

Mastercam X5 Lathe Training Course Outline, Duration: 4 Days

Day 1: Design

Function/Command	Description
Interface Layout	Explaining the Graphic User Interface and how you can Navigate through Mastercam and setting the toolbar state.
Create Point	Learn to create points using the AutoCursor feature or by entering coordinates.
Create Rectangle	Learn to create rectangles knowing the width, the height and the anchor location.
Create Rectangular Shapes	Learn to create rectangular shapes knowing the width, the height and the anchor location.
Create Lines	Learn to create lines using some of the 6 available commands.
Create Arcs	Learn to create arcs using some of the 7 available commands.
Create Fillet/Fillet chains	Learn to create fillets using both fillet entity and fillet chain commands.
Create Chamfer/Chamfer chains	Learn to create chamfers using both chamfer entity and chamfer chain commands.
Edit Trim/Break/Extend	Learn to trim, extend 1, 2 or 3 intersecting entities, to trim, break or extend an entity to a specified length and how to divide/delete entities at the nearest intersection
Edit Break at Intersection/Join	Learn to break entities at intersections and how to join collinear entities or entities that are parts of the same circle.
Xform Mirror	Learn to create mirror images of existing geometry with respect to a defined axis or points.
Xform Offset/Offset contour	Learn to move or copy one entity or a chain of entities at a given distance, creating multiple copies on either side of the selected geometry.
Xform Translate	Learn to move copy or join entities within the same view without altering their orientation, size, or shape.
Xform Scale	Learn to increase or decrease the size of entities by a factor relative to a defined point.

Day 2: Design

Function/Command	Description
Create Drafting	Learn how to create dimensions, leaders and notes and how to edit the drafting entities.
Analyze commands	Learn to view entity properties and edit some or all of the data. Learn to measure the distance between two entities, determine the position of a point, and analyze an entire chain or contour.
Levels	Learn to work with levels and how you can manage them.
Attributes	Learn to set the level, color, line style and width and how you can modify them using attributes.
Importing files	Learn how to convert a drawing from a different CAD package.
Customization	Keyboard shortcuts, toolbars and right click menu.
System Configuration	Learn how to set Mastercam startup defaults.

Day 2: Lathe Toolpaths

Function/Toolpath	Description
Chaining geometry	Learn the different options from the chaining dialog box.
Fixing chaining problems	Learn to clean up the chain by: removing duplicate or overlapping entities, projecting existing geometry in one plane, trimming entities.
Machine definition, Control definition and Post processor	Learn the role of Machine definition, Control definition and Post processor.
Toolpaths Operations Manager	Learn the role of the Toolpath Manager to generate, sort, edit, regenerate, backplot, and post any operation.
File, Tool settings, Stock setup, Chuck and tailstock	Learn to setup the parameters for each dialog box.
Lathe Face Toolpath	Learn how to face the part to prepare the face of the part for further machining.
Lathe Rough Toolpath	Learn to quickly remove large amounts of stock in preparation for a finish pass.
Lathe Finish Toolpath	Learn how to finish lathe profiles both ID's and OD's.
Lathe Drill Toolpath	Learn how to drill holes using different types of drill cycles.
Backplot and Verify	Learn how to use Backplot/Verify to check the path the tools take to cut the part.

Day 3: Lathe Toolpaths

Function/Toolpath	Description
Lathe Groove Toolpath	Learn how to machining indented or recessed areas that are not otherwise machinable by roughing toolpaths or tools.
Lathe Thread Toolpath	Learn how to create spiral shapes on a part to make a screw, bolt, or nut.
Lathe Cutoff Toolpath	Learn how to vertically cut off pieces of the part, such as sections of the bar stock.
Lathe Quick Toolpaths	Learn how to use the three quick toolpaths (rough, finish and groove) which let you create simple toolpaths by entering only a few parameters.
Lathe Canned Toolpath	Learn to create canned toolpaths (shorter NC programs that can be quickly modified by the operator at the controller).
Lathe Stock Transfer	Learn to create an operation which transfers the stock to a chuck on the other spindle.
Lathe Stock Flip	Learn to create a stock flip operation.
Lathe Stock Advanced	Learn to create a stock advance operation to reposition the stock in the spindle or to control a bar feeder.
Lathe Tailstock	Learn to create a tailstock operation to reposition the tailstock.
Create Turn Profile	Creates a 2D profile in the Top view from an existing solid, solid face or surface.
Certification	

Day 4: C-Axis Geometry / Toolpaths (optional)

Function/Command	Description
Gview, Cplane and Tplane	Explaining the difference between Graphics view , Construction plane and Tool plane .
Standard planes	Learn to set the Cplane and the Z depth of the Cplane using standard views.
(Optional) Revolved surface	Learn to create surfaces that are circular in one direction. .
Face Contour	Learn to create a toolpath on the face of the part with the tool parallel to the axis of rotation.
Cross Contour	Learn how to create a toolpath parallel to the axis of rotation which is used primarily to cut slots.
C-axis Contour	Learn to create a toolpath that wraps around a cylinder (for example, text on a round part).
Face Drilling	Learn to drill holes into the face of the part. .
Cross Drilling	Learn to drill holes that are perpendicular to the axis of rotation, as when drilling holes in a cylinder. You can also select arcs which lets you drill off-center, i.e., in a direction that is not towards the center of rotation
C-axis Drilling	Learn to drill holes that are perpendicular to the axis of rotation, as when drilling holes in a cylinder.
Mill Contour	Allows you to machine a contour in any plane, but unlike the other three C-axis contour toolpaths, you have to set the plane before creating the toolpath
Mill Drilling	Allows you to drill holes in any plane, but unlike the other three C-axis drill toolpaths, you have to set the plane before creating the toolpath