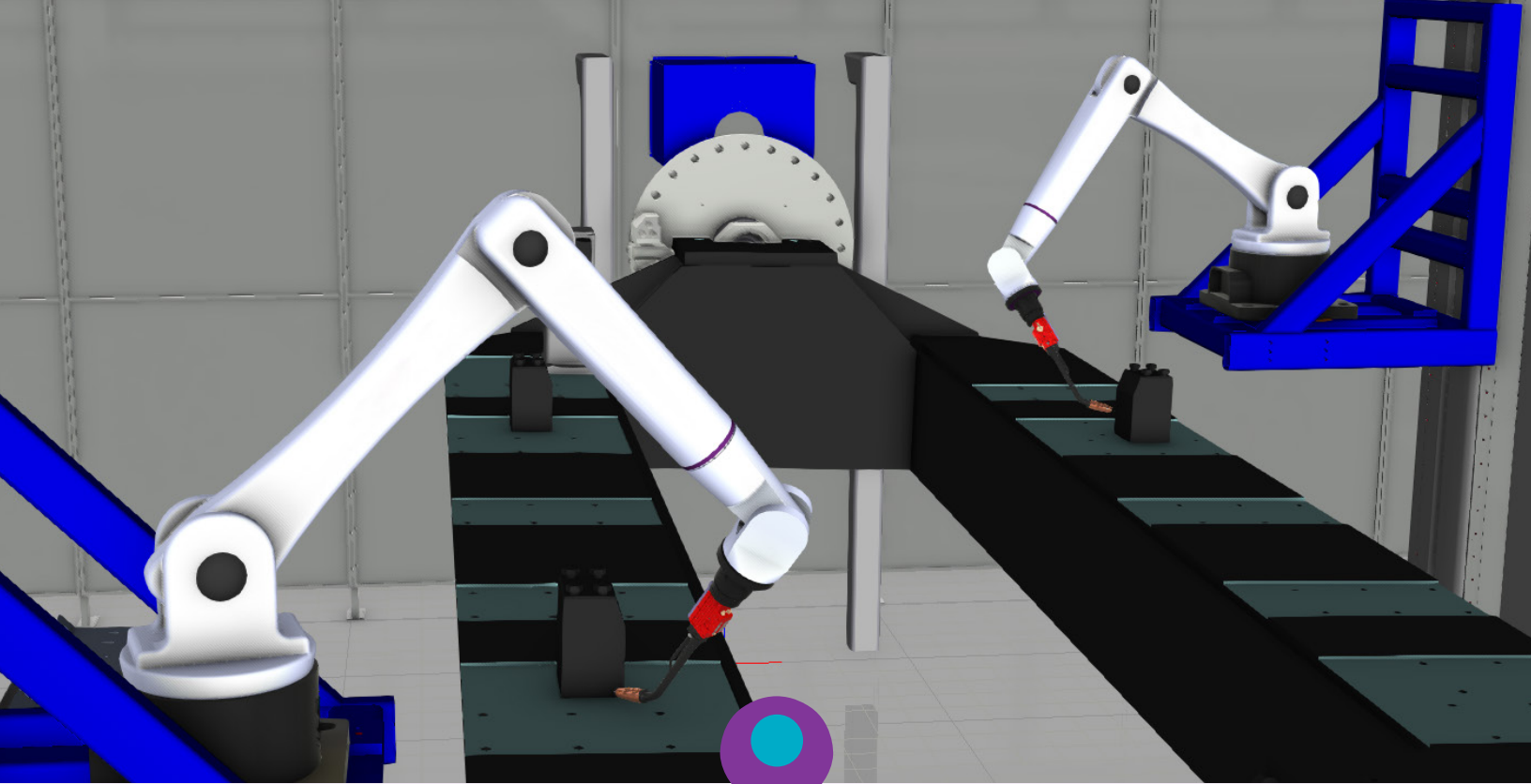
Protect your investment!
OCTOPUZ can detect
collisions offline.



Expose students to
components and robots
not available in your lab.



Bring 3D printing to life.



OCTOPUZ®

Robot Programming & Simulation Software

The robot brands YOU use.
As many robots as YOU need.
Together, in YOUR cell... at the same time

Why choose OCTOPUZ

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

Full customization
Application versatility
CAM interface
Complex kinematic systems

CONTACT US

SET UP YOUR FREE DEMO

www.OCTOPUZ.com