Mastercam® 2020

IMPERIAL | HANDBOOK SERIES

HLE DOWNLOAD INSTRUCTIONS INSIDE

HANDBOOK VOLUME 3

Mastercam 2020

HANDBOOK VOLUME 3

Mastercam 2020 Handbook Volume 3

Software: Mastercam 2020

Author: In-House Solutions

ISBN: 978-1-77146-857-2

Date: August 27, 2019

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

Trademarks

This document requires Mastercam Mill 3D.

Table Of Content

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.
- **Panel** is a window that is locked and 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.

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.

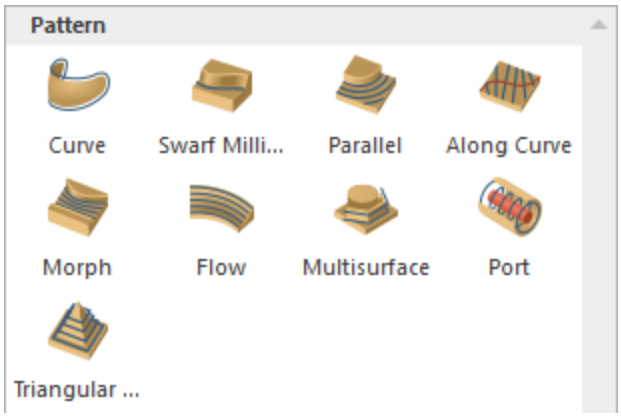
The toolpaths are grouped in two families: **Pattern** and **Application**.

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.

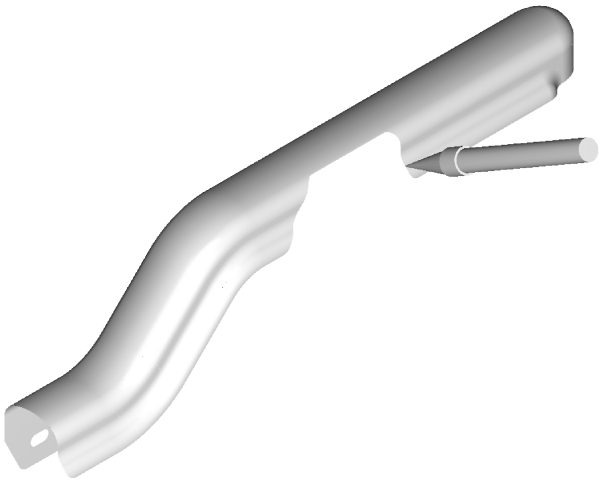
Mastercam has many methods for controlling the tool axes. The most common is to align the tool axis with the surface normal at each tool position as the tool moves along the curve.

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



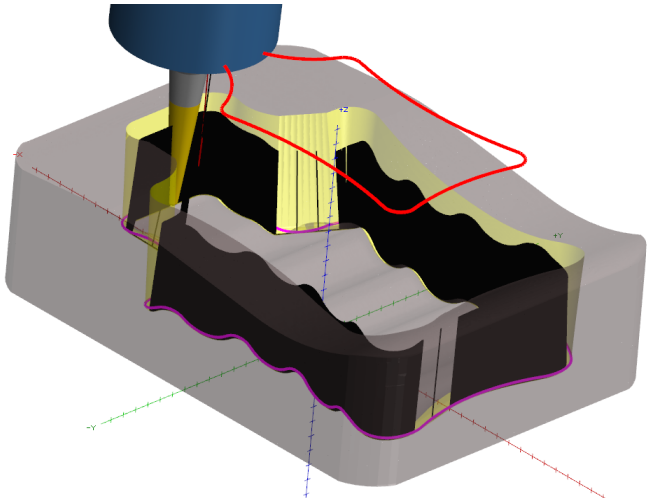
Swarf Milling

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**.

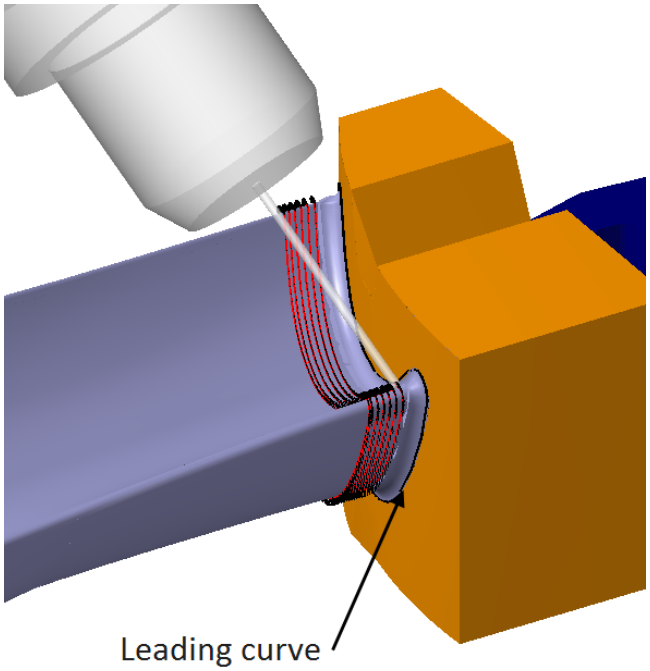
Parallel To Curves

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

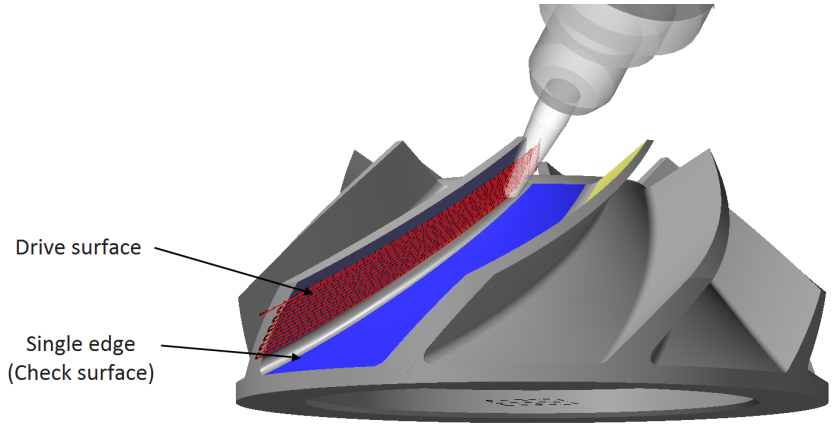


Parallel To Surfaces

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

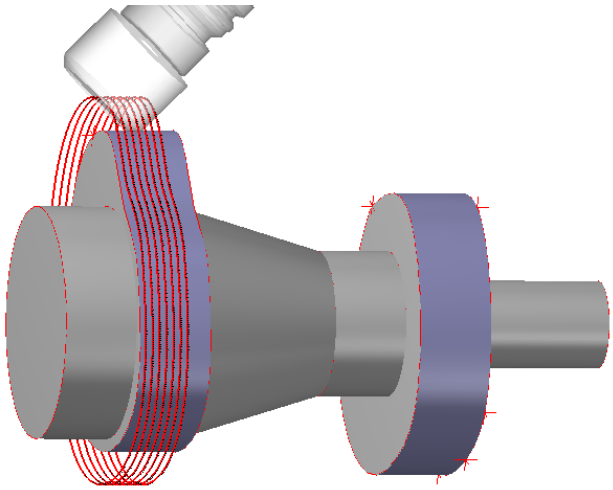


Parallel To Angle

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

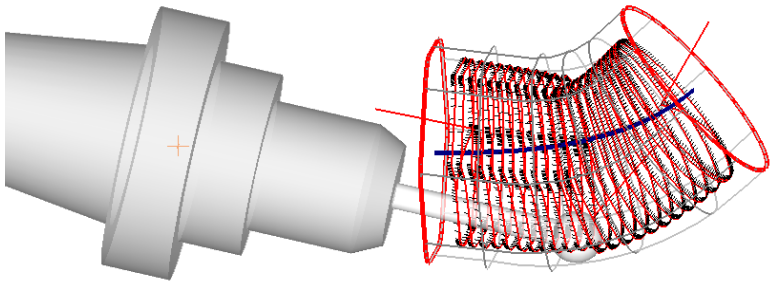


Along Curve

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: 6.

Figure: 6



Morph

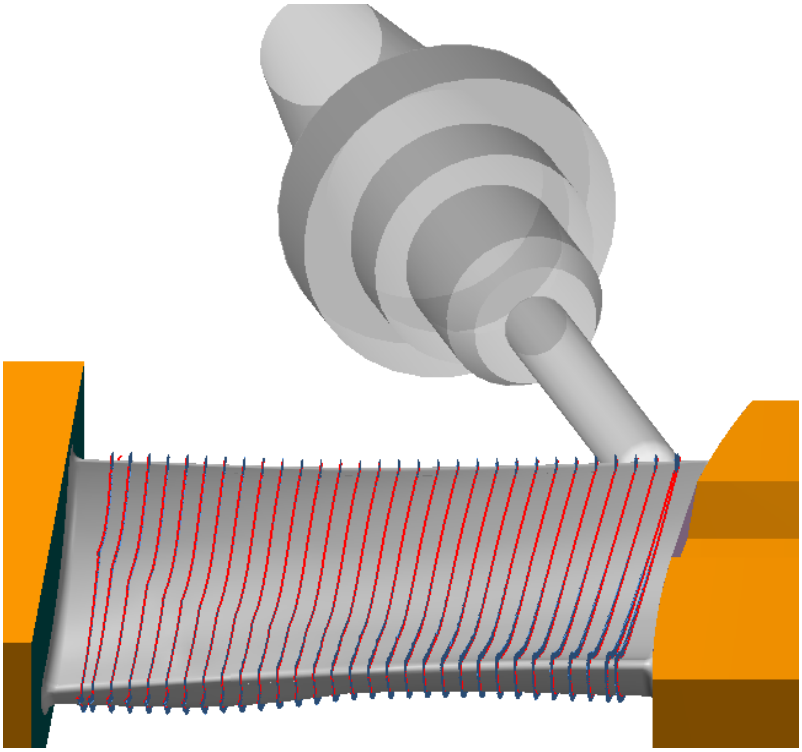
The **Morph** toolpath allows you to choose between two patterns: From Curves and From Surfaces.

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: 7.

Figure: 7

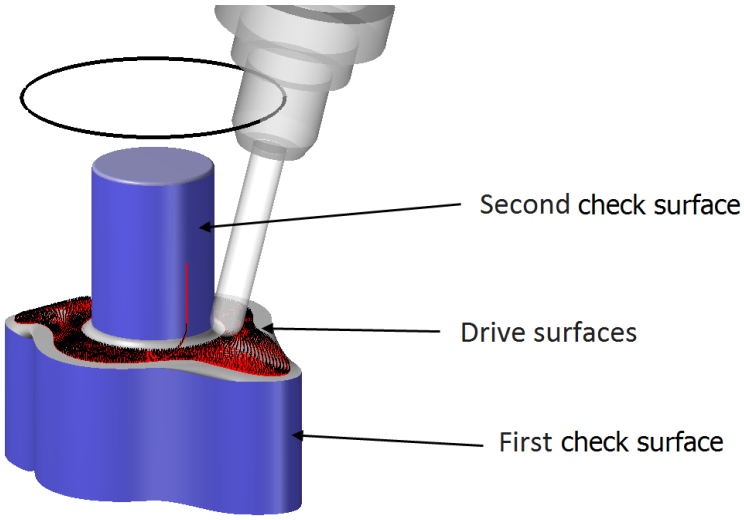


Morph - Pattern From Surfaces

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: 8.

Figure: 8



Flow

Flow 5-axis toolpaths follow the UV Curves of a surface or row of surfaces.

While mostly used to machine a single surface, Flow can sometimes be used to machine across multiple surfaces. Because they can be very efficient and allow a lot of control, Flow is a widely used toolpath strategy.

Flow provides extensive tool axis and cut control, including lead/lag, side tilt, axis limits and more. It also provides scallop height stepover control.

The following graphic shows a 5-axis Flow toolpath used to finish machine an aircraft turbine blade as shown in Figure: 9.

Figure: 9

3 Multiaxis Curve Toolpaths

OBJECTIVES

The purpose of this chapter is to introduce Multiaxis Curve toolpaths.

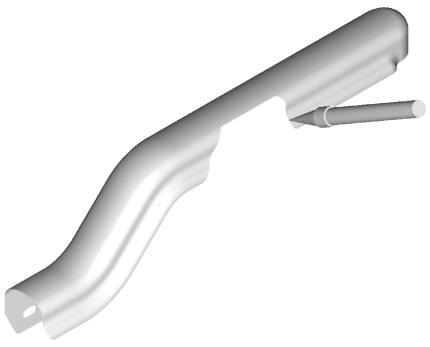
Upon completion of this chapter, you should be able to do the following:

- Understand concepts related to Multiaxis Curve toolpaths and recognize applications where they should be considered.
- Understand the difference between 3, 4, and 5-axis output formats.
- Know how to control the toolpath using 3D curves or surface edges.
- Know how to control the tool axis orientation using all available methods.
- Understand tool tip compensation as it relates to Multiaxis curve machining.
- Know the difference between Surface and Plane projections.
- Be able to define and set the tool lead/lag and side tilt angles.
- Understand all parameters and settings related to Entry/Exit and be proficient with them.

INTRODUCTION

Curve toolpaths are used to move the tip of the tool along a 3D curve while controlling the tool axis as shown in Figure: 1. This toolpath has many applications. The illustration below shows a Curve toolpath being used to control a laser to trim cut a molded manifold.

Figure: 1

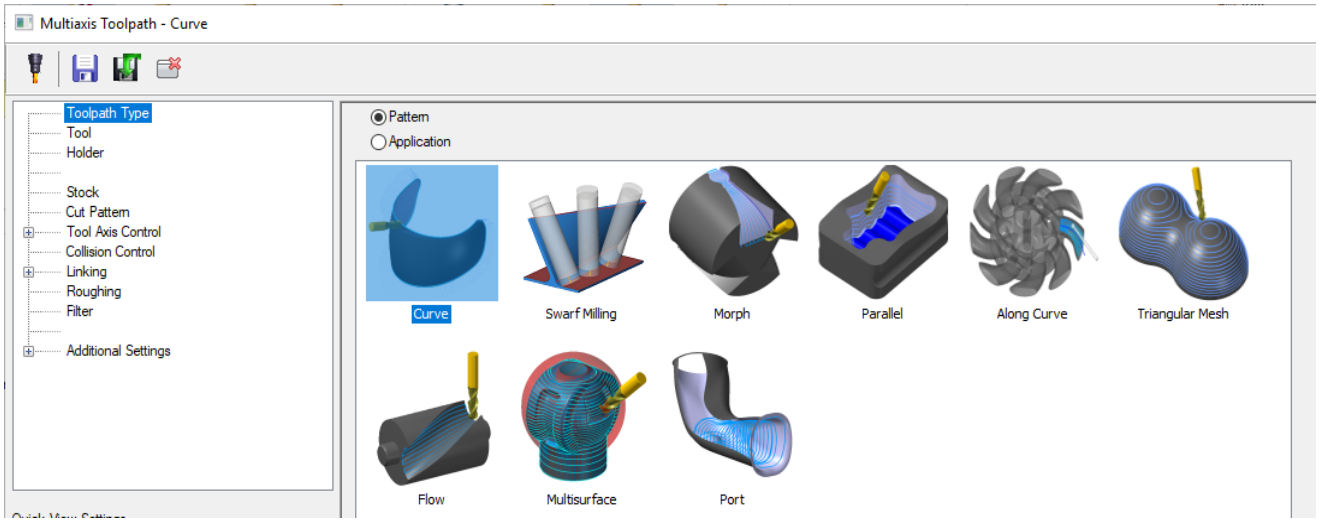


CURVE TOOLPATH

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

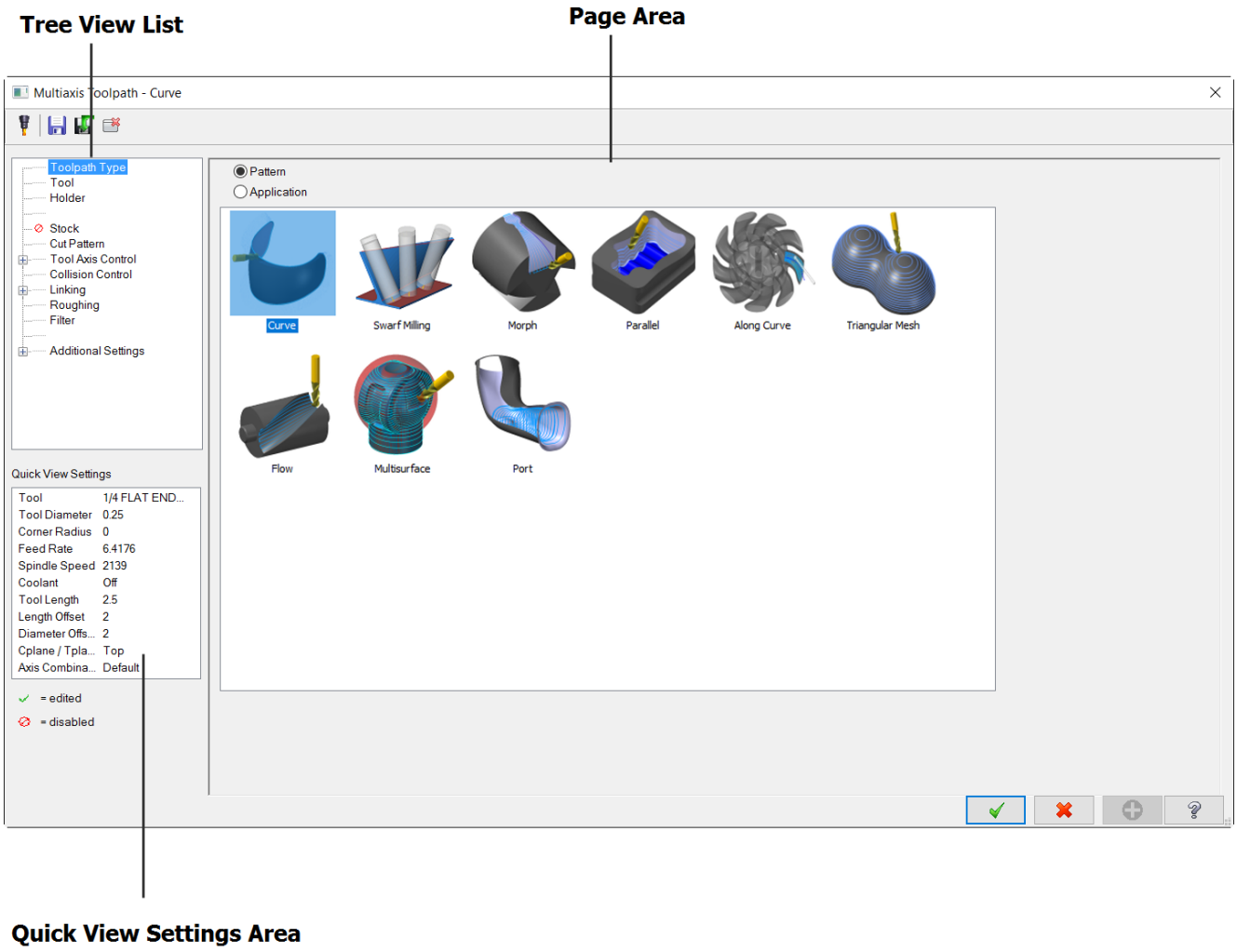
Toolpath Type

Toolpath Type controls the options available as you proceed down the tree structure. The options are based on the toolpath you previously selected, as well as the toolpath family (Pattern or Application).



Mastercam toolpath dialog boxes have three distinct areas: **Tree View list**, **Page**, and **Quick View Settings** as shown in Figure: 2.

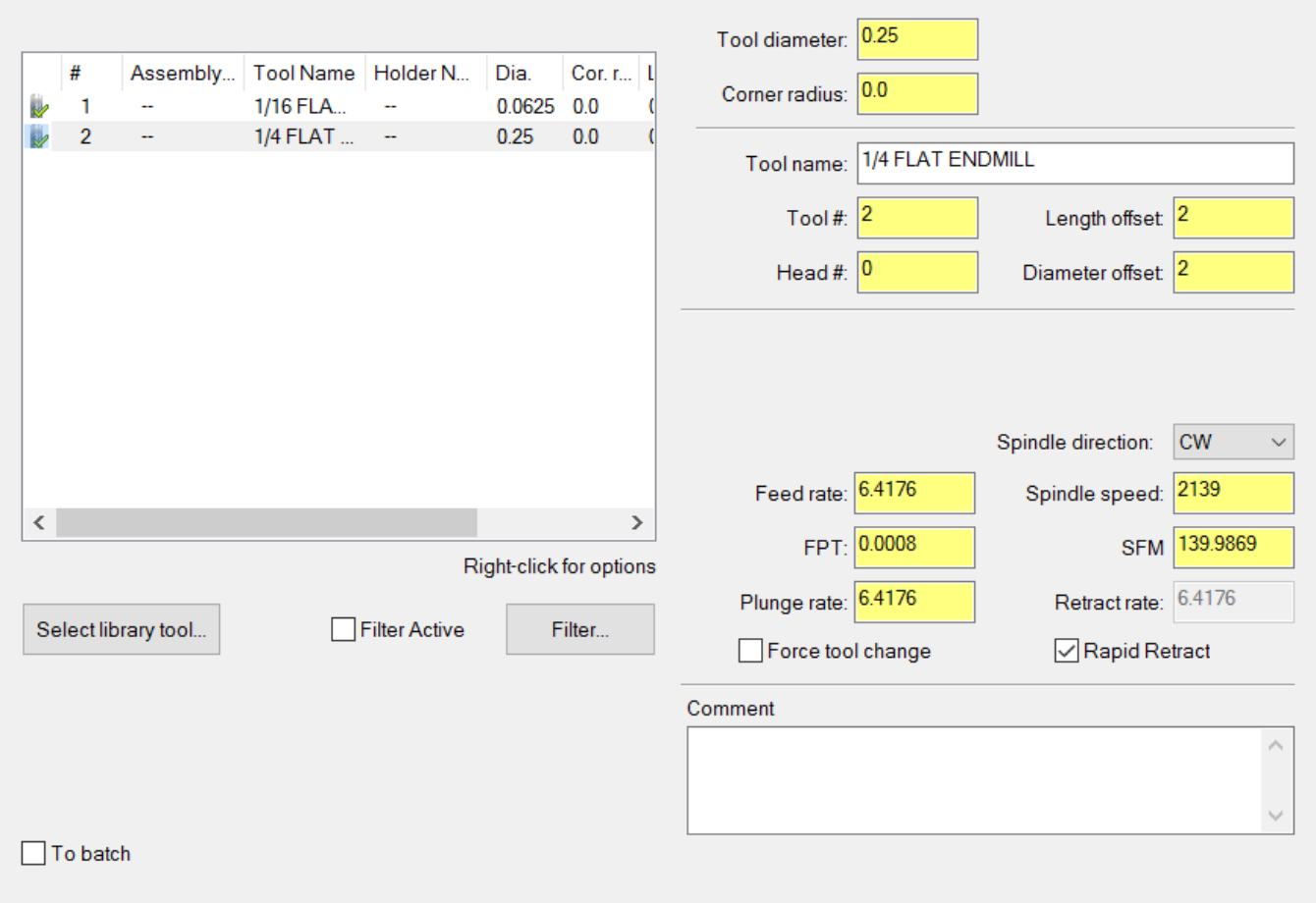
Figure: 2



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 as shown in Figure: 3. This page is identical to the **Tool** page used throughout Mastercam 2D toolpaths. For questions regarding parameters on this page, consult the online help (Alt + H) or refer to the **Mastercam Handbook Volume 1** or **Volume 2**.

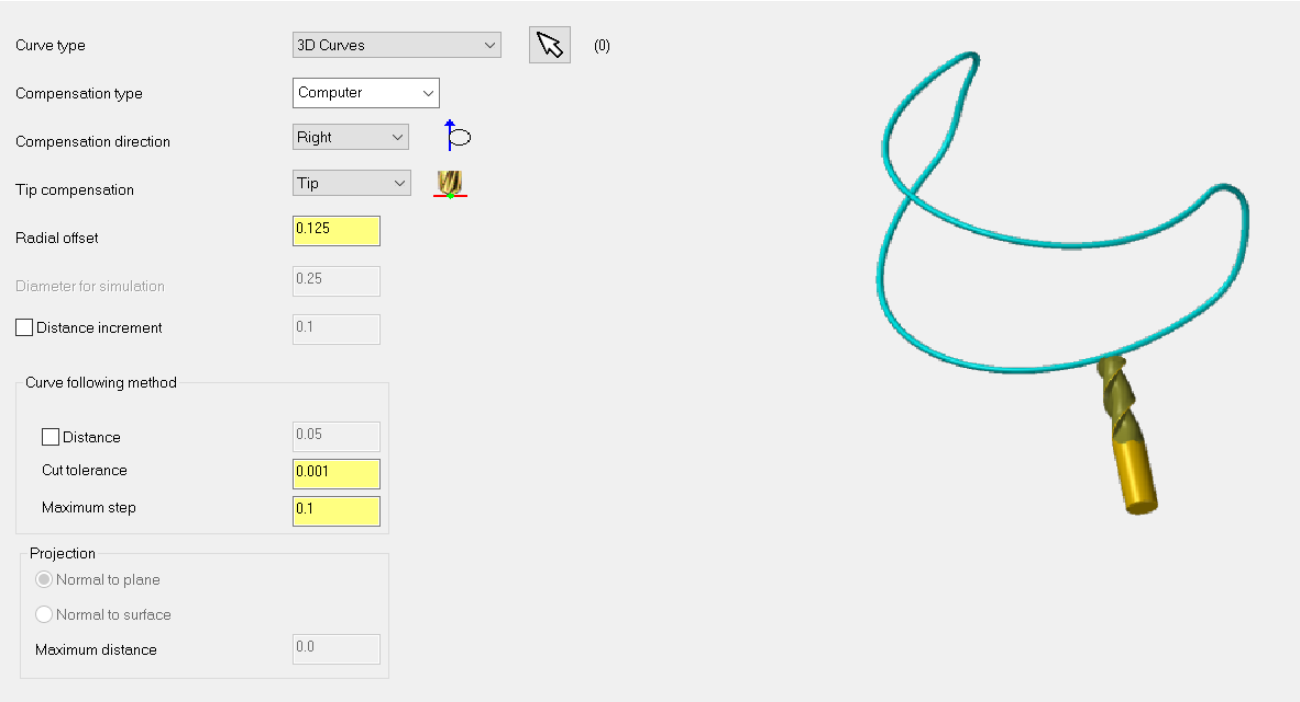
Figure: 3



Cut Pattern

Cut Pattern settings determine the geometry the tool follows and how it travels along that geometry as shown in Figure: 4.

Figure: 4



Term	Definition
Curve type	**Curve type** can be set to **3D curves** (lines, arcs or splines) or **Surface edges** when you are not using chained geometry.
Compensation type	Allows you to specify how you want to handle cutter compensation. This option is identical to that used in 2D and 3D toolpaths.
Compensation direction	Gives you the choice to offset the tool to the right or left of the toolpath.
Tip compensation	Sets the offset to be used relative to the tool's center or tip.
Radial offset	Sets the distance that the center of the tool is offset (left or right) based on the Compensation direction.
Distance increment	This value is a linear distance along the path taken by the tool. An additional vector will be added to the toolpath when the distance between calculated vectors is greater than the distance increment value.
Distance	Enable this option to use a distance value to limit tool motion. The specified value is the distance between generated vectors along the selected geometry.
Cut tolerance	**Cut tolerance** determines the accuracy of the Multiaxis toolpath.

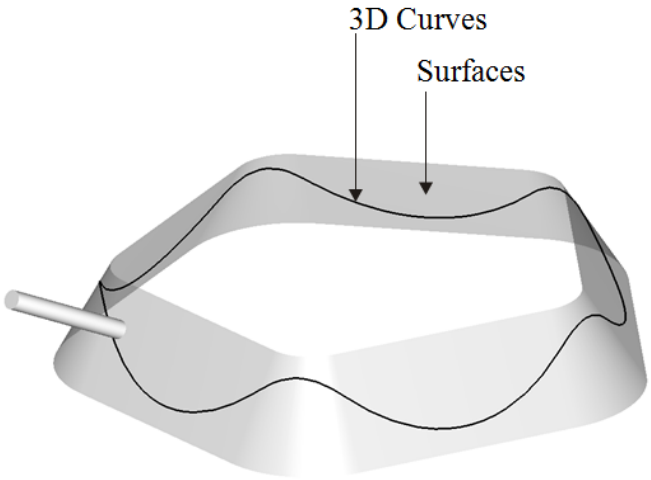
Term	Definition
Maximum step	**Maximum step** sets the largest step that can be made between tool moves so that the tool moves can be kept closer together in flat areas of the part.
Projection	**Projection** set to **Normal to plane** projects the curves to the surface normal to the current construction plane. **Projection** set to **Normal to surface** projects the curves to the surface along the surface normals.

Curve Type 3D Curves

The path of the tool is controlled by the **Curve type** group. Clicking on the **3D Curves Select** button brings up the Mastercam chaining menu. Use this menu to select the curves to machine.

Curves can be composed of lines, arcs, and NURBS splines. Parametric splines and surface curves are not allowed as shown in Figure: 5.

Figure: 5



The best results are achieved using quality data. When possible, model the curves using the simplest geometry possible i.e. using lines instead of straight splines, arcs instead of circular splines, etc.

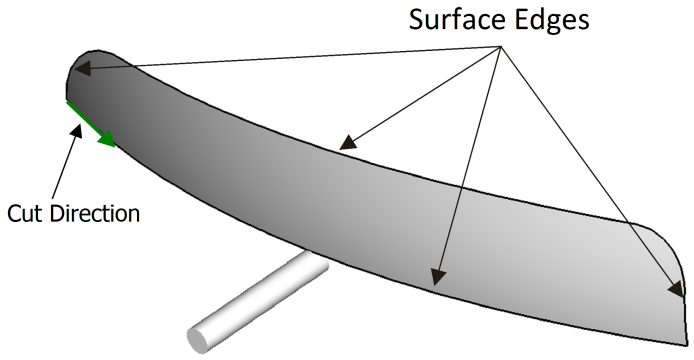
Avoid splines with excessive control points, reversals, or other undesirable characteristics. If necessary, reduce the splines or remodel them. The **Mastercam Handbook Volume 2**, **Chapter 1: Splines** provides information on spline modeling and troubleshooting.

Curve Type Surface Edge

The **Surface edge - single** option allows you to control the toolpath using a selected surface edge instead of 3D curves.

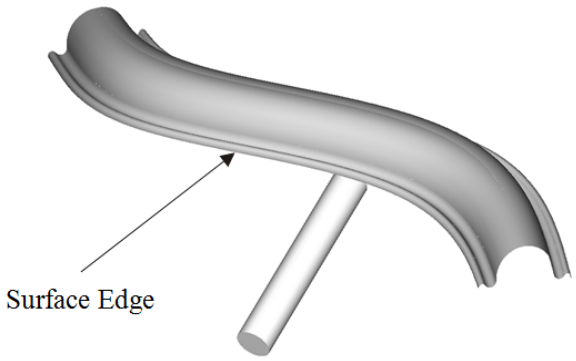
The **Surface edge - all** option drives the tool around all surface edges. The system prompts you to select the surface, and then to select the cut direction as shown in Figure: 6.

Figure: 6



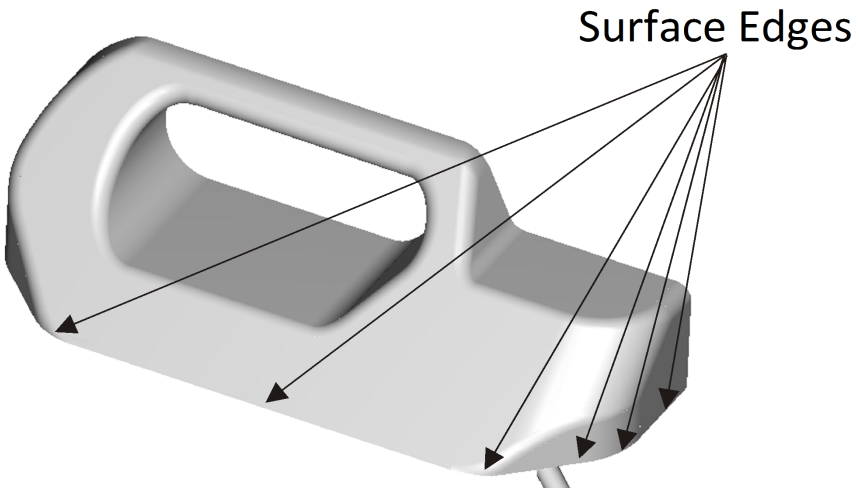
Use the **Single** setting to drive the tool around one edge of a surface or group of surfaces as shown in Figure: 7.

Figure: 7



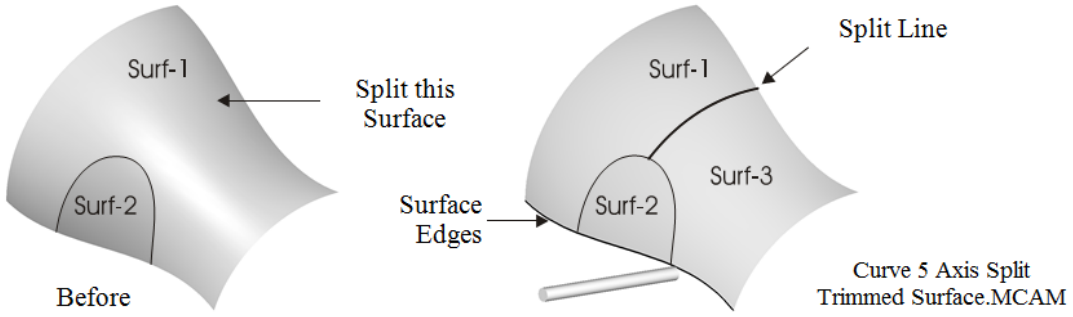
When more than one surface is selected, the system prompts you to pick the first surface, and then the edge to start on as shown in Figure: 8. It then prompts you to select the edge to use for the next adjacent surface. This is repeated for each subsequent surface until one edge of each surface is selected.

Figure: 8



In cases where multiple edges of a single surface are used as edges, first split the surface using the SURFACES/Split function as shown in Figure: 9.

Figure: 9



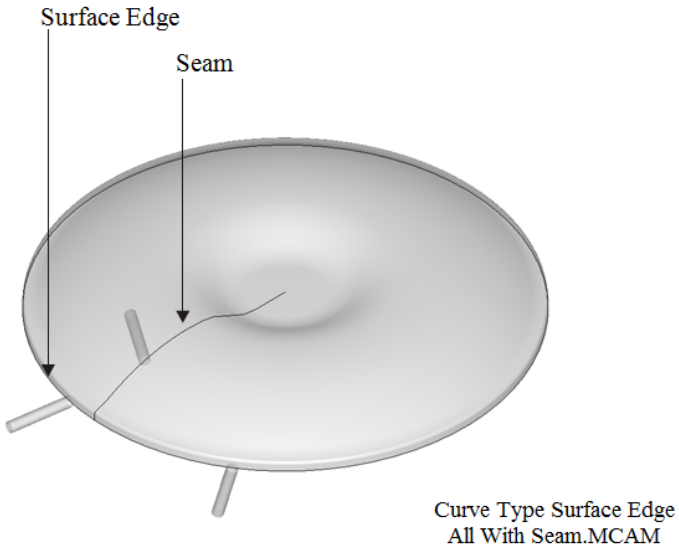
Surfaces With Seams

Surfaces generated from cross sections, such as **Revolved** or **Net** surfaces, have a seam as shown in Figure: 10.

The toolpath will drive along this seam and the perimeter of the surface when using the Surface Edge function.

Consider using the 3D Curves option rather than Surface Edge to avoid machining the surface seam.

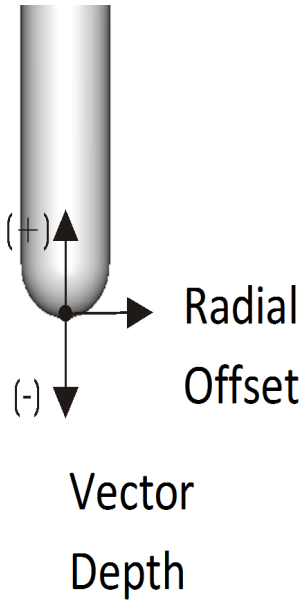
Figure: 10



Radial Offset

Radial offset is related to the cut direction as shown in Figure: 11. Set this parameter to the radius of the tool.

Figure: 11



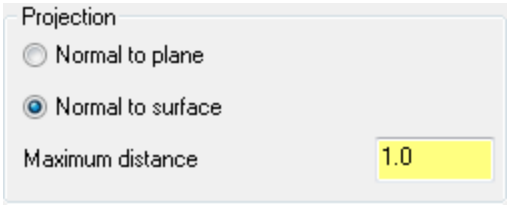
Curve Following Method

The **Curve following method** allows you to set the accuracy of the Multiaxis toolpath.

Term	Definition
Distance	Enable this option to use a distance value to limit tool motion. The specified value is the distance between generated vectors along the selected geometry. A smaller value creates a more accurate toolpath, but may take longer to generate and may create a longer NC program.
Cut tolerance	**Cut tolerance** determines the accuracy of the Multiaxis toolpath. A smaller cut tolerance value creates a more accurate toolpath but may take longer to generate and create a longer NC program. This value is also used to eliminate duplicate positions and as a tolerance when compensating to surfaces. When the Distance check box is deselected, Mastercam uses the cut tolerance you specify as a tolerance to fit tool positions to the surface based on the curvature of the surface. Otherwise, Mastercam uses the Distance value as an absolute distance for the space between each tool position.
Maximum step	**Maximum step** sets the largest step that can be made between tool moves so that the tool moves can be kept closer together in flat areas of the part.

Projection

Projection determines how the curves are projected onto the surface.



Term	Definition
Normal to plane	Causes the curve to be projected normal to the active construction plane.
Normal to surface	Projects the curve onto the surface along the surface normals.
Maximum distance	Controls how far the system looks to find the surface on which to project the curve. To reduce calculation time, set this value to a little more than the maximum distance between the curve and the surface it is projected onto. If the curve lies directly on the surface, a setting of 0.05 inches or less may be used. Use this setting to prevent extra projections that could occur when the projected curve intersects a surface more than once. For example, a curve projected through a cylinder.

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

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**

In-House Solutions

Mastercam @Courses

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

LOGIN & LEARN!

Mastercam CAD/CAM Systems 2020 Home Learning Edition Demo Software DOWNLOAD

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.

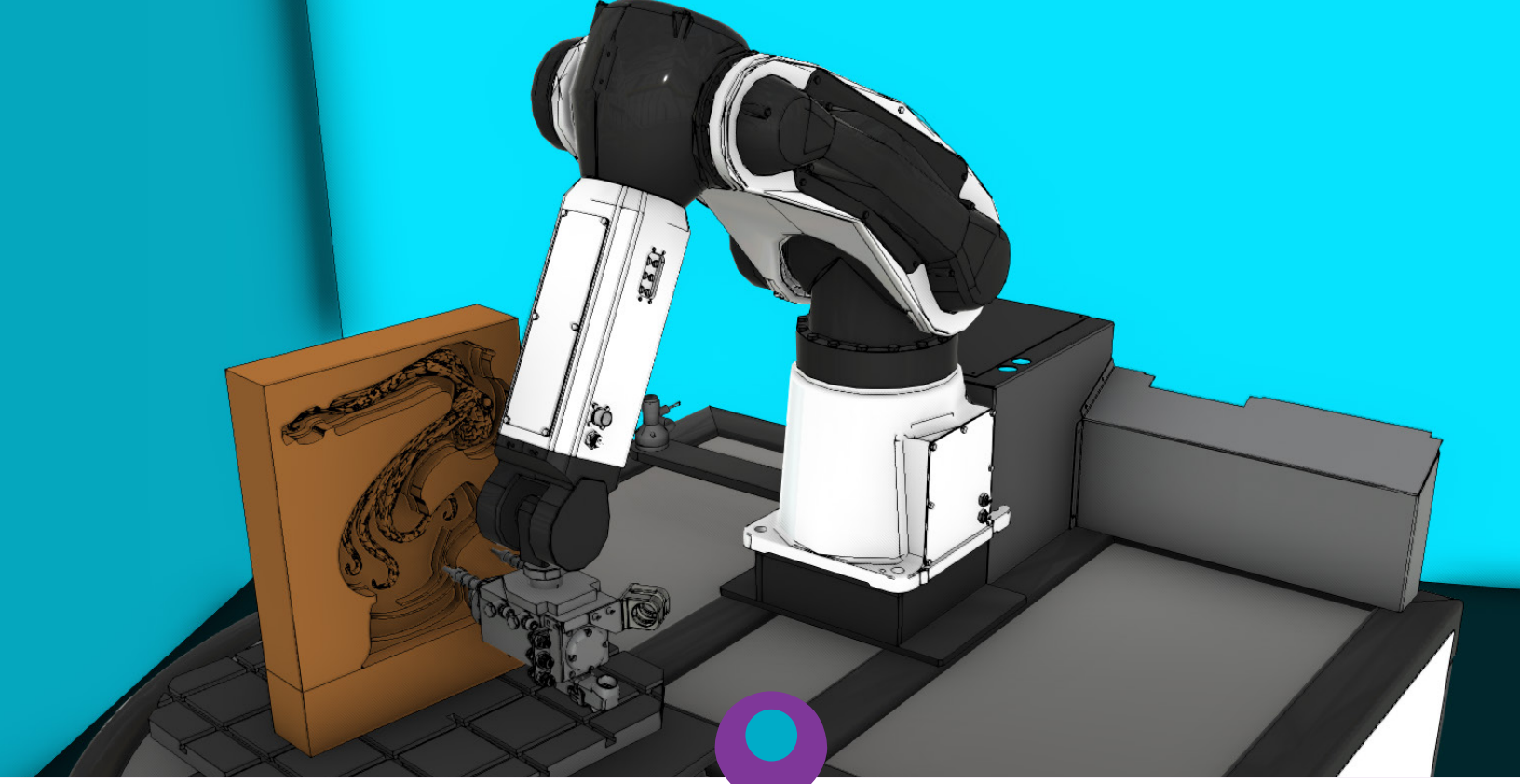
The *"log in & learn"* format of each eCourse allows users to set up their own online classroom, where each user's experience is customized and tracked. Because the eCourses are not subscription-based and have no expiration date, users can log on and off at any time, and finish the program at their convenience 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

eMastercam

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!