Mastercam® 2020

IMPERIAL | PROFESSIONAL COURSEWARE

HLE DOWNLOAD
INSTRUCTIONS INSIDE
2020

LATHE

Mastercam® 2020

LATHE
PROFESSIONAL COURSEWARE

To order more books:

Call 1-800-529-5517 or

Visit www.emastercam.com or

Contact your Mastercam dealer

HOW TO DOWNLOAD:

Mastercam Home Learning Edition (HLE) Demo Software

Visit www.**eMastercam**.com/**files**

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

Proceed to download!

Mastercam 2020 Lathe Professional Courseware

Software: Mastercam 2020

Author: Mariana Lendel

Date: June 10, 2019

ISBN: 978-1-77146-860-2

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

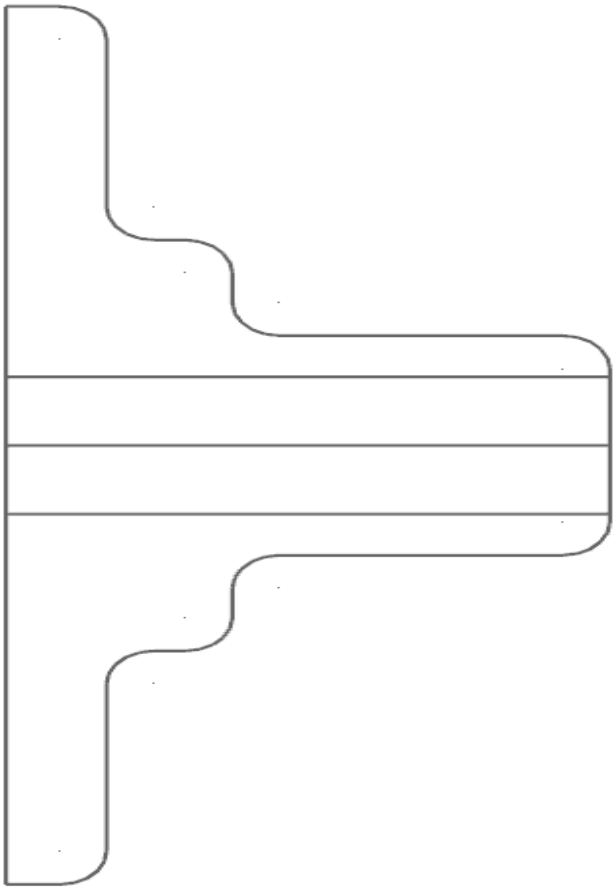
Trademarks

Mastercam is a registered trademark of CNC Software, Inc.

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

Table Of Contents

Geometry Tools - Part 1



INTRODUCTION

This lesson will introduce the Mastercam user interface in detail and will also teach you the first geometry creation tools that you will need for creating your own geometry to define toolpaths or for creating supplemental geometry to graphically control the toolpath motions in Mastercam. The main goal of the geometry lessons is to give you the basic understanding of how to create geometry in Mastercam so that you can practice to become as proficient as your job requires.

OVERVIEW OF EXERCISE:

In this lesson we will become familiar with the Mastercam screen components and learn tools and shortcuts to begin creating basic line geometry shapes.

NEW CONCEPTS COVERED IN THIS LESSON:

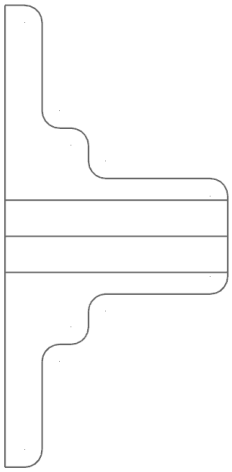
- The important components of the Mastercam Interface.
- Important Function Key shortcuts.
- How to create a Point knowing its Position.
- Line Creation - Several line creation methods with emphasis on Endpoints and Parallel.
- How to Delete Geometry.
- Fillet Creation.
- How to Trim and Extend Lines.
- How to Create Rectangles and Rectangular Shapes.

INSTRUCTOR DEMONSTRATION PREVIEW

Note: This entire lesson is a joint Instructor / Student exercise.

Topics:

- **Mastercam Interface**
- **Point Position Command**
- **Line Endpoints Command**
- **Line Parallel Command**
- **Rectangle Commands**
- **Fillet Commands**
- **Trim Commands**



NOTES:

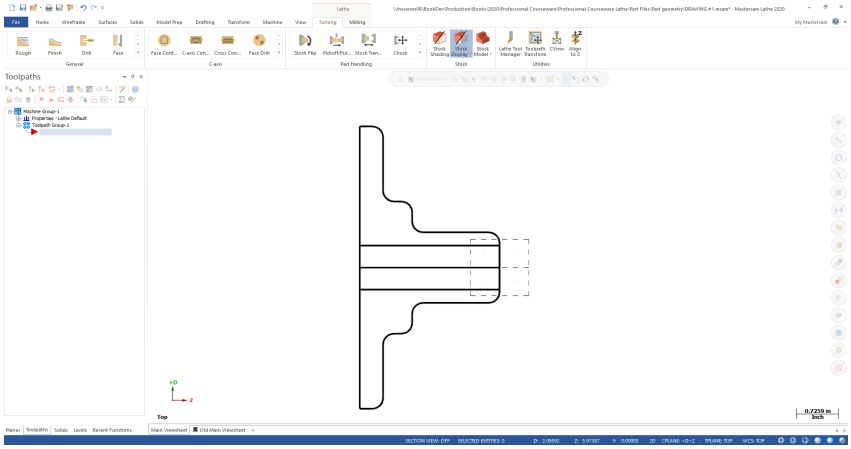
GEOMETRY TOOLS PART 1: BASIC STEPS REFERENCE

In this first lesson the students and instructor will work through the topics as a group.

EXPLORE MASTERCAM INTERFACE

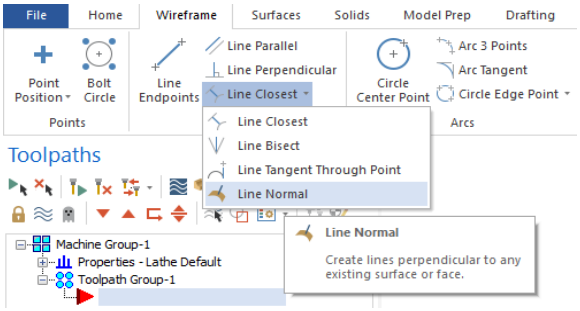
1. Load the part: "DRAWING #1 .MCAM"

We will do a "Preview" of some of the different components of the Mastercam screen. Goal is just to see where things are, not to master their usage yet.



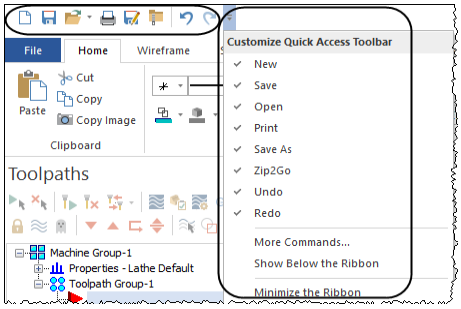
2. Tabs - Buttons - Submenus - Tooltips

Like many Windows programs, many have submenus or options. We will not go through any specific at this point, just review how they work.



3. Quick Access Toolbar

User can customize to add often used functions.

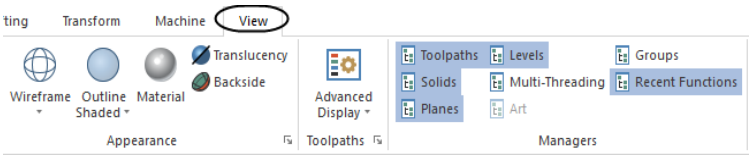


4. The ESC (Escape) Key

(Master key for ending most functions)

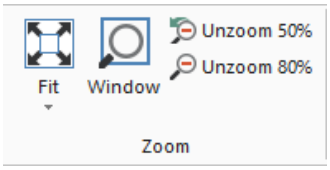
5. The View Tab

Provides access to many of the options to control the Mastercam layout.



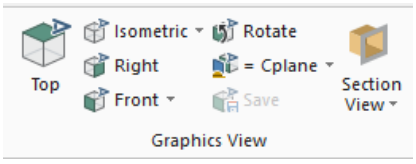
6. Zoom Commands

(Since these commands are used extensively, it may be useful to memorize the keyboard shortcut keys)

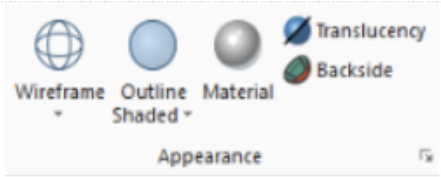


- **Fit (Alt+F1)**
- **Zoom Window(F1)**
- **Un-Zoom 80% (Alt+F2)**
- **Un-Zoom Previous / 50% (F2)**
- **Repaint (F3)**

7. Graphic Views

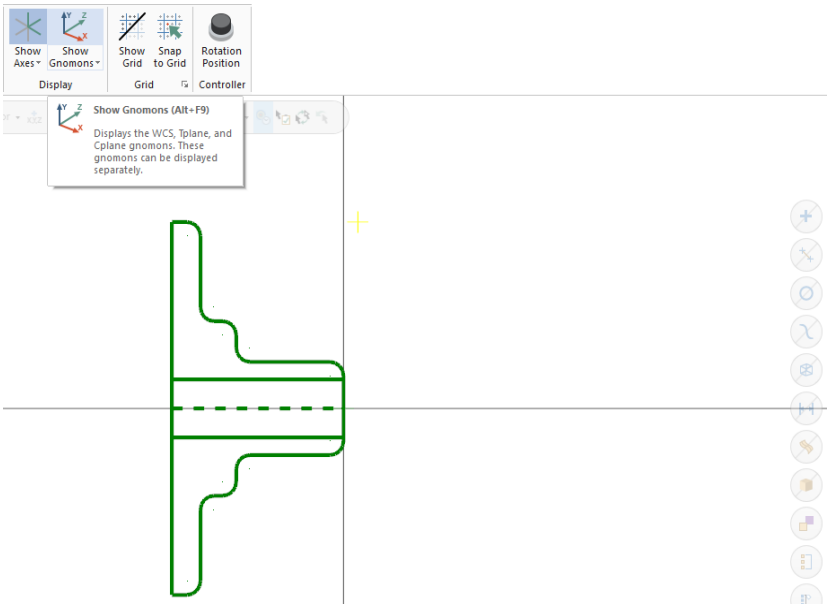


- Standard View Choices RMB (Right Mouse Button options)
- Free Spin rotation with middle mouse button
- Zoom in and out with middle mouse wheel



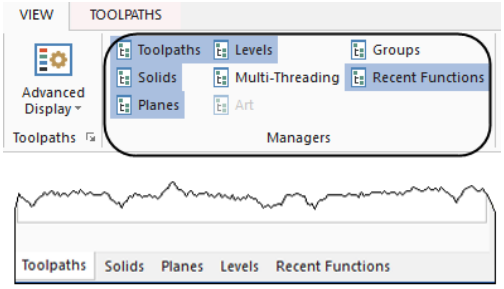
8. Appearance

- Wireframe Choices
- Shaded Choices
- Appearance
- Stock Display options
- Shaded and Wireframe Display of Solid **Alt+S**



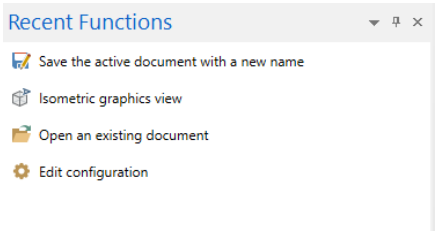
9. Axis and Gnomon Display

- **F9**
- **Alt+F9**

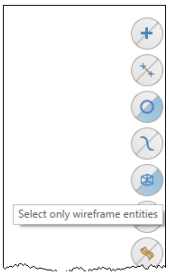


10. The Manager Panel

(Preview of the location and purpose only. Functions to be covered when needed.)



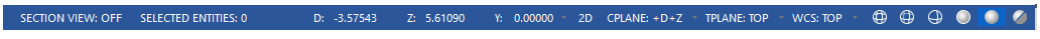
11. MRU (Most Recently Used) Panel Preview



12. Quick Mask Preview

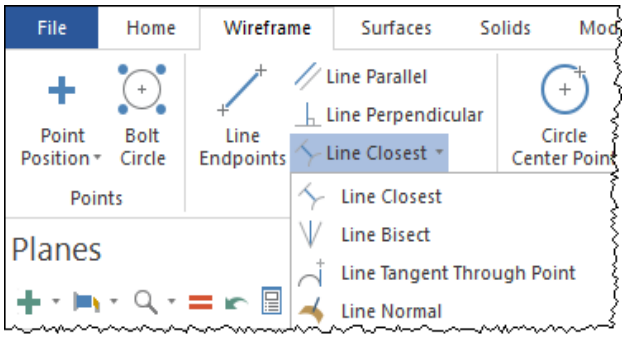
(Select all versus Select only)

13. Status Bar



14. Wireframe Tab preview

(We will focus on Lines icons)

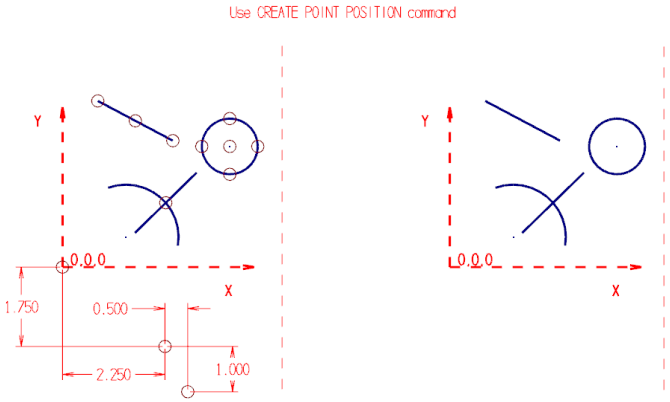


POINT POSITION COMMAND

Note: The following instructor led training exercises cover different basic geometry creation commands. The student has to complete the right side geometry based on the left side geometry and dimensions.

Resources - Download the file from www.EMASTERCAM.COM/TRAININGFILES/

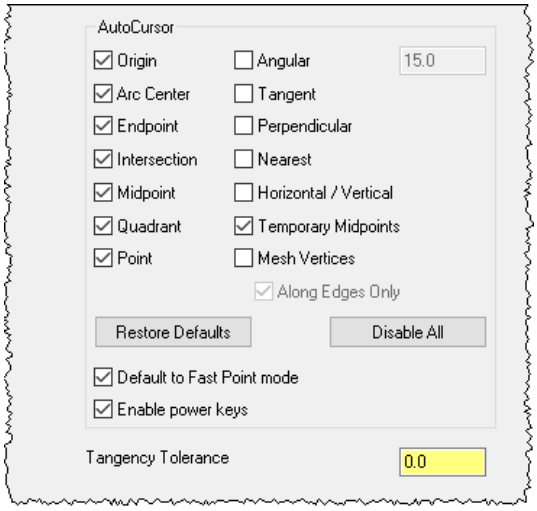
- From the **QAT**, select the **Open** icon.
- Select CREATE_POINT_POS.MCAM.



15. Set Auto Cursor in the Selection Bar

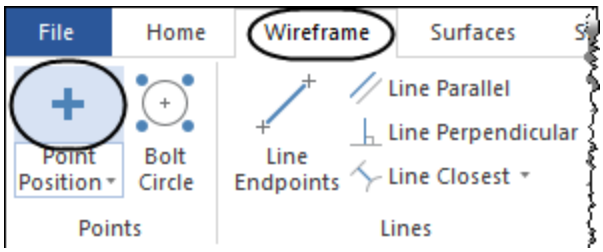
Instructor recommended settings:

- From the **General Selection Toolbar** select the **AutoCursor Configuration** icon.
- Set the settings as shown.



16. Point Position

*Note: Make sure that you are in the **Lathe +D+Z** plane. Create the points on the right side of the drawing to match the left side.*



Wireframe

- Select **Point Position**.

- Use the **AutoCursor** options to insert the points at the **Endpoint**, **Midpoint**, **Intersection**, **Quadrant**, **Origin** and **Center** locations. As Mastercam detects and snaps to the points, it displays a temporary square over the point and the corresponding cursor icon.

- Enter the coordinate values for the first point as shown.
- Press **Enter** from the keyboard to see the point created.

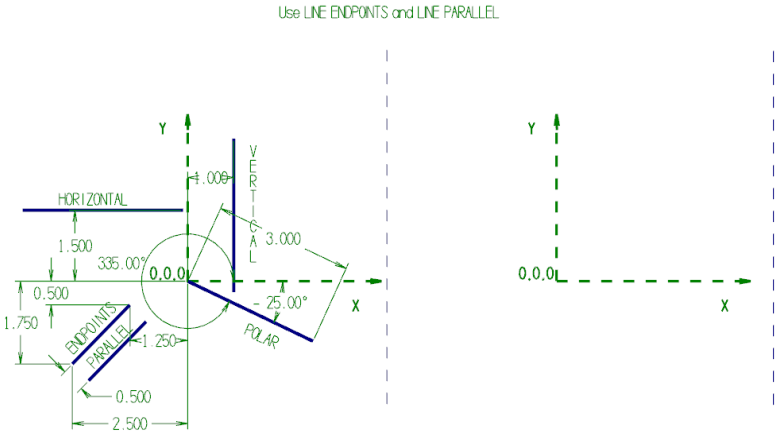
- Enter the coordinate values for the second point as shown.
- Press **Enter** from the keyboard to see the point created.

Note: Mastercam can perform simple math operations such as addition, abstracting, multiplications and divisions.
The instructor can also demonstrate how to change to Relative when entering the coordinates.

LINE ENDPOINTS & PARALLEL COMMANDS

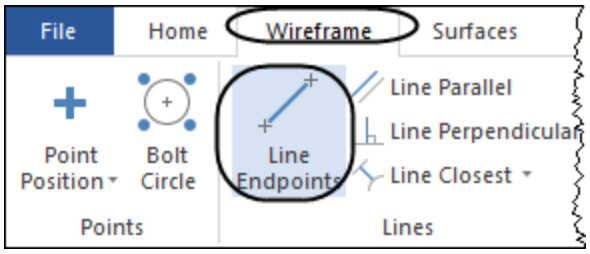
Resources - Download the file from www.EMASTERCAM.COM/TRAININGFILES/

- From the **QAT**, select the **Open** icon.
- Select CREATE LINES.MCAM.



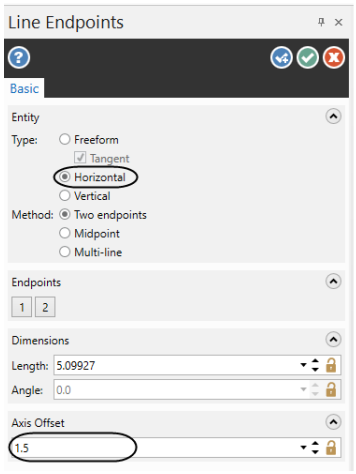
1. Line Endpoints

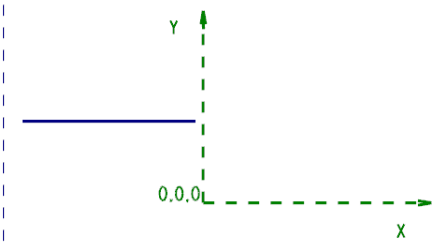
Wireframe



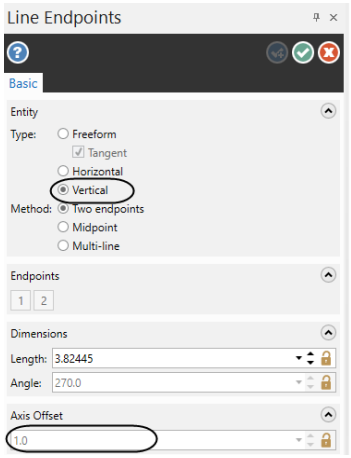
- Select **Line Endpoints**.

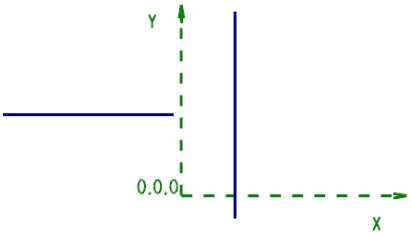
- Enable the **Horizontal** option in the **Line Endpoints** panel.
- Sketch the endpoints of the horizontal line approximately at the same location from the X and Y axes as per the left side drawing.
- In the **Axis Offset** box enter **1.5** as the **Y coordinate.**
- Press the **Enter** key to position the line.
- Press **Enter** again to finish the line or select the **OK and Create New Operation** button to continue in the same command.



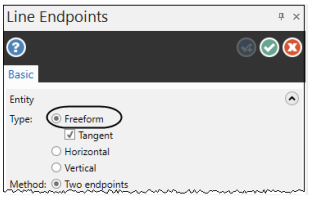


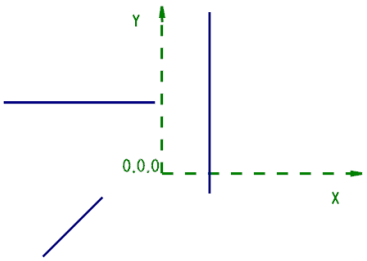
- Enable the **Vertical** option in the **Line Endpoints** panel.
- Sketch the endpoints of the vertical line approximately at the same location from the X and Y axes as per the left side drawing.
- In the **Axis Offset** box enter **1.0** as the **X coordinate.**
- Press the **Enter** key to position the line.
- Press **Enter** again to finish the line or select the **OK and Create New Operation** button to continue in the same command.



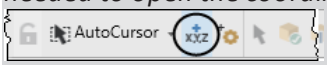


- Select the **Freeform** button in the **Line Endpoints** panel.
- Type the coordinate values for the first point (**-1.25,-0.5**) and press **Enter**.



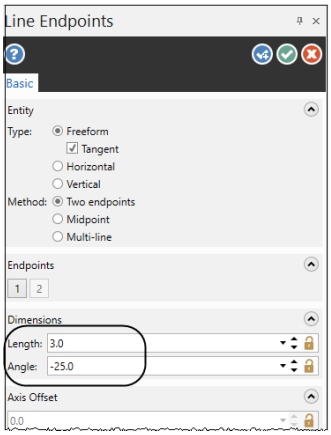


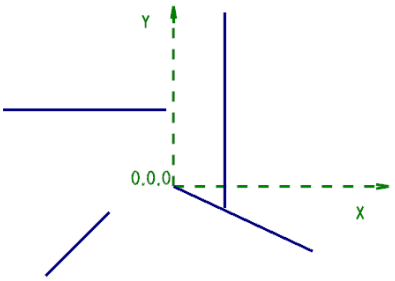
*Note: Select **the AutoCursor Fast Point** icon if needed to open the coordinates' field.*



- Type the coordinate values for the second point (**-2.5,-1.75**) and press **Enter**.
- Select the **OK and Create New Operation** button to continue in the same command.

- In the **Line Endpoints** panel, input a **Length** of **3.0** and an **Angle** of **-25.0**.
- Pick the **Origin** (center of the **Grid**) as the first endpoint.
- Select the **OK** button to exit the command.

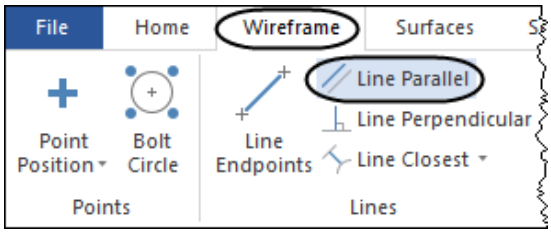




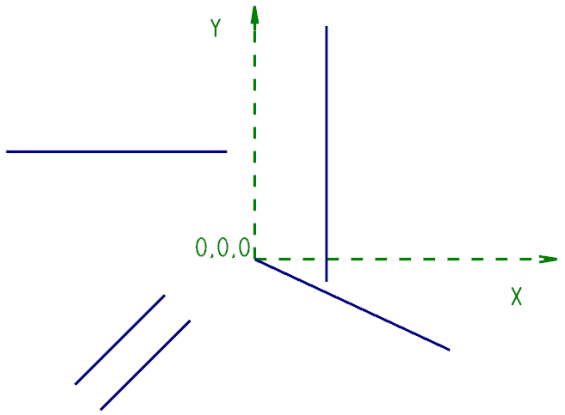
2. Line Parallel

Wireframe

- Select **Line Endpoints**.



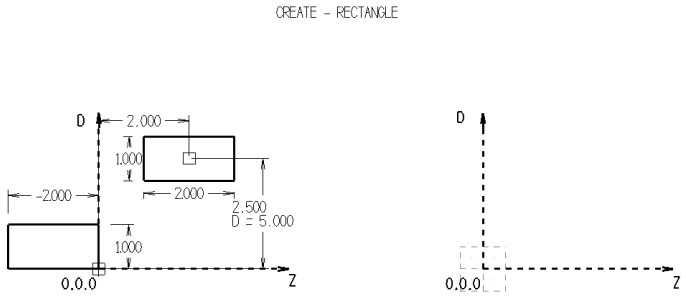
- Select the existing line and indicate the offset direction by clicking below the existing line
- In the **Line Parallel** panel, enter an **Offset Distance** of **0.5**. Press **Enter**.
- Select the **OK** button to exit the command.



RECTANGLE COMMANDS

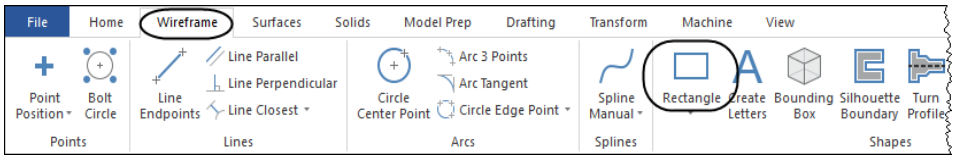
Resources - Download the file from www.EMASTERCAM.COM/TRAININGFILES/

- From the **QAT**, select the **Open** icon.
- Select CREATE_RECTANGLE.MCAM.
- Ensure that you are in the +D+Z plane.



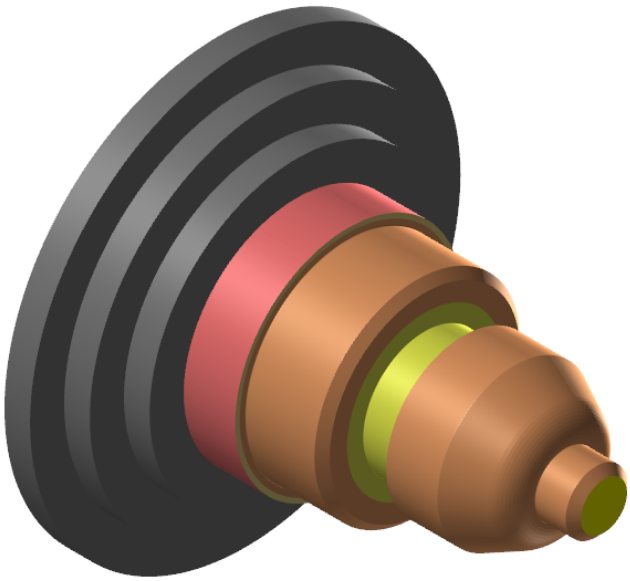
3. Rectangles

- From the **Shapes** group, select **Rectangle** icon as shown.



- Type the **Width** and the **Height** values, and make sure that you select the point placement that you know the coordinate values as per the drawing (0,0).
- Reselect the command to change the anchor to **Center** and press **Space** to enter the coordinates: (5,2.5).

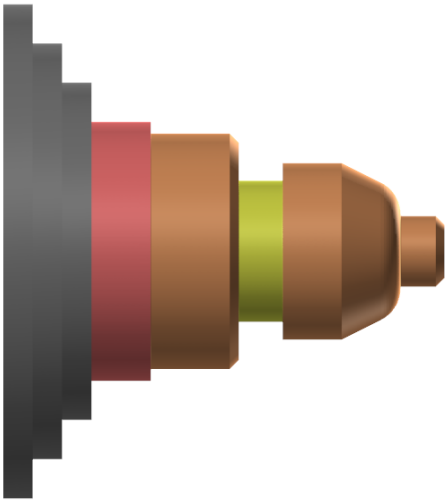
Groove Toolpath



NEW TOOLPATH INFORMATION

The following is brief description of the toolpath used in this exercise.

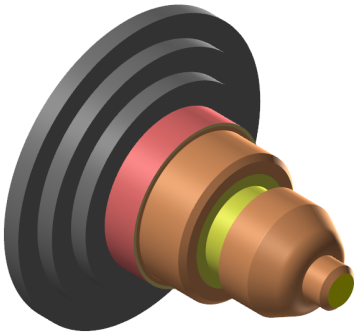
Grooving for machining indented or recessed areas that are not otherwise machinable by roughing toolpaths or tools. You can machine several grooves in a single operation, even if their geometry never connects. Mastercam integrates roughing and finishing passes, each with separate parameters, in a single operation.



INSTRUCTOR DEMONSTRATION

Topics:

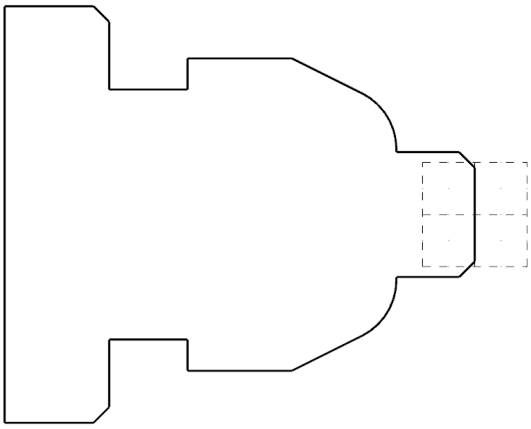
- Locate Geometry and Job Setup
- Facing Operation
- Roughing Operation
- Finishing Operation
- Grooving Operation



NOTES:

EXERCISE: DRAWING #3

STEP 1: OPEN PART GEOMETRY



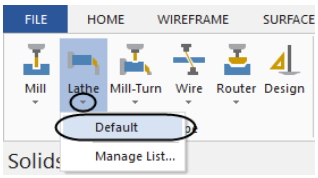
1.1 Open the file

- From the **QAT**, select the **Open** icon.
- Select **DRAWING 3.MCAM**.

STEP 2: CREATE THE JOB SETUP

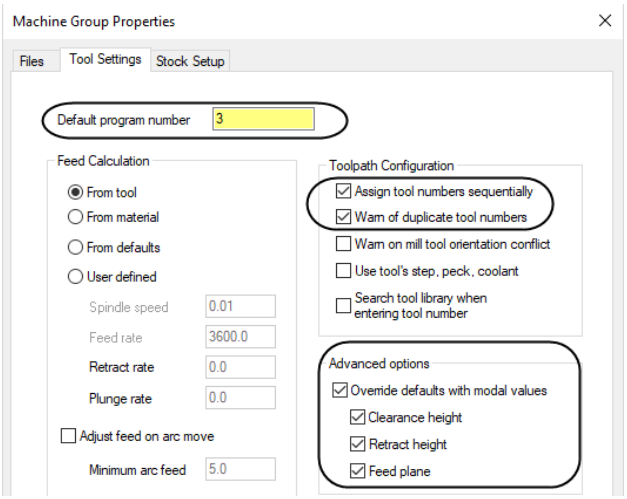
In this step you will supply Mastercam with information about tool settings, stock size and chuck jaw before starting to create the toolpaths.

2.1 Load the Lathe Default if needed



- From the **Machine** tab, select **Lathe** and **Default**.

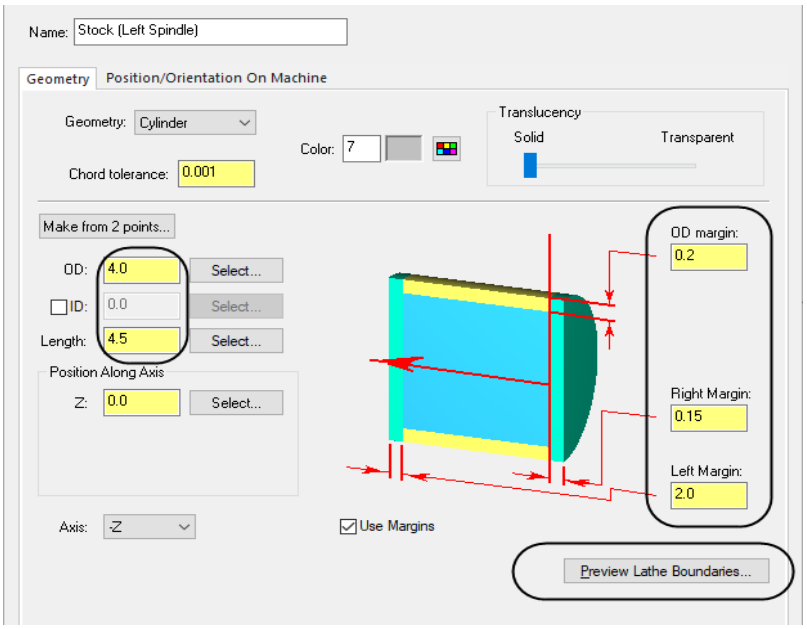
2.2 Set the Tool settings



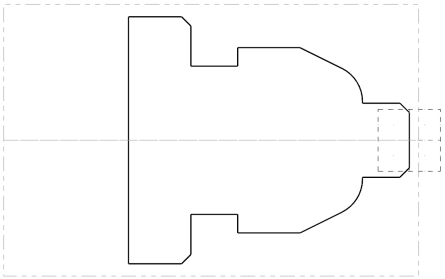
- Select the **Tool settings** icon.
- Set the **Program #** to **3**.
- Enable **Assign tool numbers sequentially**.
- Enable **Warn of duplicate tool numbers**.
- Enable **Feed Calculation From tool**.

2.3 Set the Stock settings

- Set the **OD** to **4.0"**.
- Set the **ID** to **4.5"**.
- Establish the stock as a bar giving extra margins: **OD margin = 0.2"**; **Right margin = 0.15"**.
- **Left margin = 2.0"**.
- Select the **Preview Lathe Boundaries** button to check the stock.
- Press **Enter** to continue.

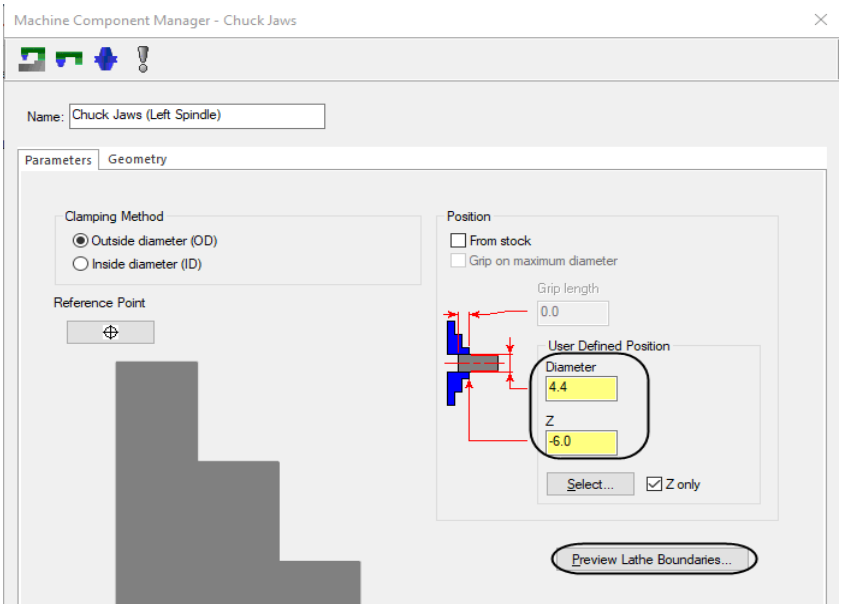


- Press **Enter** to continue.

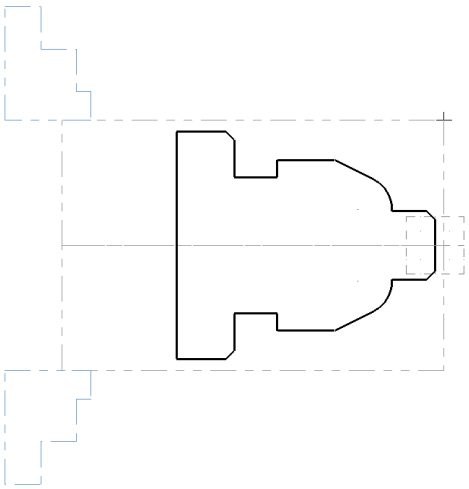


2.4 Set the Chuck Jaws settings

- **Clamping Method** set to **Outside diameter (OD)**.
- Set the chuck **Position** to **Diameter = 4.4"**; **Z = -6.0"**.
- Select the **Preview Lathe Boundaries** button to check the chuck.



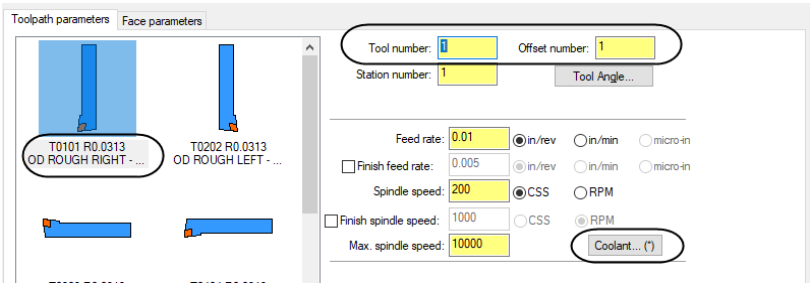
- Press **Enter** to continue.



STEP 3: LATHE FACE TOOLPATH

Turning

- From the **General** group, select the **Face** icon.

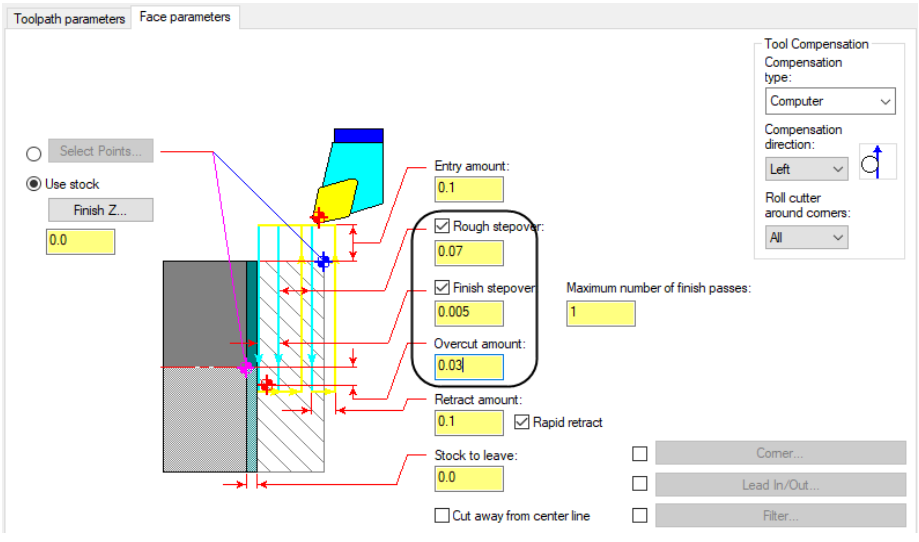


3.1 Set the Toolpath Parameters

- Select a **OD Rough Right 80 deg** tool.
- Set the **Tool/Offset Number** to **1**.
- Set the **Coolant On**.

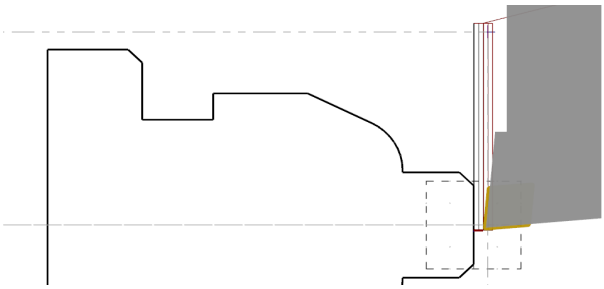
3.2 Set the Face Parameters

- Make sure that **Use stock** is enabled.
- **Finish Z = 0.0".**
- **Entry amount = 0.1".**
- **Rough stepover = 0.07".**
- **Finish stepover = 0.005".**
- **Overcut amount = 0.03".**
- **Retract amount = 0.1".**
- **Stock to leave = 0.0".**
- Set the **Cutter compensation** to **Computer** and to the **Left**.



3.3 Backplot the Face operation

- Press Alt +T to remove the toolpath display.



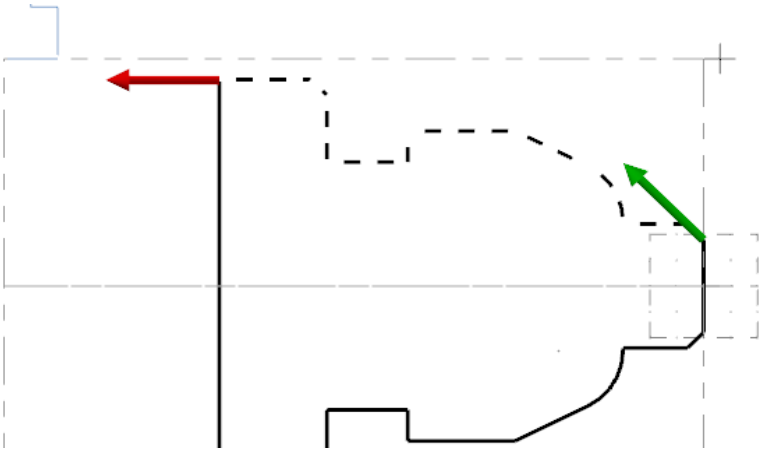
STEP 4: LATHE ROUGH TOOLPATH

Turning

- From the **General** group, select the **Rough** icon.

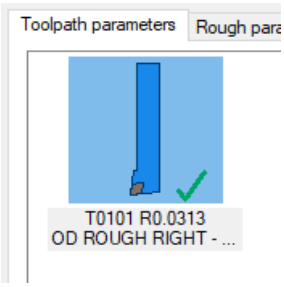
4.1 Select the geometry

- In the Selection Method, enable Partial button  and select the first entity of the chain and the last one to complete the chaining.



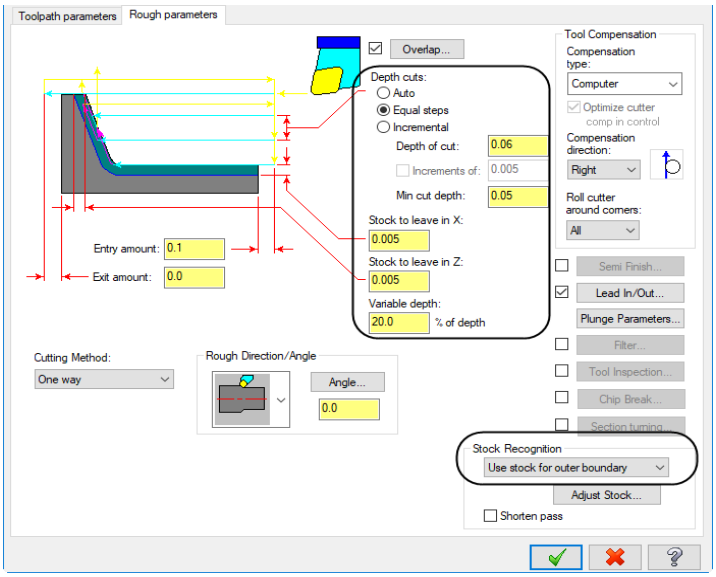
4.2 Set the Toolpath Parameters

- Select a existing **OD Rough Right 80 deg** tool.
- Set the **Tool/Offset Number** to **1**.
- Set the **Coolant On**.



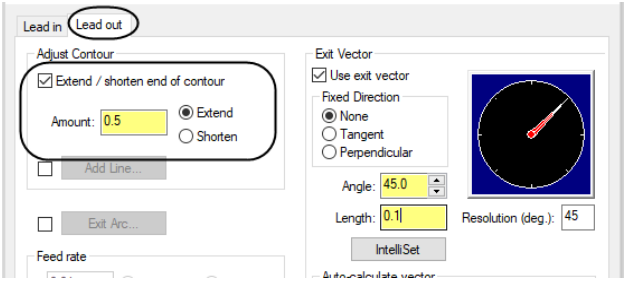
4.3 Set the Rough Parameters

- Make sure that **Overlap** is enabled.
- **Entry amount** = **0.1"**.
- Enable **Equal steps**.
- **Depth of cut = 0.06".**
- **Minimum cut depth** = **0.05"**.
- **Stock to leave in X** = **0.005"**.
- **Stock to leave in Z** = **0.005"**.
- **Variable depth** = **20%** of depth.
- Enable **Use stock for outer boundary**.



Note: Variable depth *allows you to vary the point that the surface contacts the tool insert to prevent notching and improve tool life. The variable depth can vary up to 25% of the depth of cut. The actual depth of cut can vary from 75% to 125% of the nominal depth of cut. The valid range is -25% to 25%. A positive value will result in an upward cut and a negative value will result in a downward cut. Zero will result in a straight cut.*

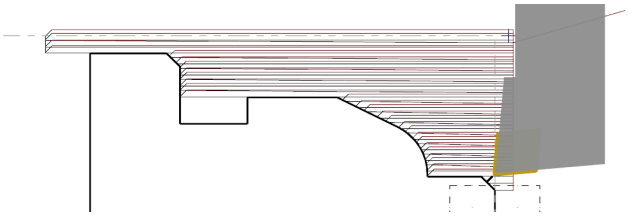
- **Cutting method** set to **One-way** and the **Rough Direction/Angle = 0**.
- Set the **Cutter compensation** to **Computer** and to the **Right**.



- Set proper **Lead In/out** parameters.
- **Extend** the end contour **0.5"**.

4.4 Backplot the Rough operation

Note: *The tool does not attempt to cut inside of the groove. This is a result of the* ***Plunge*** *parameters being set to none in the* ***Rough*** *parameters page.*

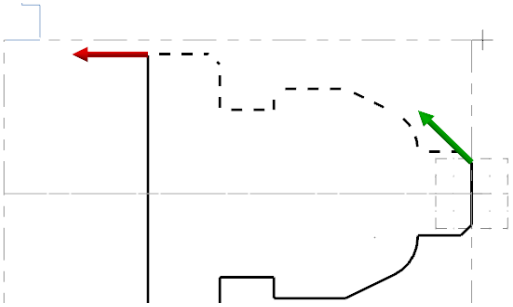


- Pres **Alt + T** to remove the toolpath display.

STEP 5: LATHE FINISH TOOLPATH

Turning

- From the **General** group, select the **Finish** icon.

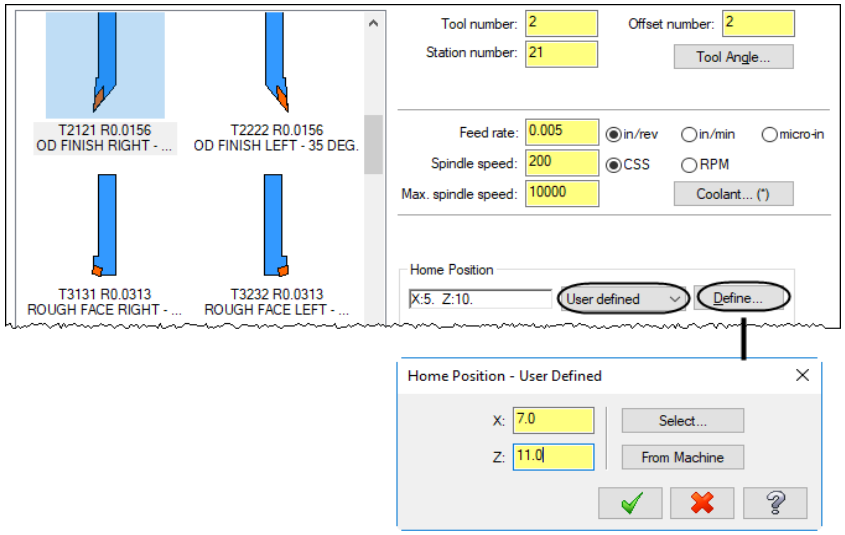


5.1 Select the geometry

- In the Wireframe Chaining dialog box, click on the **Last** button  . Mastercam selects the same chain as the one selected in the previous operation.

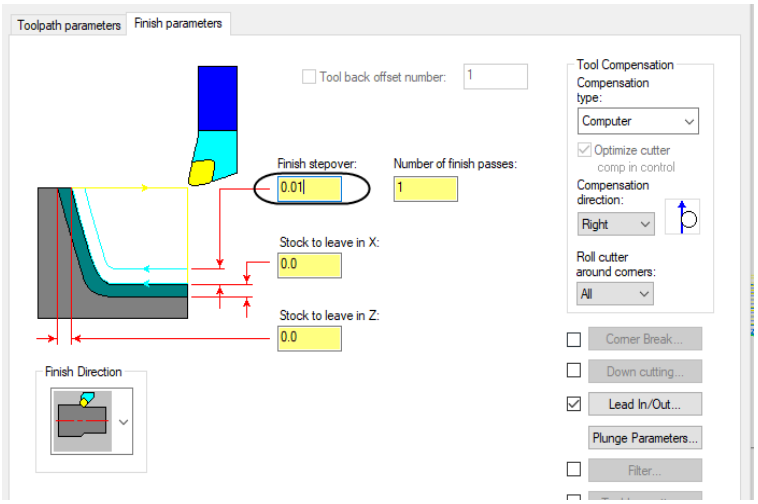
5.2 Set the Toolpath Parameters

- Select a **OD Finish Right 35 deg** tool.
- Set the **Tool/Offset Number** to **2**.
- Set the **Coolant On**.
- Set the **Home Position** as **User define; X 7.0, Z 11.0**.

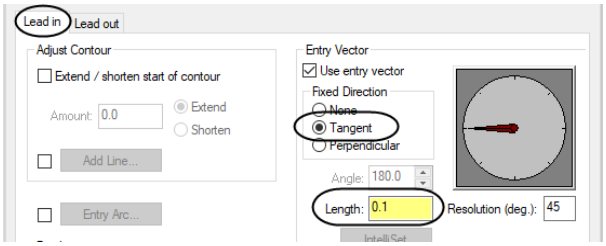


5.3 Set the Finish Parameters

- **Finish stepover** = **0.01"**.
- **Number of finish passes** = **1**.
- **Stock to leave in X** = **0.0"**.
- **Stock to leave in Z** = **0.0"**.
- **Finish direction** set to **OD**.
- Set the **Cutter compensation** to **Computer** and to the **Right**.

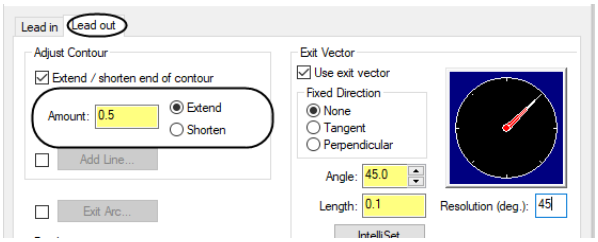


- Set proper **Lead In/Out** parameters.
- Use for **Lead In** only **Tangent entry; Length = 0.1"**.



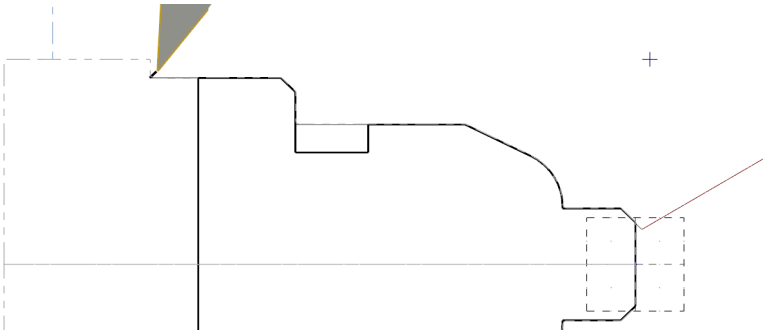
***Note:** This ensures that the entry will be extended with the length to better machine the chamfer.*

- **Extend** the end contour **0.5"**.
- Set proper **Lead In/Out** parameters; Use for **Lead In** only **Tangent entry; Length = 0.1"**.



- **Extend** the end contour **0.5"**.

5.4 Backplot the Finish operation



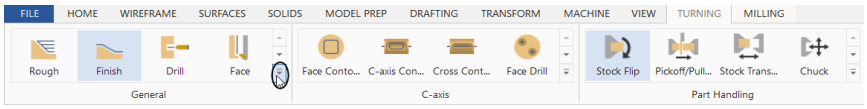
STEP 6: LATHE GROOVE TOOLPATH

Lathe Groove toolpaths are useful for machining indented or recessed areas that are not otherwise machinable by roughing toolpaths or tools.
Mastercam can machine many types of grooves based on the location of just a corner point or points; this lets you create groove toolpaths without having to create or chain geometry.
You can also use chained geometry if you prefer, or to specify complicated or intricate groove contours.

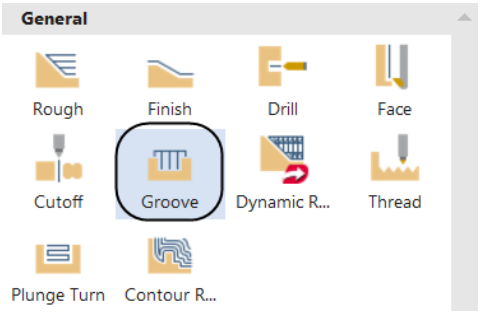
6.1 Select the toolpath

Turning

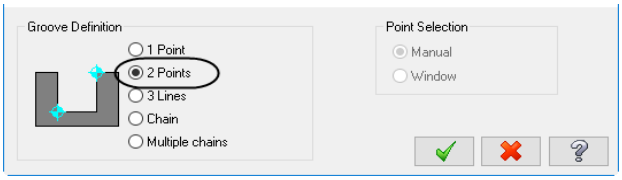
- From the **General** group, select the **Expand gallery** arrow as shown.



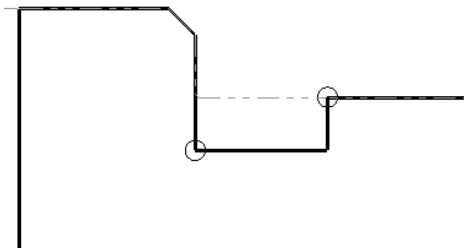
- From the **Toolpath gallery**, click on the **Groove** icon as shown.



- Select **2 Points** in the **Groove Definition**.



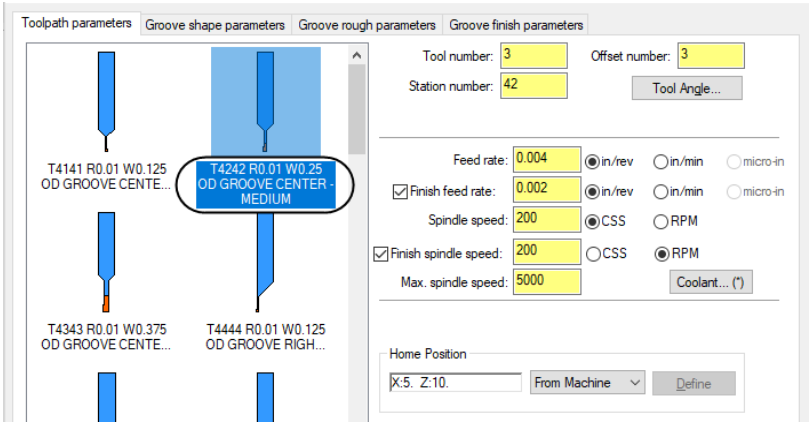
Note: *The first point should be the upper right corner of the groove and the second point should be the lower left corner.*



- Select the points as shown.
- Press **Enter** when finish.

6.2 Set the Toolpath Parameters

- Select a **OD Groove Center width** of **0.25"**.
- Set the **Tool/Offset Number** to **3**.
- Set the **Coolant On**.



In-House Solutions

Mastercam® 2020

TRAINING SOLUTIONS

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

TRAINING TUTORIALS

Our popular line of Mastercam Training Tutorials cover every Mastercam application, offering the most extensive collection of Mastercam training solutions available. Each book contains a series of tutorials that guide you through creating geometry, applying then verifying toolpaths and generating G-Code. Helpful hints, review exercises and quizzes complete the learning experience.

The Home Learning Edition (HLE) Demo Software is available for download with each eBook, and can be purchased on DVD for $5.

ARE YOU AN INSTRUCTOR?

Buy 10 or more of the Mill Essentials, Mill Advanced or Lathe Training Tutorials, get the corresponding Instructor Guide for Free! (see page 5)

Mill Essentials

The Mill Essentials Training Tutorial takes a very comprehensive look at 2D machining with Mastercam.

This book begins with the absolute basics by introducing the Mastercam user interface and will have you creating geometry, mastering 2D toolpaths and posting code before you know it.

- 7 Step-by-Step Tutorials with Review Exercises
- Includes Video Training DVD and Mastercam HLE Demo Software

Price $65 (eBook)
ISBN: 978-1-77146-870-1
Price $90 (Print)
ISBN: 978-1-77146-833-6

Lathe

The Lathe Training Tutorial is ideal for both beginners and current Mastercam users.

This book introduces 2D geometry creation and lathe 2D toolpaths providing an excellent foundation for many lathe applications.

- 7 Step-by-Step Tutorials with Review Exercises
- Includes Video Training DVD and Mastercam HLE Demo Software

Price $45 (eBook)
ISBN: 978-1-77146-872-5
Price $65 (Print)
ISBN: 978-1-77146-835-0

ORDER: 1.800.529.5517 • books@inhousesolutions.com • www.eMastercam.com

Mill Advanced

The Mill Advanced Training Tutorial picks up where Mill Essentials leaves off, introducing 3D concepts including surface high speed toolpaths.

It is strongly recommended that you have a good understanding of 2D machining before beginning this book.

- 6 Step-by-Step Tutorials with Review Exercises
- Includes Video Training DVD and Mastercam HLE Demo Software

Price $60 (eBook)
ISBN: 978-1-77146-871-8
Price $80 (Print)
ISBN: 978-1-77146-834-3

Multiaxis Essentials

The Multiaxis Essentials Training Tutorial is intended for the advanced Mastercam user looking to learn Mastercam multiaxis programming.

A firm grasp on both 2D and 3D machining is required which can be attained by completing the Mill Essentials and Mill Advanced Training Tutorials.

- 10 Step-by-Step Tutorials
- Includes Video Training DVD and Mastercam HLE Demo Software

Price $85 (eBook)
ISBN: 978-1-77146-878-7
Price $90 (Print)
ISBN: 978-1-77146-841-1

Multiaxis Advanced

The Multiaxis Advanced Training Tutorial is intended for the advanced Mastercam user.

This book contains projects demonstrating a variety of Multiaxis Advanced milling applications such as pattern application toolpaths.

- 6 Step-by-Step Tutorials
- Includes Mastercam HLE Demo Software

Price $75 (eBook)
ISBN: 978-1-77146-879-4
Price $80 (Print)
ISBN: 978-1-77146-842-8

Lathe C & Y Axis

The Mastercam 2020 Lathe C & Y Axis Training Tutorial is intended for the advanced lathe user that wants to learn how to program lathe parts that include milling operations.

Advanced C-Axis toolpaths as well as the Y-Axis rotation will be described in this book. Examples on how to use the Mill toolpaths on a Lathe with Live Tooling are also incorporated.

Price $65 (eBook)
ISBN: 978-1-77146-887-9
Price $85 (Print)
ISBN: 978-1-77146-848-0

Solids

The Solids Training Tutorial provides concise step-by-step instructions on creating and manipulating 3D wireframe and solid geometry.

This book details commands such as extrude, loft, revolve, sweep, boolean add, fillet, chamfer and more.

- 9 Step-by-Step Tutorials with Review Exercises
- Includes Mastercam HLE Demo Software

Price $40 (eBook)
ISBN: 978-1-77146-873-2
Price $60 (Print)
ISBN: 978-1-77146-836-7

Mastercam for SOLIDWORKS

The Mastercam 2020 for SOLIDWORKS Programming Exercises book provides a comprehensive "hands on" method of learning Mastercam for SOLIDWORKS. You will learn how to program a variety of different parts that require most of the toolpath types available in Mastercam for SOLIDWORKS. Extensive emphasis is put on making parametric changes and toolpath updates to match the SOLIDWORKS model changes. Primary focus is on toolpath creation on SOLIDWORKS models.

Price $65 (eBook)
ISBN: 978-1-77146-881-7
Price $85 (Print)
ISBN: 978-1-77146-844-2

Router

The Router Training Tutorial offers detailed coverage of 2D geometry creation, drilling, contouring, pocketing, nesting, block drilling and importing graphics to machine.

Instruction on tool settings, stock setup and custom profile tool creation are also included.

- 8 Step-by-Step Tutorials with Review Exercises
- Includes Mastercam HLE Demo Software

Price $65 (eBook)
ISBN: 978-1-77146-876-3
Price $85 (Print)
ISBN: 978-1-77146-839-8

Beginner

The Beginner Training Tutorial provides a comprehensive step-by-step approach to learning the basics of three Mastercam modules: Mill Essentials (2D), Lathe and Solids.

It is an excellent choice for new Mastercam users looking to get a broader overview of the software.

- 8 Step-by-Step Tutorials with Review Exercises
- Includes Mastercam HLE Demo Software

Price $65 (eBook)
ISBN: 978-1-77146-874-9
Price $85 (Print)
ISBN: 978-1-77146-837-4

Design

The Design Training Tutorial provides a comprehensive step-by-step approach to learning geometry creation within Mastercam.

Its focus is 2D and 3D geometry creation with explanations given on how to dimension the parts.

- 12 Step-by-Step Tutorials with Review Exercises
- Includes Mastercam HLE Demo Software

Price $45 (eBook)
ISBN: 978-1-77146-875-6
Price $65 (Print)
ISBN: 978-1-77146-838-1

CAD Import & Mill Essentials

The CAD Import & Mill Essentials Toolpaths Training Tutorial is intended for anyone looking to understand the ins and outs of Mastercam Mill Essentials toolpaths, while learning the best practices for importing geometry from various CAD software packages.

- 7 Step-by-Step Tutorials with Review Exercises
- Includes Mastercam HLE Demo Software

Price $70 (eBook)
ISBN: 978-1-77146-878-7
Price $90 (Print)
ISBN: 978-1-77146-846-6

CAD Import & Mill Advanced

The CAD Import & Mill Advanced Toolpaths Training Tutorial consists of 6 projects and 6 accompanying practice exercises. It is intended for intermediate to experienced Mastercam users who are primarily importing 3D geometry from another CAD software package.

- 6 Step-by-Step Tutorials with Review Exercises
- Includes Mastercam HLE Demo Software

Price $60 (eBook)
ISBN: 978-1-77146-884-8
Price $80 (Print)
ISBN: 978-1-77146-847-3

Wire

The Wire Training Tutorial provides users with an excellent resource for learning how to use Mastercam to program wire EDM machines.

In addition to geometry creation, the book focuses on wirepaths for dies, taper angle projects and more.

- 6 Step-by-Step Tutorials with Review Exercises
- Includes Mastercam HLE Demo Software

Price $40 (eBook)
IBSN: 978-1-77146-877-0
Price $60 (Print)
ISBN: 978-1-77146-840-4

Project Workbook

The Mastercam 2020 Project Workbook is ideal for High School students, hobbyists and those who prefer engaging in project-based learning. The Workbook includes an overview of CAD/CAM and basic machining followed by a series of step-by-step projects for both mills and lathes. *Note: no machining instructions included.*

- Five projects included: Art Lithophane, Bowling Pin, Shield, F1 Car, and Jewelry Box
- Includes Mastercam HLE Demo Software

Price $60 (eBook)
ISBN: 978-1-77146-880-0
Price $75 (Print)
ISBN: 978-1-77146-843-5

Nesting eBook

This tutorial includes a variety of projects that are using Nesting options in Mastercam to fit parts onto a sheet of material for best yield.

You will learn how Nesting operates on geometry and how you can create and use tabs to assist in holding down the nested parts. You will also learn how Nesting operates on toolpaths and how you can use WCS (Work Coordinate System) to set the parts of an assembly in the proper view for machining.

Price $35
(eBook only)
ISBN: 978-1-77146-885-5

SITE LICENSE

An eBook site license allows an educational facility to purchase one or several of our Training Solutions in eBook format (PDF). This in turn grants lifetime access to the content from any workstation on campus for unlimited users.

For more information on eBook Site Licenses, email **eBooks@inhousesolutions.com** or call **1.800.529.5517**.

INSTRUCTOR KITS

These are the ultimate toolkits for instructors looking to enhance their 2D mill, 3D mill and lathe classes while minimizing prep time. Each kit contains tests, quizzes, mid-terms, finals, powerpoint presentations, lesson plans, answer sheets, and more.

Instructor Guide & Training Tutorial are included in each kit. Each print kit includes an Instructor DVD, Video Training DVD, and Mastercam HLE Demo Software. Download links are provided for eBook kits.

eBook

Bundle of all three:
$200
ISBN: 978-1-77146-882-4

Print

$100 Mill Essentials Kit
ISBN: 978-1-77146-864-0
$100 Mill Advanced Kit
ISBN: 978-1-77146-865-7
$100 Lathe Kit
ISBN: 978-1-77146-866-4

Bundle of all three:
$250
ISBN: 978-1-77146-845-9

ORDER: 1.800.529.5517 • books@inhousesolutions.com • www.eMastercam.com

NEW!

MASTERCAM PROFESSIONAL CERTIFICATION CURRICULUM

This 3-part series is designed to help you prepare for your Professional Level Mastercam Certification™ Test. The skills exam must be administered by a Mastercam Certified Instructor or a Mastercam Certified Reseller.

Part 1 – Mill 2D

Part 1 – Mill 2D begins at the absolute basics by introducing the Mastercam user interface. It will have you creating geometry, drilling and creating contour toolpaths before you know it. Its gradual progression leads to more advanced concepts such as multiple setup scenarios and 2D High Speed Toolpaths. All of the parts within the book are designed with machinability in mind.

A multitude of topics are covered including 2D geometry, tool settings, stock setup, drilling, tapping, contouring, pocketing, circle milling. You will learn the 2D High Speed Toolpaths such as dynamic mill, area mill, dynamic rest mill, how to import a solid and machine it and how to use the WCS in multiple fixture applications.

Price $100

ISBN: 978-1-77146-867-1

Part 2 – Solids

Part 2 – Solids will teach a variety of useful solids commands, that include extrude, loft, revolve, sweep, boolean add and remove, fillet, chamfer, shell, trimming, and creating solid patterns. You will learn how to change a solids color and create a solid geometry from surfaces, as well as how to manipulate a solid by using commands from the Model Prep menu such as Push-Pull, Move and Split Solid Face. You will use create Layout and learn how to dimension parts to create blueprints.

Price $100

ISBN: 978-1-77146-868-8

Part 3 – Advanced Mill 2D

Part 3 – Advanced Mill 2D is dedicated to the power user. You will learn how to machine an imported solid model using the WCS for multiple setups. To better organize the parts, you will master how to use levels and view sheets. You will practice indexing a 4-axis part using the tool planes defined in the Plane manager. Fixtures will be used and the part will be align to their faces. You will learn how to save operations to a library and how to import them for different parts. A brief introduction to the basic 3D machining will be also covered.

Price $100

ISBN: 978-1-77146-869-5

Part 1, 2 & 3 Bundle

This part 1, 2 & 3 bundle provides you all three Mastercam Professional Certification Curriculum at a better price.

Price $250

ISBN: 978-1-77146-904-3

ORDER: 1.800.529.5517 • books@inhousesolutions.com • www.eMastercam.com

HANDBOOKS

The Mastercam 2020 Handbooks provide an academic approach to teaching the theory and application of Mastercam. The Handbook series is designed to teach the fundamentals of Mastercam, gradually working up to more complex material with each volume. Each Handbook comes with a Student DVD that contains files referenced within the book, and the Mastercam 2020 HLE Demo Software.

Handbook Volume 1
Mill 2D & Solids

The Mastercam 2020 Handbook Volume 1 is an excellent resource that teaches the theory of Mastercam Mill and Solids. The material intertwines conceptual subject matter with practical applications suitable for beginners and intermediates alike. You not only learn how to use Mastercam but why things work as they do.

Price $90 (eBook)
ISBN: 978-1-77146-893-0
Price $98 (Print)
ISBN: 978-1-77146-855-8

Handbook Volume 2
3D Modeling & Machining

The Mastercam 2020 Handbook Volume 2 takes an academic approach to teaching Mastercam 3D modeling and machining. The material is most suitable for intermediates (including individuals that have completed Volume 1). The book teaches more advanced CAD modeling techniques and explains surface creation.

Price $90 (eBook)
ISBN: 978-1-77146-894-7
Price $98 (Print)
ISBN: 978-1-77146-856-5

Handbook Volume 3
Multiaxis & Machining

The Mastercam 2020 Handbook Volume 3 is an excellent resource that teaches the theory of multiaxis machining with Mastercam. This book covers the classic family of multiaxis toolpaths including drill 5-axis, curve 5-axis, swarf 5-axis, multiaxis flowline and multiaxis multi-surfaces as well as the drill & circle mill family.

Price $90 (eBook)
ISBN: 978-1-77146-895-4
Price $98 (Print)
ISBN: 978-1-77146-857-2

FREE eBOOKS

How-To Tips Collection

The In-House Solutions support specialists take note of the questions that are asked. We realize that if you're asking a question there are ten more people who want to know the answer but haven't asked. Our goal is to make your life easier, so we've collected the answers to your most popular questions here, in one convenient eBook.

Price – FREE!
(eBook only)

Horizontal Machining with Indexing

The Indexing Training Tutorial explains how to use the Plane Manager for working with tool planes and work offsets.

Examples include horizontal machining using the WCS and a multiple fixtured tombstone project.

Price – FREE!
(eBook only)

Download at www.**eMastercam**.com/**freebooks**

Mastercam @Courses

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

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

LOG IN & LEARN TODAY!

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**

$149 each

ORDER: 1.800.529.5517 • books@inhousesolutions.com • www.eMastercam.com

Mill Essentials eCourse

The Mill Essentials eCourse introduces students to 2D CAD and milling toolpaths. It covers wireframe and solids creation as well as 2D mill toolpaths such as contour, drilling, blend, peel, dynamic area, transform, Feature Based Drilling, and more. This course serves as an excellent introduction to Mastercam.

Mill Advanced eCourse

The Mill Advanced eCourse builds on what students have learned in the Mill Essentials eCourse. It moves into more advanced CAD and demonstrated 3D wireframe, solid, and surface creation commands. 3 axis toolpaths such as Area Roughing, Dynamic OptiRough, Scallop, Pencil, Waterline, Radial, Hybrid, and more are covered.

Lathe eCourse

The Mastercam Lathe eCourse covers wireframe creation, working with imported part files, stock setup, facing, roughing, finishing, grooving, drilling, and cutoff toolpaths. Stock operations such as advance, flip, and tailstock are also covered. You will also learn how to program parts in a VTL.

Multiaxis Essentials eCourse

The Multiaxis Essentials eCourse covers 4 & 5 axis toolpaths. Toolpaths include contour with axis substitution, drilling with axis substitution, drilling with rotary axis positioning, rotary 4-axis, curve 5-axis, swarf 5-axis, drill 5-axis, circle mill 5-axis, flow 5-axis, and multisurface 5-axis. This course skips most CAD in favor of focusing on toolpaths.

www.**eMastercam**.com/**eCourses**

PROFESSIONAL COURSEWARE

The Mastercam Professional Courseware titles are intended for industrial training settings. Instead of step-by-step instructions, these books introduce concepts through sequences of specialized training exercises followed by parts the users are expected to produce with minimal guidelines.

Professional Courseware Mill Essentials

The Mastercam 2020 Mill Essentials Professional Courseware provides in-depth coverage of 2D wireframes and solids geometry, as well as contour, pocket, drilling, circle milling and slot milling toolpaths. More advanced exercises explain the use of the Work Coordinate System (WCS), 2D high speed toolpaths, Feature Based Machining (FBM) and more.

Price $60 (eBook)
ISBN: 978-1-77146-896-1
Price $70 (Print)
ISBN: 978-1-77146-858-9

Professional Courseware Mill Advanced

The Mastercam 2020 Mill Advanced Professional Courseware covers a multitude of features that teach a user to create 3D wireframes, surfaces and solids for 3D modeling and toolpaths. Interactive training exercises introduce 3D geometry functionality, while newer surface high speed toolpaths are thoroughly investigated along with their various parameter settings.

Price $50 (eBook)
ISBN: 978-1-77146-897-8
Price $60 (Print)
ISBN: 978-1-77146-859-6

Professional Courseware Lathe

The Mastercam 2020 Lathe Professional Courseware offers an in-depth look at Mastercam Lathe geometry and toolpath creation. Advanced toolpaths such as Misc Ops and C-Axis toolpaths are also described. Additional Mastercam files are provided along with guidelines for creating the toolpaths to machine each part.

Price $55 (eBook)
ISBN: 978-1-77146-898-5
Price $65 (Print)
ISBN: 978-1-77146-860-2

Professional Courseware Multiaxis

The Mastercam 2020 Multiaxis Professional Courseware details numerous toolpaths that allow a user to successfully machine 4-axis and 5-axis parts. Multiaxis Advanced toolpaths have been included with more complex parts along with instructions on how to machine them. Machine Simulation is used to check for any collisions between the part, the tool and any of the machine's components.

Price $65 (eBook)
ISBN: 978-1-77146-899-2
Price $75 (Print)
ISBN: 978-1-77146-861-9

Join the eMastercam community!

eMastercam is one of the largest and oldest online forums for swapping knowledge of CNC machines, tools, manufacturing processes and technology. After hours, eMastercam continues to be a place for Machinists, Engineers and others in the manufacturing industry to connect, share stories, opinions and get to know each other beyond the parts we make and the tools we use.

- Active user forums with an education-specific sub forum
- Free downloads and educational resources
- Free post processors and advanced development requests
- Overviews of Mastercam 2020 and related products
- The place to find our Mastercam Training Solutions
- Industry insights and much more!

www.**eMastercam**.com/**forums**

eMastercam.com
YOUR ONLINE MASTERCAM SOURCE

We don't just educate, we inspire!

eMastercam
We appreciate your feedback

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**

In-House Solutions

240 Holiday Inn Drive, Unit A
Cambridge, Ontario, Canada N3C 3X4
T: 800.529.5517 F: 519.658.1335
books@inhousesolutions.com
www.**inhousesolutions**.com