

Mastercam 2019

IMPERIAL – HANDBOOK VOLUME 3



Demo Software Download Instructions Included

MULTIAXIS & MACHINING

Mastercam[®] 2019

Handbook Volume 3

To order more books:

Call 1-800-529-5517 or

Visit www.emastercam.com or

Contact your Mastercam dealer

Mastercam 2019 Handbook Volume 3

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

Software: Mastercam 2019

Author: In-House Solutions

ISBN: 978-1-77146-783-4

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, 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; Windows 7 and Windows 8 are registered trademarks of Microsoft Corporation.

This document requires Mastercam Mill Essentials and Solids for Chapter 5.

Table of Contents

Legend	1
---------------------	----------

Chapters:

Introduction To Multiaxis Machining	3
Advanced Multiaxis Common Parameters	43
Multiaxis Curve Toolpaths	107
Multiaxis Swarf Milling Toolpaths	145
Multiaxis Parallel Toolpaths	159
Multiaxis Drill Toolpaths	167
Multiaxis Along Curve Toolpaths	185
Multiaxis Morph Toolpaths	191
Multiaxis Flow Toolpaths	199
Multiaxis Multisurface Toolpaths	215
Multiaxis Port Toolpaths	227
Multiaxis Triangular Mesh Toolpaths	239
Multiaxis Swarf Toolpaths	263
Multiaxis Rotary Toolpaths	281
Multiaxis Project Curve Toolpaths	297
Multiaxis Roughing Toolpaths	303
Multiaxis Circle Mill Toolpaths	309
Multiaxis Port Expert & Blade Expert Toolpaths	315

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.

An accompanying CD ROM contains files needed to complete all exercises in this handbook to demonstrate certain topics and examples.

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

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/Panel** is a window that opens to allow input of information and setting of defaults.
- **Drop Down/Fly Out 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.

1 Introduction To Multiaxis Machining

OBJECTIVES

This chapter introduces the basic concepts and principles of multiaxis machining.

Upon completion of this chapter, you should be able to do the following:

- Identify common Multiaxis machine tool configurations.
- Understand the difference between Multiaxis position and Simultaneous Multiaxis toolpaths.
- Understand the different types of Multiaxis toolpaths, and the capabilities and advantages they provide over conventional toolpaths.
- Understand basic setup and programming of Multiaxis machine tools, including collision avoidance and machine envelope limitations.
- Use the Right Hand Rule to determine the correct direction sign of an axis of rotation.
- Know how to properly select and set the NC program datum and tool length offsets for different multi-axis machine tool configurations.

INTRODUCTION

There are many different Multiaxis machine tool configurations. Most control three linear axes (XYZ) plus two rotational axes. The rotational axes are some combination of the ABC axes, depending on the machine design.

Multiaxis machines are indispensable for manufacturing parts common in the aerospace, mold, and die industries.

This chapter presents the basic concepts and knowledge you need to understand Multiaxis machines and toolpaths. By the time you finish this book, you will have a good understanding of Multiaxis machines. You will also know how to use Mastercam Multiaxis to program these interesting and versatile machines.

MULTIAXIS TOOLPATHS

Multiaxis toolpaths can be classified into two broad categories:

- Axis Positioning.
- Simultaneous Multiaxis.

AXIS POSITIONING

Axis Positioning involves moving one or more rotary axes to orient a feature towards the spindle. The rotary axes are then locked in position and only the linear axes are moved simultaneously to machine the feature.

Since three axes move and two are fixed, this type of motion is referred to as "3 plus 2" machining. With the right post processor, any Mastercam mill toolpath will output 3+2 code. Just set the WCS tool plane to any plane other than Top.

Mastercam Axis Positioning Multiaxis toolpaths include:

- Multiaxis Drill.
- Multiaxis Circle Mill.
- All the toolpaths that can set the **Tool Axis Control** to a **Plane**.

SIMULTANEOUS MULTIAXIS MACHINING

Simultaneous Multiaxis Machining controls up to five axes at once: three linear and one or two rotary.

Mastercam Simultaneous Multiaxis toolpaths include:

- Curve.
- Swarf Milling.
- Parallel
- Along Curve.
- Morph.
- Flow.
- Multisurface.
- Port.
- Triangular Mesh.
- Swarf.
- Rotary.
- Project Curves.
- Roughing.
- Port Expert.
- Blade Expert.
- Rotary Advanced.

Most of the 5 axis toolpaths can be set to output in the 3 axis format and use an existing Plane.

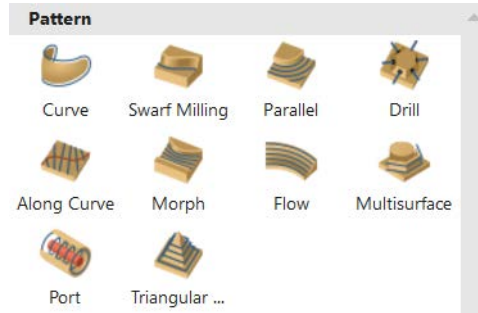
The toolpaths are grouped in two families: **Pattern** and **Application**.

The **Pattern** toolpaths are general toolpaths that can be used to machine any kind of shapes. The **Application** toolpaths are designed to machine specific parts or features, automating some of the processes.

- **From the Pattern family toolpaths:**
 - Curve.
 - Swarf Milling.
 - Parallel
 - Drill.
 - Along Curve.
 - Morph.
 - Flow.
 - Multisurface.
 - Port.
 - Triangular Mesh.
- **From the Application family toolpaths:**
 - Swarf.
 - Rotary.
 - Project Curves.
 - Roughing.
 - Deburr.
 - Circle Mill.
 - Port Expert.
 - Blade Expert.
 - Rotary Advanced.

The Multiaxis toolpaths will be described in this book based on how they are grouped in these two families.

PATTERN FAMILY TOOLPATHS



Curve

Curve is a 5-axis toolpath that moves the tool tip along a curve while controlling the tool axis.

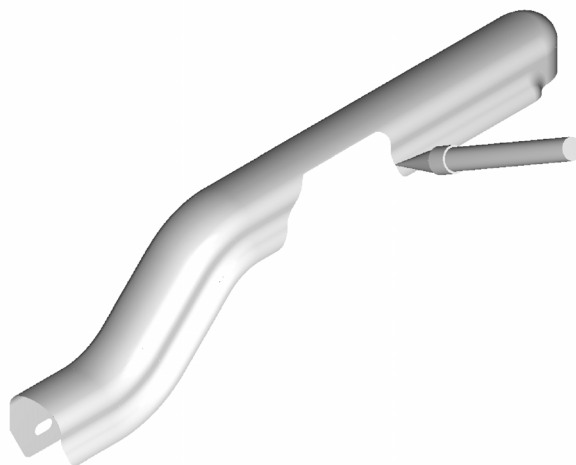
Mastercam has many methods for controlling the tool axes. The most common is to align the tool axis with the surface normal at each tool position as the tool moves along the curve.

The tool can also be made to tilt forward or backward in relation to the cut direction. This is called a lead/lag angle, and it is used to help improve machining efficiency in some cases.

It can also be made to tilt to the left or right in relation to the cut direction. This is called a side tilt angle and is used to accommodate machine rotary limits or for collision avoidance.

In the illustration below, a 5-axis laser is trimming a formed sheet metal part as shown in [Figure 1](#). The tip of the tool moves along the trim curve while two rotary axes keep the tool axis normal to the part surface.

Figure 1



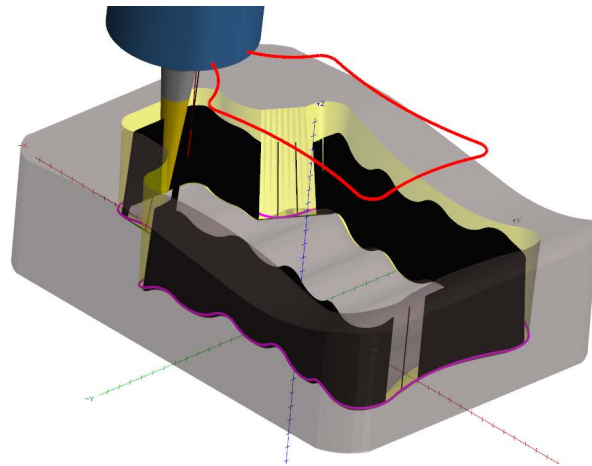
Swarf Milling

Swarf Milling (Side Wall Axial Relief Feed) toolpaths allow you to machine wall surfaces with the side of the tool. The advantage of this 5-axis toolpath is that you can machine the drive surface with a single cut using the whole flute length of the tool.

The swarf milling toolpath will keep the tool axis aligned between 2 points on the upper and lower curve which means that the surface to be machined has to be straight between the points. The system tries to get the upper and lower curves from the surfaces. If this does not work then the user has to provide the upper and lower curve separately.

The following graphic shows a **Swarf Milling** toolpath as shown in [Figure 2](#).

Figure 2



Parallel

The **Parallel** toolpath pattern can be set parallel to **Curves**, **Surfaces** or aligned at an **Angle**.

Parallel To Curves

The **Parallel to Curves** pattern will align the cut direction with a leading curve. Adjacent cuts are parallel to each other. It is important to note that the cuts will not simply be copied next to each other. Every new cut is created at an offset from the previous cut.

The curve must be located exactly on the surface edge, so the best curve would be the edge itself. This is very important for toolpath generation. If you don't have a proper curve, an incorrect toolpath can be generated.

The following graphic shows a **Parallel to Curves** pattern used to finish the turbine blade fillets as shown in [Figure 3](#).

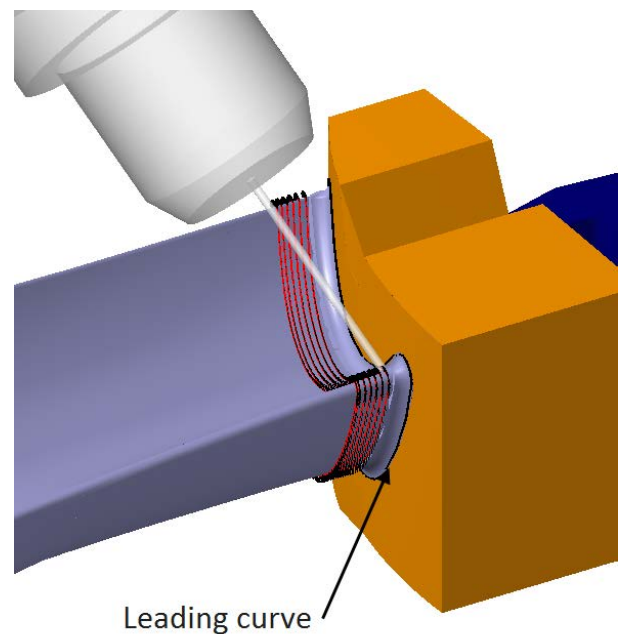


Figure 3

Parallel To Surfaces

The **Parallel To Surfaces** pattern generates cuts on the drive surface parallel to a check surface. There are special options for the edge between the check surface and the drive surface. You can define a margin to define the exact position where the tool is located along the edge, positioned as close as possible to both faces.

The following graphic shows a **Parallel to Surfaces** pattern used to finish the blade of an impeller as shown in [Figure 4](#).

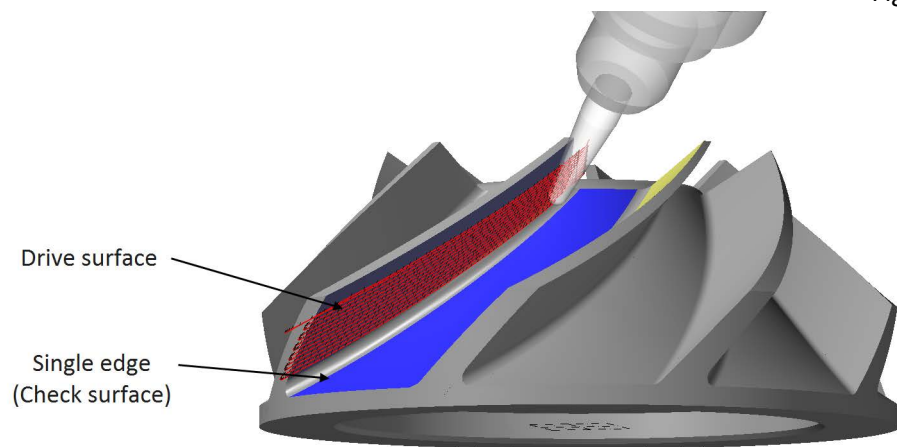


Figure 4

Parallel To Angle

The **Parallel To Angle** pattern lets you create tool motions with cuts that are parallel to each other. The direction of the cuts is defined by the two angles: the **Machining angle in X,Y** and the **Machining angle in Z**.

The following graphic shows a **Parallel To Angle** pattern used to finish a cam as shown in [Figure 5](#).

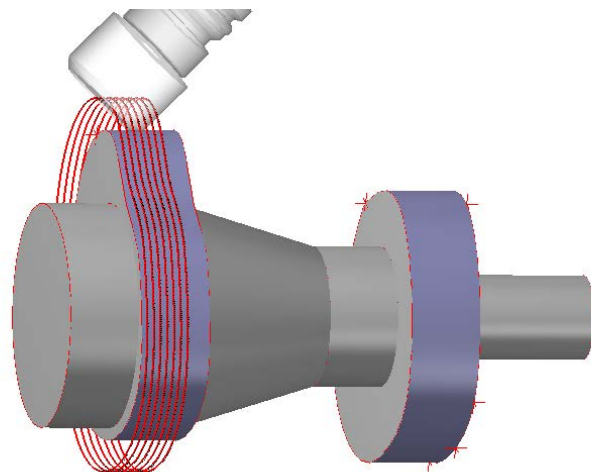
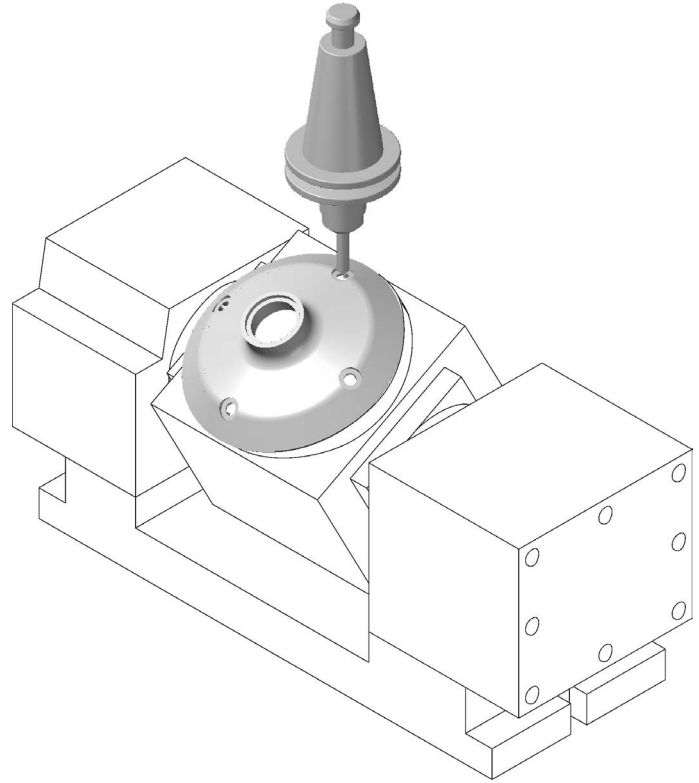


Figure 5

Drill

The **Drill** 5-axis toolpath controlling a trunnion table is an example of a Multiaxis positioning toolpath as shown in [Figure 6](#). The part is oriented towards the spindle using the rotary axes. The XYZ axes are used to position the tool over the hole and drill it.

Figure 6

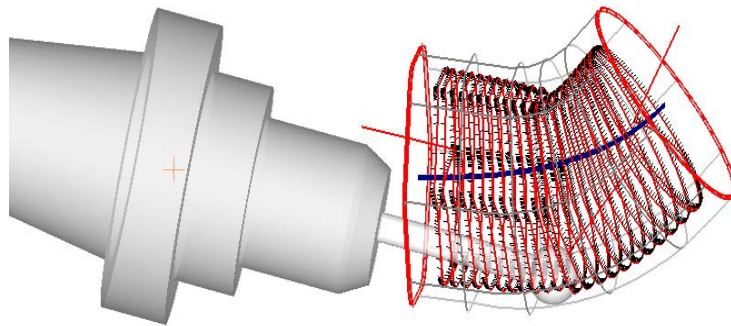


Along Curve

Along Curve generates the toolpaths orthogonal to a leading curve. This means that when your selected curve is not a straight line, the cuts are not parallel to each other. The distance between two neighboring cuts (at the intersection of the curve and toolpath) is the maximum stepover.

The following graphic shows an **Along Curve** toolpath used to finish an enclosed surface as shown in [Figure 7](#).

Figure 7



Morph

The **Morph** toolpath allows you to choose between two patterns: From Curves and From Surfaces.

Morph - Pattern From Curves

Morph between 2 curves will create a morphed toolpath between two leading curves. A "morphed" toolpath is one that is approximated between the tilt curves and evenly spread over the surface. This toolpath will ensure a better finish with less retraction moves.

The following graphic shows a **Morph between 2 curves** toolpath used to finish a turbine blade's surfaces as shown in [Figure 8](#).

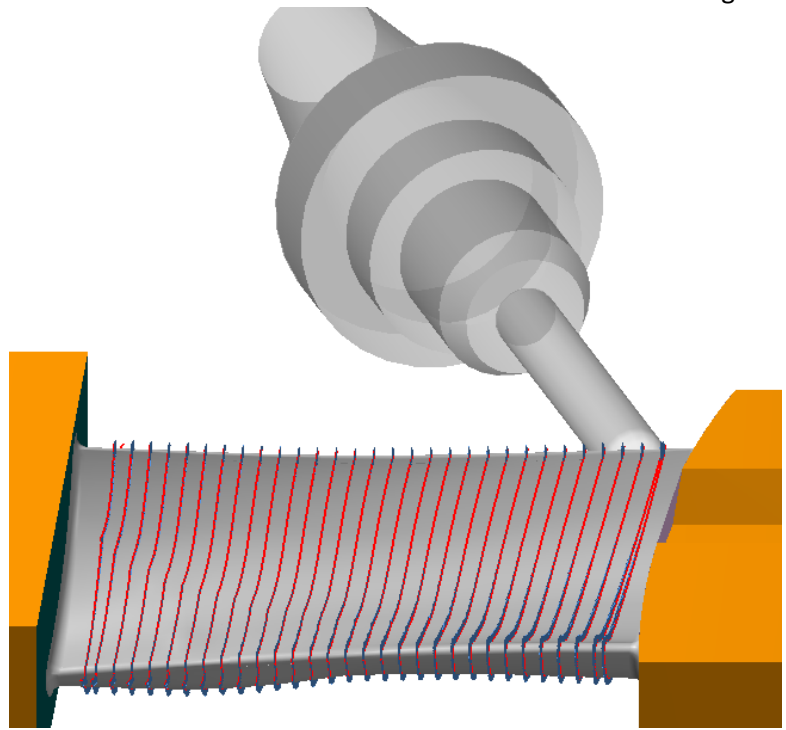


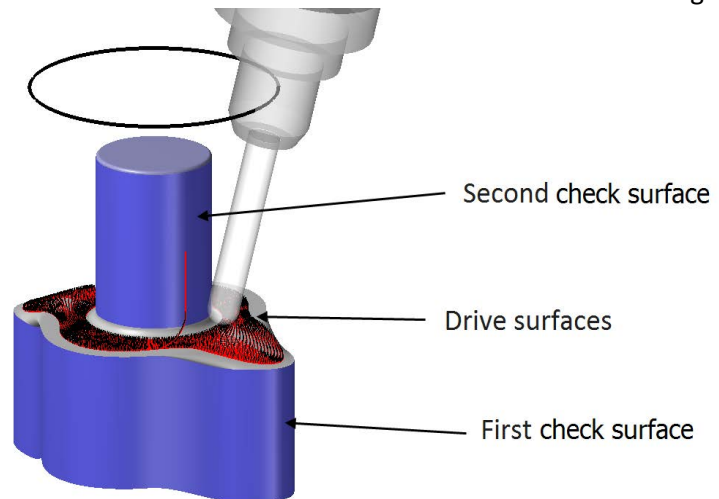
Figure 8

Morph - Pattern From Surfaces

Morph between 2 surfaces toolpaths create a morphed toolpath on the drive surfaces. The drive surface is enclosed by two check surfaces. A "morphed" toolpath is one that is approximated between the check surfaces and evenly spread over the drive surface. The main advantage is the possibility to compensate the tool to the drive surface and check surface in the left and right corner of the workpiece. All you need to do is set the margin, or distance between the tool center and the surfaces, equal to the tool radius.

The following graphic shows a **Morph between 2 surfaces** toolpath as shown in [Figure 9](#).

Figure 9



6

Multiaxis Drill Toolpaths

OBJECTIVES

The purpose of this chapter is to introduce the Multiaxis **Drill** toolpath.

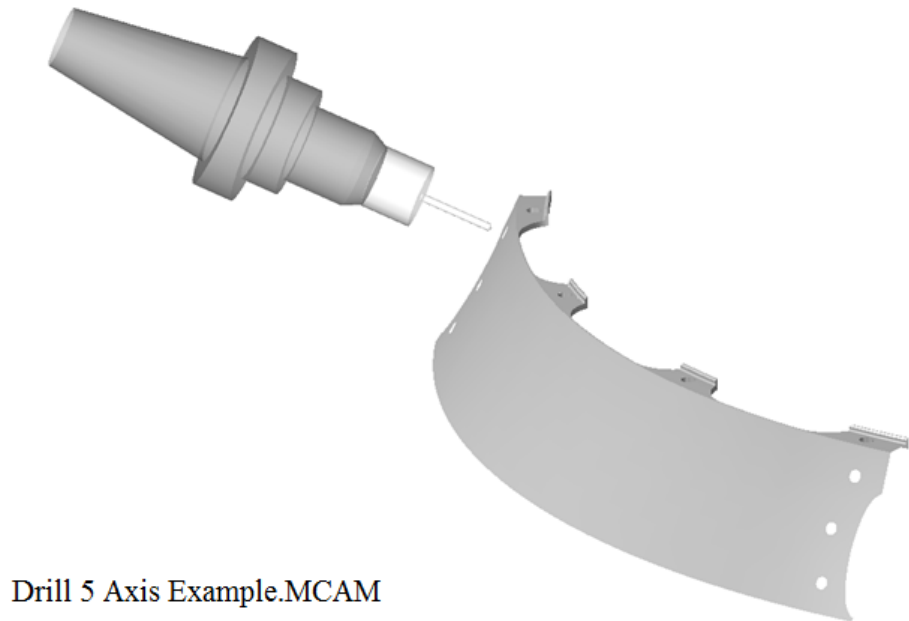
Upon completion of this chapter, you should be able to do the following:

- Understand concepts related to Multiaxis Drill toolpaths.
- Know how to select drill locations using points or points and lines.
- Know how to control the tool axis for drilling operations.

INTRODUCTION

Multiaxis drilling operations involve moving the tool linearly once the tool is positioned over the hole with the desired tool axis orientation as shown in [Figure 1](#).

Figure 1



Motion between holes, where up to five machine axes can be moving at once and all at different rates, provides many opportunities for problems.

Exercise extreme caution between positioning moves as it is very easy to crash the tool or machine head or over-travel a linear or rotary axis. Until a program has been proven, set generous clearance values and use the rapid and feed over-ride. These values can always be reduced once the program has been proven.

Once positioned and oriented, drill motion looks and acts similar to three axes drilling, but there are differences. Most multiaxis machines with rotation in the head usually do not support drill cycles such as G81 or G83. All drilling motion is output as G0/G1 moves. Therefore, the code can be much longer for multiaxis drilling.

Multiaxis machines are often less rigid than three axis machines, so be prepared to slow cutting speeds and feeds accordingly.

Another concern is economy of motion, in particular trying to minimize the number of unwinds the head must make for machines with limited rotary travel.

MULTIAXIS DRILL TOOLPATH

To start the toolpath, select the **TOOLPATHS** tab and, from the **Multiaxis** group, select the **Drill** icon as shown.



Tool

The **Tool** page allows you to select a tool, set the feeds and speeds, enter a comment about the operation, and set other general toolpath parameters. This page is identical to that used throughout Mastercam 2D toolpaths as shown in [Figure 2](#). For questions regarding parameters on this page, consult the online help (Alt + H) or refer to the **Mastercam Handbook Volume 1** or **Volume 2**.

Figure 2

The screenshot displays the Mastercam Tool dialog box. On the left, a table lists available tools:

#	Assembly...	Tool Name	Holder N...	Dia.	Cor. r...	L
1	--	1/2 SPOT...	--	0.5	0.0	4
2	--	1/2 DRILL	--	0.5	0.0	4

Below the table is a scroll bar and the text "Right-click for options". At the bottom left, there is a "Select library tool..." button, a "Filter Active" checkbox, and a "Filter..." button. A "To batch" checkbox is located at the bottom left of the dialog.

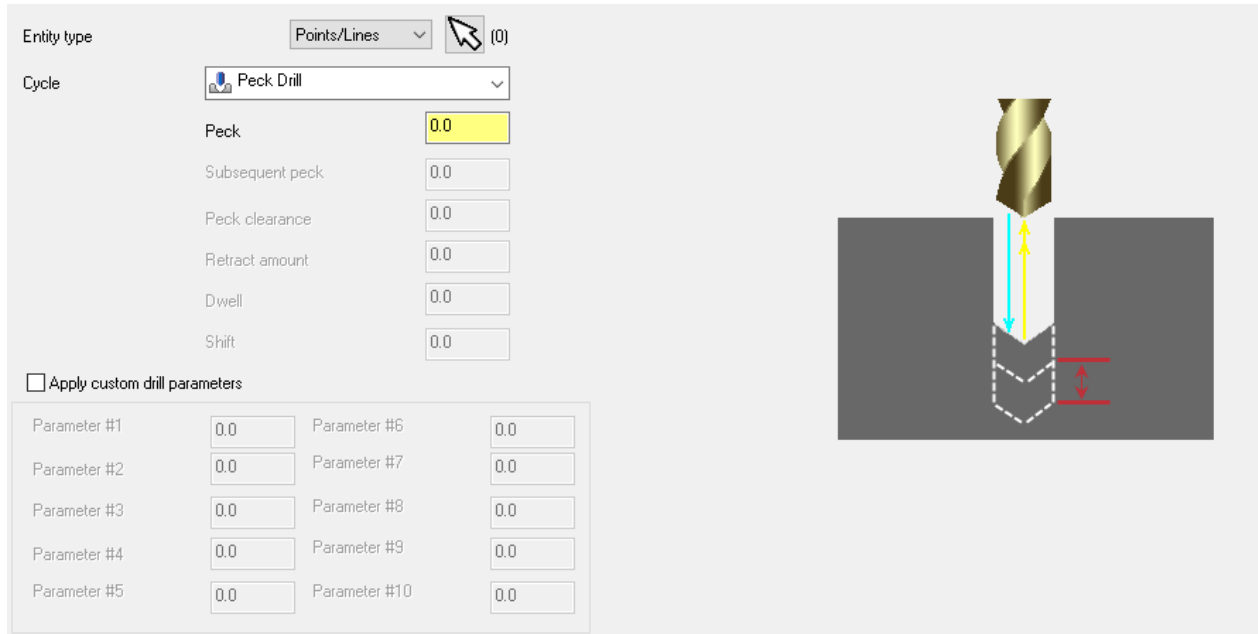
On the right side, various parameters are set:

- Tool diameter: 0.5
- Corner radius: 0.0
- Tool name: 1/2 DRILL
- Tool #: 2
- Length offset: 2
- Head #: 0
- Diameter offset: 2
- Spindle direction: CW (dropdown)
- Feed rate: 4.2784
- Spindle speed: 534
- FPT: 0.004
- SFM: 69.8953
- Plunge rate: 4.2784
- Retract rate: 4.2784
- Force tool change: ☐
- Rapid Retract: ☐
- Comment: Drill the 1/2 holes.

Cut Pattern

Cut Pattern settings determine the geometry the tool follows and how it travels along that geometry to create 3-axis, 4-axis or 5-axis drilling output to machine the holes in the part as shown in [Figure 3](#).

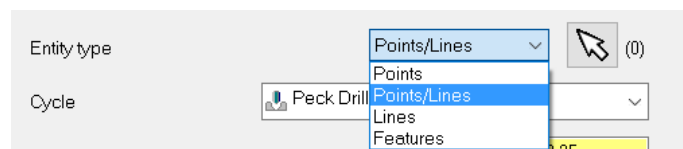
Figure 3



Item	Definition
Entity type	Determines if Points (Points and a surface), Points/Lines (Points at end of normal lines), Lines or Features are used to define the tool axis.
Cycle	Allows you to choose between several pre-defined (canned) drilling cycles and numerous custom cycles. Selecting a drill cycle determines the drill toolpath parameters. This option is identical to that used in 2D drilling. To learn more about any setting on this page refer to the online help or the Mastercam Handbook Volume 1 .

Entity Type

The **Entity type** allows you to select the geometry used to define the hole from **Points**, **Points/Lines**, **Lines** and **Features**.



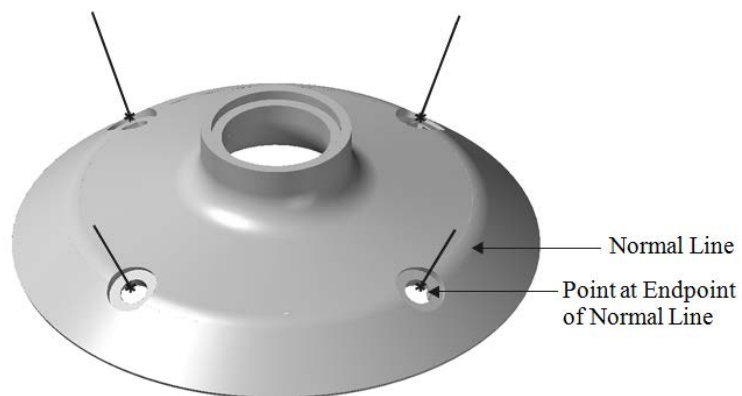
Item	Definition
Points	You can select the points located at the center of the holes. This allows the Tool Axis Control to be set parallel to a line, normal to a surface, or normal to a plane.
Points/Lines	You need to have the points and the lines starting from the points already created. This allows you to select the points and determines the tool axis vector based on the lines.
Lines	Causes all holes to be aligned with a line. This temporarily closes the Drill dialog box and prompts you to select a line on the part model. After picking the line, the system displays an arrow on the line, indicating the direction from which the tool will approach. This arrow should point towards the machine spindle.
Features	Selects the holes in solid bodies using Mastercam's Hole Axis functionality to display vectors and points. The hole geometry is associative, and the drilling depth can be calculated to include the height of the holes.

Points selects only the drill position. If this method is used, the **Tool Axis Control** settings become active and one of these options must be used to set the tool axis.

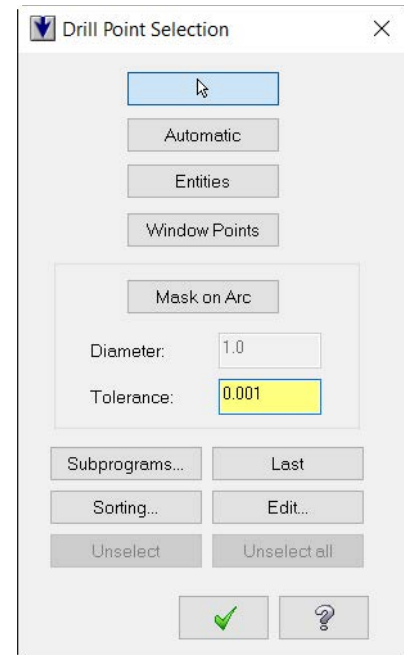
After selecting the **Selection** button for **Points** or **Points/Lines**, the **Drill Point Selection** dialog box opens. This dialog box is identical to that used to select holes for 2D drilling. For more information about any of these settings, refer to the online help (Alt + H) or the **Mastercam Handbook Volume 1, Chapter 6: Drill Toolpaths**.

The **Points/Lines** option uses points to define the drill location and lines to define the tool axis orientation. The point is located at the starting point of the hole. One endpoint of the line touches this point. The line acts as a normal line, pointing away from the hole towards the machine spindle as shown in [Figure 4](#).

Figure 4



To use this option, click on the **Points/Lines Select** button and select the drill points. This opens the **Drill Point Selection** dialog box.



Just select the points. Mastercam automatically looks for lines that share an endpoint with the selected points, and chooses them as normal lines. Only one tool axis control line is allowed per point.

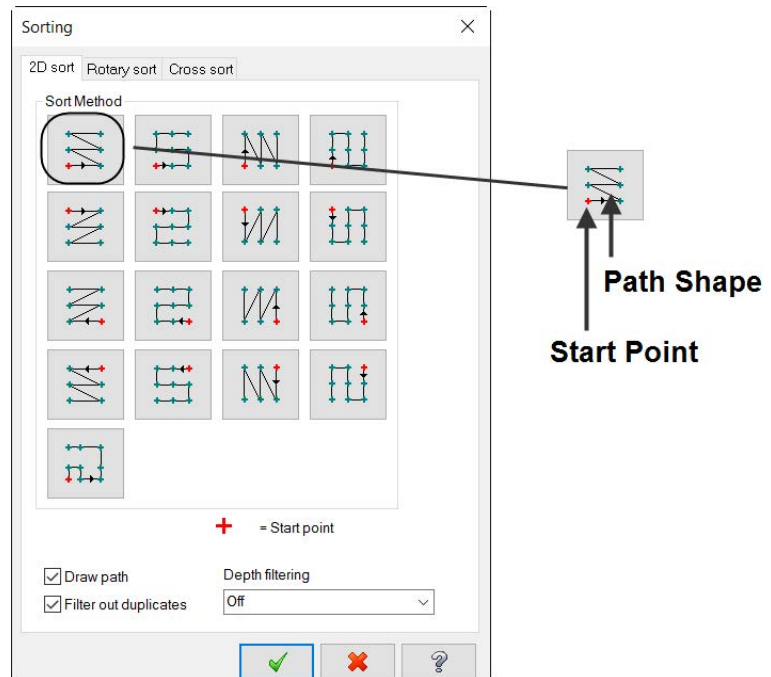
Item	Definition
Select drill point position	Individually select points using the cursor. This function also works on the centers of solid holes.
Automatic	Pick a string of geometric points by selecting the first, second and last point.
Entities	Select entities on the screen. This function picks points, the endpoints of lines and splines, and the centers of selected arcs.
Window Points	Window selects geometric points.
Mask on Arc	<p>Pick arcs by matching a selected arc diameter.</p> <ul style="list-style-type: none"> • Diameter shows the value to match. This is set by clicking on an arc in the graphic area. • Tolerance sets the arc matching tolerance.
Sorting	Sorting finds an efficient path through an array of selected points.
Last	Reuses the last set of geometry selected for a toolpath operation. This is useful for reselecting holes for multiple operations.
Edit	Edit selected drill points.

Sorting Options

The **Sorting** options automatically find an efficient path through an array of points. These options are modal; once set, they remain in effect for programming other drill operations. An array of icons lets you select different search paths through the selected points.

2D sort is used to select zigzag movements between the holes. The red point indicates the location of the start point. The black lines show the path between the drill locations as shown in [Figure 5](#).

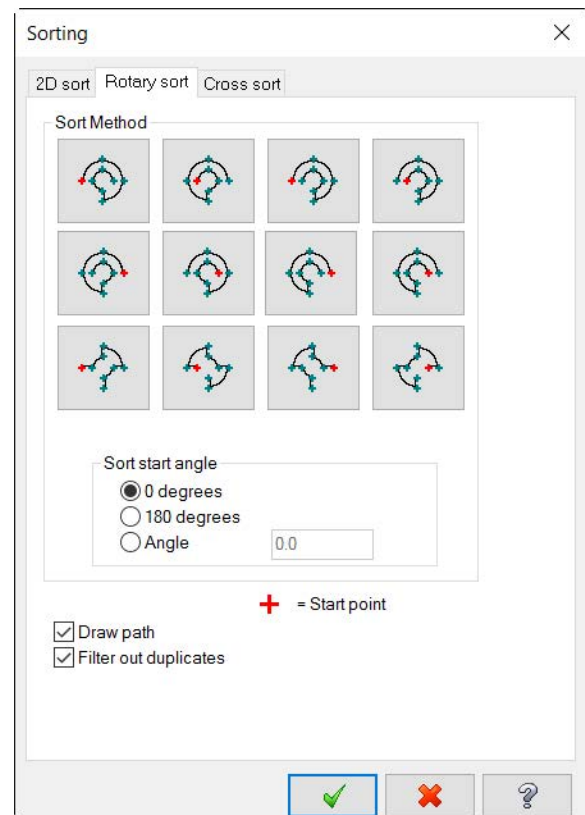
Figure 5



Item	Definition
Depth Filtering	When active, depth filtering is used to control the lowest or highest Z depth of the entity for the drill point as shown in Figure 5 .

Rotary sort is used for holes arranged in a circular pattern as shown in [Figure 6](#).

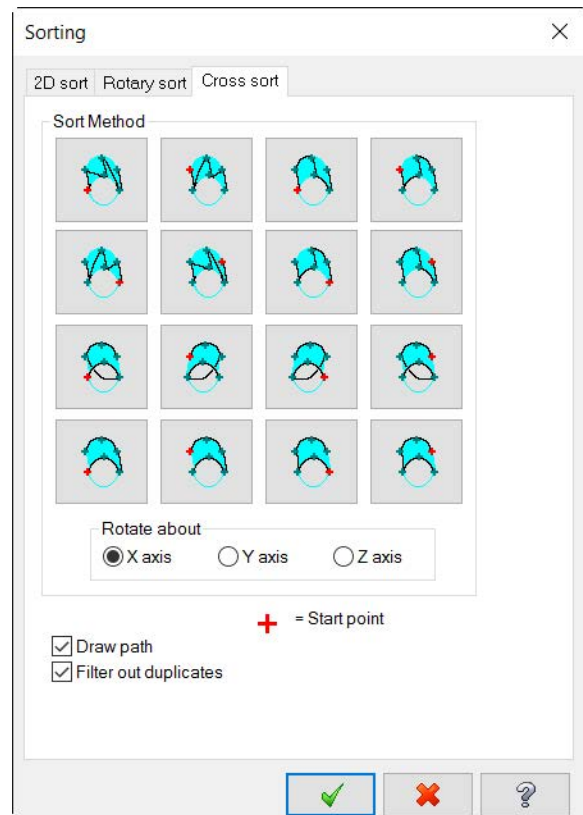
Figure 6



Item	Definition
Sort start angle	Sets where the beginning of the hole pattern should be located.

Cross sort is used to optimize drill holes in a rotary pattern and is used for drilling using a rotary 4th axis as shown in [Figure 7](#).

Figure 7



Item	Definition
Rotate about	Sets the axis the rotary sort rotates about.

Tool Axis Control

The **Tool Axis Control** settings determine the tool's orientation in relation to the geometry being machined as shown in [Figure 8](#).

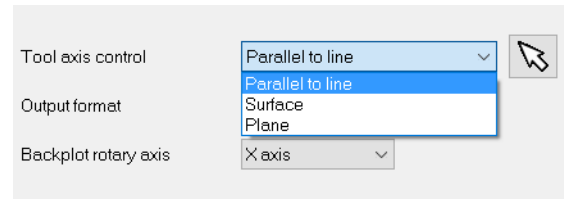
Figure 8



Item	Definition
Tool axis control	The tool axis control group sets how the tool axis orientation is controlled.
Output format	Determines if the output code is 3 axis only (no rotations), 4 axis (one rotary axis), or full 5 axis (two rotary axes).
Backplot rotary axis	Allows you to select the axis to be used as the rotary axis. You should set the rotary axis to match your machines capabilities.

Tool Axis Control Options

If the **Points** entity type is selected, a **Tool axis control** option must be selected.



Item	Definition
Parallel to line	Causes all holes to be aligned with a line. To use this method, click on the Parallel to line Select button. This temporarily closes the Drill dialog box and prompts you to select a line on the part model. After picking the line, the system displays an arrow on the line, indicating the direction from which the tool will approach. The arrow should point towards the machine spindle.
Surface	Keeps the tool normal to a selected surface.
Plane	Keeps the tool normal to a selected plane.
Points/Lines	This is displayed when, in the Entity type, you choose Points/Lines.

Numerics

2d Containment	48
3 - Axis Output Format	128
4 - Axis Output Format	128
4-axis angle limits	274
5-Axis Output Format	129

A

Air Move Safety Distance	100
Angle From Curve	77
Angle increment	120, 131, 273
Angle Limits	218
Angle Range	55
Angle range	48
Area Type	48
AutoFind	218
Automatic	172
Axis dampening length	287
Axis Limits	132
Axis Positioning	4

B

Blade expert	319
Boundary	209
box cut pattern	222
by Branch	272
by Entity	272
by Node	272
by Point	272

C

Center Axis	218
Chain	125
Change Normals	202
Check Surface	278
Check surfaces	133
Circle Mill	20, 21
Closed walls	266
Closest Point	76
Closest point on chain	126
Collision Control	156, 276
Collision Issues	25
Compensation direction	111, 146, 265
Compensation surfaces	211
Compensation Type	111, 146, 265
Creating Wall Chains	270
Cross sort	175
Curve	6, 107
Curve following method	116
Curve type	111

Curve Type 3D Curves	112
Curve Type Surface Edge	113
Custom Applications	315
Cut Pattern	111
Cut tolerance	112, 116, 148, 266
Cylinder Length	218
cylinder pattern	217
Cylinder Position	218

D

Depth Filtering	173
Determine by number of cuts	51
Distance	111, 116, 266
Distance increment	111, 147, 265
Drill	19

E

Edit	172
Entities	172
Entry/Exit Settings	138
Extend/trim	48

F

Fan Distance	275
Fanning	273
Feed Distance	100
Flip step over	56
Flow	14
Flow Toolpath	200
Flowline Data Visual Cues	203
From Point	124
From Start To End	79
From start to end for each contour	80
Full, avoid cuts at exact edges	49
Full, start and end at exact surface edges	50

G

Gouge process	133
Gouge Processing	136
Gradual Lead Angle Change	68
Gradual Side tilt Angle Change	68

H

Horizontal tangential arc	99
---------------------------------	----

K

Keep Tool Down Within	137
-----------------------------	-----

L

Last	172
Lead/Lag Angle	130
Lead/lag angle	120
Limit cuts by one or two points	51

Limits	85
Lines	122

M

Machine by Lanes	58
Machine by Regions	58
Machine Inside	218
Machine Limits	25
Manual	272
Manual/Density	273
Mask on Arc	172
Maximum distance	117
Maximum step	112, 116, 148, 150, 266
Maximum/Minimum Radius	218
Minimize corners in toolpath	274
Morph between 2 curves	12, 191
Morph between 2 surfaces	13
Msur	15
Msurf Toolpath	216
Multiaxis Basic	24
Multiaxis Circle Mill	309
Multiaxis drilling	167
Multiaxis Machine Types	28
Multiaxis toolpaths	4
Multiaxis Wireframe	165, 188, 197

N

None	272
Normal to plane	117
Normal to surface	117
Nutating Head	33

O

ordering	323
Orthogonal arc	99
Orthogonal line	99
Output format	273

P

Parallel cuts	9, 163
Parallel to multiple curves	8, 160
Parallel to surface	9, 161
Parallel Triangular mesh	241
Path Between Lines	123
Pencil Triangular mesh	241
Pivot Length	35
Plane	124
Port	15
Port Expert	316
Port Toolpath	228
Project Curves	19

Project curves	297
Projecting Curves	119
Projection	112, 117

R

Radial offset	111, 116
Rapid Distance	100
Reverse tangential arc	98
Reverse tangential line	99
Reverse vertical profile ramp	99
Reverse vertical tangential arc	99
Rib cut	202
Right Hand Rule	22
Rotary	18
Rotary Axes	22
Rotary indexing	282
Rotation In Head Only	29
Rotation In Table And Head	34
Rotation In Table Only	31
Roughing	101
Round corners	48
Run Tool	83

S

Scallop height	202
Select drill point position	172
Servo rotary tables	282
Side Tilt Angle	130
Side tilt angle	120
Side Tilt Definition	

Ortho To Cut Direction At Each Contour 69

Simultaneous Multiaxis machining	4
Single edge	162
Sorting	172
Sorting Options	173
spherical cut pattern	220
Step across - Distance	202
Step along - Distance	201
Step along whole chain	126
Stock to leave	133
Stock to leave on walls	146, 265
Surface	123
Surface quality	59
Surfaces With Seams	115
Swarf	145, 263
Swarf Milling	145
Swarf milling	17
Swarf Toolpath	156, 264
Sync	271

T

Tangential arc	98
Tangential line	99
Tilt -Tilt Head	30
Tilt-Head Rotary Table	35
Tilting tool away with max angle	91
Tilt-Rotary Table	32
Tip Comp	111, 265
Tip control	133
To Point	125
Tool	110
Tool Axis Control	120
Tool axis control	120
Tool Axis Tilting Strategies	64
Tool Length Compensation	26

Tool Vector Length	131
Tool vector length	120, 274
Triangular Mesh	7, 16
Triangular mesh	239

U

Use center point	286
Use wall surface rulings	266

V

Vector depth	133, 135
Vertical tangential arc	99

W

Walls	265
Wind/unwind	26
Window Points	172

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

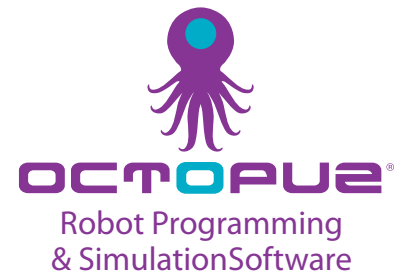
DOES YOUR SCHOOL HAVE ROBOTS? YOU SHOULD CHECK OUT OCTOPUZ.

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.



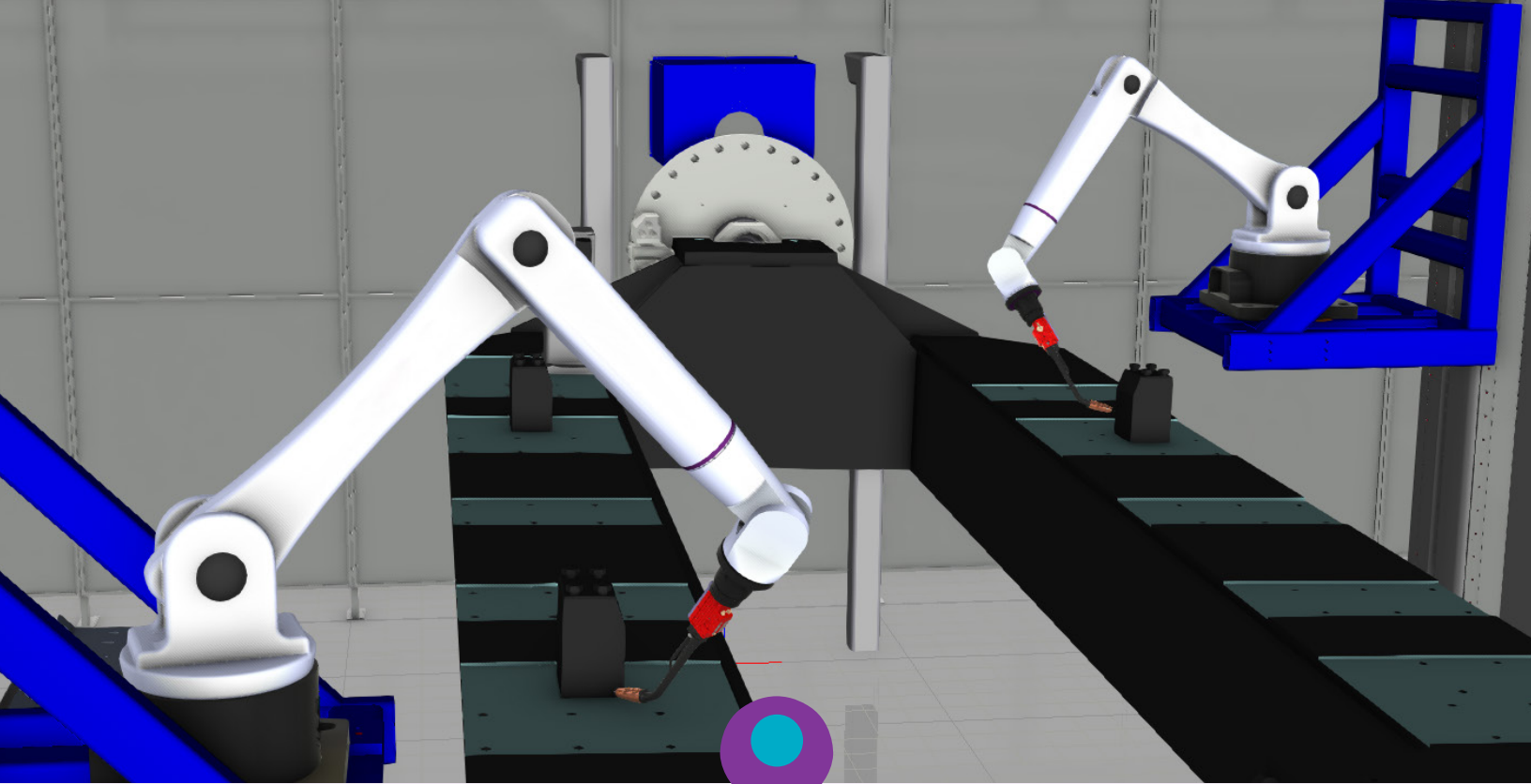
Protect your investment!
OCTOPUZ can detect
collisions offline.



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