Mastercam 2023 Training Guide



MILL-TURN LESSON-2

camInstructor

Objectives

This lesson will add to what we learned in Lesson 1 by using some simple milling on both the main and sub spindles. We will still use some of the turning techniques from lesson 1 along with the POCO and syncing.

Rough net shape by turning:

Mastercam will create a turning profile of the net part shape we can use to rough the hex with a turning tool.

Job setup:

Make adjustments in the Job Setup for Mill-Turn.

Main spindle turning:

Face with lower turret. OD rough with upper B axis head. OD finish with lower turret.

Main spindle milling:

Machine the hex with C-Axis rotation. Spot drill thru hole with milling spindle. Drill thru hole with turning spindle.

POCO (Pick Off Cut Off):

Transfer the part to the sub spindle.

Sub spindle turning:

Face with lower turret. Rough and finish the pocket with C-Axis rotation

Syncing:

Syncing ops to create efficient programs and avoid crashes.

Simulation:

Final verification of the machining process using complete machine verification.

Posting:

Additional settings before posting and view code in dual stream

MILL-TURN LESSON-2 DRAWING



MILL-TURN LESSON-2- THE PROCESS

- **TASK 1:** Import the solid model and select the machine type
- TASK 2: Job setup
- **TASK 3:** Main spindle turning
- **TASK 4:**Main spindle milling
- TASK 5: POCO (Pick Off Cut Off)
- **TASK 6:** Sub spindle turning and milling
- TASK 7: Machine syncing
- **TASK 8:** Machine simulation
- TASK 9: Posting code

TASK 1: IMPORT THE SOLID MODEL AND SELECT MACHINE TYPE

- We will start Mastercam in a **blank document** then open our solid model followed by selecting our Mill-Turn machine from the Machine Type menu.
- 1. Launch Mastercam.
- 2. Open the solid for Lesson 2. **millturn lesson2.x_t** If you have the online course download this file from the website.







 Select the Mill-Turn machine. Machine > Mill-Turn > Generic Fanuc Mill-Turn LTZ.machine. Mastercam will now launch Mastercam Code Expert which we will be using later.

06	R 💕 -	8	-	5 6	Ŧ								C:	Users\cdav	va\Download
File	Hom	ne V	Virefram	e S	urfaces	Solids	Model	Prep	Mesh	Drafting	Tra	ansform	Art	Machine	View
Ţ		μ <mark>Έ</mark> ς	X	Ξ	4		Σ		Ŧ	\approx	1		G1	I.	R Image (
Mill	Lathe N	Aill-Turn	Wire •	Router	Design	Control Definition	Machine Definition	Materia	Mill Tool Manager	Backplot	Verify	Simulate	Generate	Create	
		C:\U	sers\Pu	blic\Docu	ume\Ger	neric Fanuc I	Mill-Turn L	TZ.machi	ne	Sir	nulator	Γ ₃	Post		Setup Sheet
Toolp	aths	Man	age List	t											
▶ _k × _k	ĭ⊳ ĭ×	转-	8	• 📸	G1 Ĩ→	70									

4. Mastercam will now automatically launch the **Job Setup** dialog box.

Tool Settings Job Setup Machine Configuration Work Holding Work Support Setup Type WCS Left Spindle Part Geometry Bar Stock Toolplane Origin Z StickOut Dicht Spindle	Work Holding Left Spindle Default 10in_Chuck_Le Default Chuck Jaws Right Spindle Default Sin_Chuck_Righ Default Chuck Jaws	ft s (Left Spindle) nt s (Right Spindle)
Toolplane Origin Z	Left Spindle Options Minimum spindle speed: Maximum spindle speed:	0
Quick View Settings	Chuck angle about C axis:	0.0
Setup Type Continuous B Initial Spindle Left Spindle Spindle Separ 30.866142	Clamping distance: Right Spindle Options	0.0
WCS Top Part Length 0.000000	Minimum spindle speed:	0
Stock Type Bar Stock Pickoff Position 15.800000	Maximum spindle speed:	7000
Cutoff Width 0.000000 Left Spindle Z 7.900000	Chuck angle about C axis:	0.0
Front Face Sto 1000000000 Back Face St 10000000000 Right Spindle Z 55.327100 Part Stickout 0.000000	Clamping distance:	0.0

TASK 2: JOB SETUP

➔ In this task you will work through the Job Setup dialog.

To simplify job setup, Mastercam Mill-Turn uses a top to bottom series of steps. The order of these steps serves two main purposes.

- 1. Makes it easier to for the end user and guide them through the process.
- 2. Help Mill-Turn automate the display of appropriate and logical parameters based on previous settings.

This interface workflow reduces the chance that a user will enter a parameter that does not make sense and limits instances of error.

The general Job Setup workflow:

- Work Holding Select chucks, jaws, and collets
- Work Support Select steady rest when applicable
- Setup Type Assign the spindle the work will start in and part handling
- S WCS Sets the Work Coordinate System
- Left Spindle
 - Part Geometry Define your part
 - Bar Stock Define the type of stock

 \circ $\,$ Toolplane Origin Z – Set toolplane origin based on part geometry

- Stick Out Define bar stock projection from chuck face or jaws
- Right Spindle

 Toolplane Origin Z Set toolplane origin based on part geometry
- Pickoff Set pickoff position and tool for cut off

Note: Left spindle and right spindle order are determined by the initial spindle selection under Setup Type.



1. Work Holding – Use the default values for left and right spindle chucks.



2. Work Support – Leave work support empty



3. Setup Type – Initial Spindle set to Left and Stock Type – Part Handling set to Single Pieces of Stock - Pickoff.

ichine Group Pro	perties	
es Tool Settings	Job Setup	
Machine Co Work Ho Work Satup Type WCS Left Spindle Part Geo Bar Stoc 2 Toolplar StickOut Right Spindl 2 Toolplar Pickoff	nfiguration Jiding pport metry k e Origin Z le e Origin Z	Setup Type Initial Spindle © Left spindle © Right spindle
Quick View Settings	i i	Stock Type - Part Handling Single Pieces of Stock - Pickoff
Setup Type Initial Spindle Spindle Separat WCS Part Length Stock Type Pickoff Position Cutoff Width Left Spindle Z Front Face Stock	Single Pieces o Left Spindle 30.866142 Top 0.000000 Bar Stock 15.800000 0.000000 7.900000 10000000000	Continuous Bar Stock - Pickoff, Stock Pull, Cutoff Continuous Bar Stock - Pickoff, Cutoff Single Pieces of Stock - Pickoff Single Pieces of Stock - Pickoff Single Pieces of Stock - No Pickoff Two Separate Parts

Mill-Turn Lesson-2 - 7

4. WCS - Ensure the work coordinate system is set to TOP

Machine Group Properties			×
Files Tool Settings Job Setup			
Machine Configuration Work Holding Work Support Setup Type VCS Left Spindle Part Geometry	Group name: WCS Top	Machine Group-1	

5. **Part Geometry** – Select your part model. Mastercam will pull dimensions from the model for other Job Setup settings. The model will also be used in simulation and in the Sync Manager.



6. Select the solid by clicking on it. Hit End Selection.



Mill-Turn Lesson-2 - 8

7. Enable the options to Create Right Spindle Geometry and Create turn profiles. For the geometry, select Offset by and enter 100. This will put this model on the same level as the original model +100. So, if the original is on level 1 the created model will be on 101. For the turn profile, set computation method to Spin and set the Level to 20. Changing the color for the turn profile to something unique will help to distinguish it later.

Machine Group Properties		×
Files Tool Settings Job Setup		
Machine Configuration Work Holding Work Support Setup Type KCS	Part Geometry Now Delete previously created geometry	
Left Spindle Part Geomatry Bar Stock Stock Stockout Right Spindle Yz Toolplane Origin Z Pickoff	Create Right Spindle Geometry Keep the same Copy to level:	Spin: The part geometry is rotated around an axis. The profile is generated from the furthest extents of the spin.
Quick View Settings	♥ Onset by:	Slice: The part geometry is sliced in the lathe ZX (WCS
Setup Type Continuous Bar Initial Spindle Left Spindle Spindle Separat 30.866142 WCS Machine Group	Computation method: Spin Slice Color: 9 Select	limited to what the slice encounters and may not include the full extents of the
Part Length 5.00000 Stock Type Bar Stock Pickoff Position 15.800000 Cutoff Width 0.000000 Left Spindle Z 7.900000	Level: 20 Select	part geometry.
Front Face Stock 100000000000 Back Face Stock 100000000000 Right Spindle Z 55.327100 Part Stickout 0.000000		
< >		
	o 0 8	

 Bar Stock – You can set your stock type, number of parts, outside diameter, inside diameter and amount of extra stock here. Mastercam will read the OD from the model and round up to the next closest stock size. We are not going to turn the OD so set it back to 2.00. Add 0.05 extra stock to the face and to the back.

achine Group Prop	perties					$\times \parallel$	
les Tool Settings	Job Setup						
Machine Co Work Ho Work Sup Setup Type WCS	nfiguration Iding pport	Stock Geometry:	Bar Stock	~			Geometry selection is based on previou choices made in Jo
Part Geo	metry	Outside Diamete	r				Setup. Since we
	k e Origin Z	Diameter:	2.0	Ø	-	- 11	have selected Sing
StickOut 	e e Origin Z	Extra stock:	0.0				Pieces of Stock – Pickoff in setup typ
Pickoff		Inside Diame	ter				we will use bar sto geometry.
		Diameter;	0.0	Ø	θ		0 ,
		Extra stock:	0.0				
Quick View Settings		Face / Back					
Setup Type Initial Spindle	Single Pieces o Left Spindle	Extra face	0.05	-			
Spindle Separat WCS	30.866142 Top	Extra back	0.05	-			
Part Length Stock Type Pickoff Position Cutoff Width Left Spindle Z Front Face Stock Back Face Stock Right Spindle Z Part Stickout	2.875000 Bar Stock 15.800000 0.000000 7.900000 1000000000000 1000000000000 55.327100 0.000000						
<	>						
2	à						

9. Left Spindle – Toolplane origin Z. Set to Compute>Right Face.

Machine Group Properties		×
Files Tool Settings Job Setup		
Machine Configuration Work Holding Work Support Setup Type WCS Left Spindle Part Geometry Bar Stock	Tool Plane Origin Z Compute Left face Right face	
Toolplane Origin Z	OManual	

10. **Stickout** is calculated from the face of the chuck or jaw. We are using a single piece of stock so we need to leave enough material sticking out for the pickoff. Part length is 2.875 with 0.05 stock on front and back face. Use a part Stickout of **4.5**. Enter this value in the **Chuck Face:** box.



11. Pickoff – We need to calculate our pick off point. The hexagon on the right is 0.88 long, plus the chamfer of 0.030 so we need to be sure to stay clear of that. We should be safe grabbing the part close to the mid-point. Select the midpoint of the 2.00" diameter stock. Distance from chuck face should be 2.625.

Hit OK	
Machine Group Properties	×
Files Tool Settings Job Setup	
Machine Configuration Work Holding Work Support Setup Type WCS Part Geometry Bar Stock Toolplane Origin Z Pickoff Distance from chuck face: 2625 C Distance from chuck face: 2625 Pickoff Distance from chuck face: 2625 Pickoff Pickoff Pickoff Pickoff Pickoff Pickoff	
Select Pickoff Z Positi	on
Setup Type Single Pieces o Initial Spindle Left Spindle Spindle Separat 30.866142 WCS Machine Group Part Length 2.875000 Stock Type Bar Stock Pickoff Position 6.562008 Cutoff Width 0.000000 Left Spindle Z 3.937008 Front Face Stock 8.987992 Back Face Stock 6.012992 Right Spindle Z 33.936024 Part Stickout 4.500000	
< >>	
O	0

12. **Review of Job Setup Results.** Look at what Mastercam has done so far because of the Job Setup inputs. Mastercam creates a **new WCS** which is a copy of WCS-Top and has the origin set at machine zero as defined in your machine setup. You can also see **chuck** and **stock boundaries** on **both spindles** as well as the new solid. These are generated using inputs from the Job Setup.



13. Mastercam will also **create view sheets** for each spindle. You may need to turn viewsheets on. To do so, **View > Viewsheets > On/Off**

View	Turning Milling				
Advanced Display 👻	Image: Toolpaths Image: Levels Image: Groups Image: Solids Image: Multi-Threading Image: Recent Functions Image: Planes Image: Art	Show Show Axes Gnomons	w Show Snap Grid to Grid	Rotation Position On/Off	New
Toolpaths 🕞	Managers	Display	rs Grid rs	Controller	Viewsheets 5



Mill-Turn Lesson-2 - 12

14. Mastercam also **creates planes** that relate to standard programming orientations for both the main (left) and sub (right) spindles. The origins of these planes are keyed to the locations selected in the Job Setup.

		0	. 11	- t	2.	0	
Nan		6	WCS	C	т	Offset	Displa
J	Top	9	mes			onset	Dispid
	Front						
	Back						
1	Bottom						
	Right						
	Left						
	Isometric						
	Isometric reverse						
	Trimetric						
1	Tool Spindle - Machine Orient.		E.				
1	Machine Group-1 WCS	G	WCS	С	т		
1	Machine Group-1.Turning.Left.Uppe					0	
1	Machine Group-1.Turning.Left.Low					0	
1	Machine Group-1.Face Mill.Left 1					0	
1	Machine Group-1.Face Mill.180.Left 1					0	
1	Machine Group-1.Cross Mill.Left 1					0	
1	Machine Group-1.Axis Sub.Left 1		-	Ne	w	Plan	ies
1	Machine Group-1.Turning.Right.Up					1	
1	Machine Group-1.Turning.Right.Lo					1	
1	Machine Group-1.Face Mill.Right 1					1	
1	Machine Group-1.Face Mill.180.Rig					1	
1	Machine Group-1.Cross Mill.Right 1					1	
1	Machine Group-1.Axis Sub.Right 1					1	
	Machine Group-1.Top Right Spindle	_				1	

15. The **names** of the views, WCS and Viewsheets are **keyed** to the **Group name** entered in Job Setup.

Machine Group Properties		•	×
Files Tool Settings Job Setup			
Machine Configuration Work Holding Work Support Setup Type Left Spindle Part Geometry Bar Stock	Group name: Machine C WCS Top	Group-1	

16. **Save** your file as MillTurnLesson2

TASK 3: MAIN SPINDLE TURNING AND THRU HOLE DRILLING

I In this task you will face the part and turn the OD profile in the main (left) spindle.

1. Facing the part. Select Turning > Face

File	Home	Wireframe	Surfaces	Solids	Model Pre	p	Mesh	Drafting	Transform	Art	Machine	View	Turning	Milling
Rough	Finish	Drill	Pinch Tur	n Fa	ice (Cutoff	*	Pickoff/Cut	Pickoff	Stock Pull	Tailstock (1	Stock Shading Di	Stock Sto splay
			General						Part	Handling				Stock
Toolpat ▶ _k × _k ∣ ĭı	hs Ix 🛱			20	* 4 >	<								

2. When the Lathe Face Properties window opens, select Axis Combination / Spindle Origin by clicking the small graphic. This will open the Axis Combination / Spindle Origin window. Select Lower Left as we will be using the lower turret to turn on the main (left) spindle. Then hit the green checkmark to accept.

olpath parameters	Face parameters			1					
P	p		î	Tool number.	1		Offset nur	mber. 1	ale.
	Axis Combination	/ Spindle Origin						TOOLAN	×
T0101 R0.0313 ROUGH FACE LE	Axis Combination		Sp	spindle Origin					1
				Name Machine Group-1.Face Mill.18	Origin Z 11.287008	Work Offset	Display M Radial		micro-in
	Upper Left	Upper Right		Machine Group-1.Turning.Righ.	. 30.974016	1	Radial		🔿 micro-in
T0303 R0.0154 OD FINISH LEFT	Lower Left	Lower Right							
6		2	X:	c 0.0 12 10	lork Offset Manual	0	Display M	ode	
T1212 R0.031; OD 55 deg Rig			Z:	2: 11.287008 Z only	Automatic		Radial	0 0	
Show library tools	i pol 🗸	Right-click for options							> 3
Axis Combination / S	pindle Origin								
Upper Left	in: Machine Group	-1. Turning.Left.Upper.		Iool Display		Stock Upd	ate		Refpoint_

3. On the **Lathe Face Properties** window choose **T0101**. We will not spend time adjusting speeds and feeds. Using defaults will suffice for these lessons.



Note: The default settings on the Face Parameters page will work for our setup. No need to make changes here. Check that yours are the same.



Mill-Turn Lesson-2 - 15

4. You can do a quick **backplot** to check your toolpath now. However, we will hold off on the complete verification until all ops are done.



5. Roughing the OD. Select Turning > Rough...

File	Home	Wireframe	Surfaces	Solids	Model Prep	Mesh	Drafting	Transform	Art	Machine	View	Turning
Rough	Finish	Drill	Pinch Turn	Fac	ce Cutoff	*	Pickoff/Cut	Pickoff	Stock Pull	nistock O		Stock Stock Shading Display
			General					Part	Handling			5

6. Chain from the 0.05 chamfer to the intersection of the taper and the OD. Make sure you chain the geometry on the upper side of the part. Hit the green check to accept.



7. The roughing will take place with the Upper Turret (B-axis head). Select an appropriate tool for OD roughing. Check that **the Axis Combination/Spindle Origin** is set to **Left Spindle Upper Turret**. The tools in the library are shown in the toolchange position, for our machine this is 0 degrees. Since we are turning the OD, we set the **Tool B Angle at 90.0** for the correct orientation.

Lathe Rough Toolpath parameters Rough parameters	×	🔓 🎼 AutoCursor - 🎿 🍗
T0101 R0.0313 ROUGH FACE LEF OD ROUGH LEFT	Tool number: 1 Offset number: 1 Tool Angles A angle: 0.0 B angle 90.0	M Tool Angle X Tool Angle Plunge Direction Feed Direction
T0303 R0.0156 OD FINISH LEFT OD 55 deg Left	Feed rate: 0.01 ● in/rev ○ in/min ○ micro-in ✓ Plunge Feed rate: 0.005 ● in/rev ○ in/min ○ micro-in Spindle speed: 200 ● CSS ○ RPM Max spindle speed: 5000	90.0 Tool Orientation on Machine
T1212 R0.0313 T2121 R0.0156 OD 55 deg Right OD FINISH RIGHT Y	Comment	Cutting Direction
Select library tool Image: Tool Eilter Axis Combination / Spindle Origin Upper Left Spindle origin: Machine Group-1. Turning.Left.Upper	To batch Tool Display Stock Update Ref point	
	Generate toolpath	

Mill-Turn Lesson-2 - 17

8. **Roughing Parameters**. Use the default settings. **Enable Lead In/Out** and set to appropriate values. Set Stock Recognition to **Use stock for outer boundary**. Green check to accept.



9. Again, you can run backplot for a quick visual of the toolpath.



10. Finish the OD. Select Turning > Finish...



11. Since we will be using the lower turret to finish the OD, select geometry on the **lower side of the model**. Chain the same section as the Roughing op. Green check to accept the chain.



12. Lathe Finish Properties. Select a finish turning tool and check Axis Combination is set to Left Spindle Lower Turret and Turret Index Position is set to Left spindle.



13. **Finish Parameters**. Default setting are good. Check that **Lead In/Out is enabled** and are set to appropriate values, keep in mind the tool orientation when setting your leads. Green check to accept.

🚮 Lathe Finish		×
Toolpath parameters Finish parameters		
Finish Direction	Tool back offset number. 5 Finish stepover. Number of finish passes: 0.1 Stock to leave in X: 0. Stock to leave in Z: 0.	Tool Compensation Compensation type: Computer Optimize cutter comp in Compensation direction: Left Roll cutter around corners: All Corner Break Down cutting Lead In/Out Plunge Parameters Filter Chip Break Extend contour to stock Adjust Contour Ends
	Generate toolpath	0 0

Mill-Turn Lesson-2 - 20

14. Run **Backplot** to verify the toolpath.

	Backplot ×		
+X			

15. Next, we will drill the $\frac{1}{2}$ " diameter thru hole with the lower turret. Since this will be a static tool, we need to select **Drill** from the **Turning menu**.

File	Home	Wireframe	Surfaces	Solids	Model Prep	Mesh	Drafting	Transform	Art	Machine	View	Turning
Rough	Finish	Drill		rn Fa	ce Cutof	^ • f =	Pickoff/Cut	Pickoff	Stock Pull	Tailstock O.		Stock Stock Shading Displa
		30 mg	General					Part	Handling			

16. Lathe Drill Properties. Select a ½" diameter drill and check Axis Combination is set to **Left Spindle Lower Turret** and Turret Index Position is set to **Left spindle**.

oolpath parameters G83	3/G87 Drilling Cycle					
T122122 0.25 Dia. DRILL .25 DIA.	7 123123 0.375 Dia. DRILL .375 DIA.	' Tool number	3	Offset	number. 3 Tool Ang	le
—						
T124124 0.5 Dia.	T125125 0.75 Dia.	Feed rate:	0.01) in/rev) in/min	() micro-ir
DRILL 5 DIA	DRILL .75 DIA.	Spindle speed:	200	Ocss	• RPM	
		Max. spindle speed:	5000			
T126126 1. Dia. DRILL 1. DIA	T127127 1.25 Dia. DRILL 1.25 DIA.	Force tool chan	ge			
Show library tools	Right-click for options					^
Select library tool	Tool <u>F</u> ilter					~
Axis Combination / Spin Lower Left Lindle origin: I	dle Origin Machine Group-1.Turning.LeftLower			Stock Update	To ba	atch Refpoint
		_				

17. **Drilling Cycle Parameters.** Change the depth to -3.0 to drill thru the stock, then check the **Drill tip compensation** box to ensure the drill goes all the way thru. Green check to accept.

oolpath parameters G83/G87 Drilling Cycle	Depth	Drill Cycle Parameters Cycle:	
	Absolute Incremental	G83/G87 Drilling Cycle	~
	Drill Point	First peck	0.0
X	0.0 Z: 0.0	Subsequent peck	0.0
		Peck clearance	0.0
	Clearance 0.25	Retract amount	0.0
100	Absolute Incremental	Dwell (W)	0.0
	From stock	Shift	0.0
	Retract 0.1		
	⊖ Absolute ● Incremental □ From stock ■ Bre	Drill tip compensation	

18. Next, we will chamfer the thru hole using the upper turret. Since this will be a rotating tool, we need to select **Drill** from the **Milling menu**.

File	Home	Wireframe	Surfaces	Solids	Model Prep	Mesh	Drafting	Transform	Art	Machine	e View	Turning	Milling
			-	^ *	4				*	6	4	1	1
Contour	Drill	Proamic N	Aill Face	Ŧ	OptiRough	Area Roug	Waterline	Raster	Ŧ	Curve	Swarf Milling	Unified	Deburr
		2D	•				3D				Mu	Itiaxis	

19. **Toolpath Hole Definition.** Select the outer diameter of the chamfer on the Z0 face. This will set the initial depth to the correct point on the drilling parameter page.

Toolpath Hole Defini	tion # ×		👔 🍂 AutoCursor - 🤹 🍗 🐂 👘 🌒
Selection Advanced Features	8 (S)	Select one or more entities to add to or remove from the Features Click or window select solid holes, solid arc edges, wireframe arcs, - [Ctrl+click] to select all matching radius solid features. - [Ctrl+Shift+click] to select all matching radius solid features on the	ist. lines, points or AutoCursor positions. he same vector as the initial selection.
Vipe Solid Arc 1	Diameter 0.55	 [Double-click] to select a solid hole. [Ctrl+Double-click] to select all solid holes of the same type. Click on a selected solid feature's arrow to change direction. 	
Depth Filters Use highest Z depth Use lowest Z depth Off	۲		
Sort Selected Order	<u>،</u>		

20. Next, we will create a tool. Highlight tool on the left side menu, then right click in the white box and select **Create tool**.



21. Select Spot drill from the menu, then click the Next button.



Define Tool						× 🛛 - 🛄 - 🔍 M	001)
Current Step: Select Tool Type	Define Spot drill						
Define Tool Geometry	Adjust geometric proper	ties used to define the too	I shape.				
Finalize Properties			1				
	Standard sizes))	0 0 L ·	Sci	alable	
			-uorealit		in an	anti	
	Size	Drill Diameter	Cutting Length	Overall Length	Tip Angle	Shoulder Length	Shank Diameter
	4 1/8 X 90	0.125	0.375	1.75	90	0.375	0.125
	3/16 X 90	0.1875	0.5625	2.25	90	0.5625	0.1875
	1/4 X 90	0.25	0.75	2.75	90	0.75	0.25
	5/16 X 90	0.3125	0.75	3	90	0.75	0.3125
	3/8 X 90	0.375	0.75	3.5	90	0.75	0.375
	= 1/2 X 90	0,5	1	4	90	1	0.5
	1 5/8 X 90	0.625	1	4.5	90	1	0.625
	3/4 X 90	0.75	1.125	5	90	1.125	0.75
	1.0 X 90	1	1.25	6	90	1.25	1
	Non-cutting geometry		•				
		4.0					
	Shoulder length:	1.5					
	Shank diameter:	0.25					
			200				
					0.4310 in	<u>u</u> 3	
			- LCC-	INES	Inch		

22. From the Standard sizes dropdown, select the 5/8 X 90 spot drill size.



Define Tool							>
Current Step: Select Tool Type	Define Spot drill						
Define Tool Geometry	Adjust geometric properties used to de	fine the tool shape.					
Finalize Properties							
	Standard sizes	۲	\$ \$ \$ °	0 L ·		Sci	alable
	5/8 X 90		GCEIL	668	a sinte		
	Overall dimensions	۲		[LILLS			
	Drill diameter:	0.625					
	Overall length:	4.5					
	Cutting lengths	1					
	Cutting length.						
	Tip treatment	۲			alit.		
	Tip apple	90	agau	ME			
	rip angle.	30		11.00			
	Non-cutting geometry	۲					
	Shoulder length:	1					
	Shank diameter	0.625					
	Shark diameter.	0.025					
					- dike	0.0740	.19
			alace	MEST		Inch	30
						4	
Help				Cancel	Back	Next F	Finish
	Work type IINDEETNED						

24. We will use the default properties for this lesson. Click the Finish button	to create the tool.
--	---------------------

Current Step:	Finalize miscellaseo	us properties				
Select Tool Type	This is a second s	as properties.				
Define Tool Geometry	Adjust any miscellaneous	properties before fin	alizing tool creat	ion.		
Finalize Properties				_		
	Operation		(General	۲
	Tool number:		4		Name:	0.625 5/8 X 90 Spot Drill
	Length offset:		4		Description:	
	Diameter offset:		4		Manufacturer name:	Mastercam - 🕂
	Head number:		0		Manufacturer's tool code:	
	SFM:		65.44503		Tool Grade:	Mastercam Default (* 🕂
	FPT:		0.0025			•
	Number of flutes:		2		Drilling	۲
	Feed rate:		2		Peck 1 (% dia):	0
	Plunge rate:		25		Peck 2 (% dia):	0
	Retract rate:		50	-	Peck clearance (%):	0
	Spindle speed:		400		Dwell:	0
	Spindle direction:	Clockwise	-		Chip break (% dia):	0
	Material:	Carbide			Cycle:	Drill/Counterbore *
		*		Ŧ		\

25. From the **Setup** menu on the left, select the **Machine Group-1.Face Mill.Left 1** from the dropdown.

2D Toolpaths - Drill/Circles	G83/G87 Drilling Cycle	×
Y 🔚 🖬 📑		
Toolpath Type Tool Holder Setup Stock Cut Parameters Tool Axis Control Linking Parameters Unking Parameters Mole Segments Linking Parameters Nef. Points Safety Zone Arc Filter / Tolerance	Mill type Image: C-Axis Face Image: C-Axis Substitution Image: C-Axis Face Image: C-Axis Substitution Image: C-Axis Face Image: C-Axis Cross Image: C-Axis Face <td>Upper Left Upper Right</td>	Upper Left Upper Right
Quick View Settings Tool 0.625 5/8 X 9. Tool Diameter 0.625 Corner Radius 0 Feed Rate 2 Spindle Speed 400 Tool Length 4.5 Length Offset 2 Diameter Of 2 Cplane / Tpl Machine Grou. Axis Combin Upper Left Tip comp Off Work twne Plane Rotation	Toolplane Machine Group-1.Face Mill.Left 1 Top Front Back Bottom Right Left Isometric Isometric reverse Trimetric Tool Spindle - Machine Orient. Machine Group-1.Face Mill.Bol.Left 1 Machine Group-1.Face Mill.Bol.Left 1 Machine Group-1.Face Mill.Bight 1 Machine Group-1.Face Mill.Bight 1 Machine Group-1.Cross Mill.Right 1	Spindle origin Machine Group-1.Turning.Left.Upper 1 Work Offset Manual Automatic
✓ = edited Ø = disabled	Machine Group-1.Top Right Spindle Turret Tool Locator - Machine Orient. Top:LEFT Top:LEFT:LEFT:B90C90	Generate toolpath 🧿 🔇 🔇

26. Jump down to **Linking Parameters** on the left side menu. Select the **Depth Calculator** button, set the **Finish diameter to 0.55**, select the **Overwrite depth** radio button, then select the green check.



27. Notice the **Incremental Depth** is now automatically calculated for you based on your finished diameter and cutting tool parameters. Green check to accept.



Mill-Turn Lesson-2 - 27



29. Save your file.

TASK 4: MAIN SPINDLE MILLING

- In this task you will mill the hex on the main (left) spindle.
- 1. To mill the hex, we will use a simple contour path using wireframe geometry. If you do not have wireframe geometry, create it first by selecting **Curve All Edges**, then selecting the entire solid model.

File	Home	Wireframe Surfaces	Solids Model Prep Mesh	Drafting	Transform Art Machine View Turning Milling	
Point Position V Poir	Bolt Circle	+ // Line Kallel Line _ Line Perpendular Endpoints /> Line Closest * Lines	Circle Center Point Circle Edge Point * Arcs	Spline Manual Splines	Rectangle Create Box Box Box Box Profile Grove Door Geometry Shapes	Curve One Edge All Edges by Plane ♥ Curves

2. Select Milling > Contour.



3. Chain the hex using wireframe geometry.



4. Select a 1/2 Bull Endmill with a 1/32 rad from the library. In the Setup page set the Mill Type to C-Axis Face. This will force the chuck to rotate and will also keep X above the centerline which is a must for some mill-turn machines. The output code will consist of C and X movements. If we had used Y-Axis Face the tool would profile the hex much the same as a milling machine would with a combination of X+Y movements.



5. Enable Lead In/Out and set appropriate values. On the Linking Parameters page make the changes below if needed. Green check to accept.



Mill-Turn Lesson-2 - 31



6. **Backplot** the milling operation and **save** your work.

7. With the operation set to C-Axis face it is not important where the chain starts. The start of the cut will always be rotated to the top. For example, copy the op we just made after the original. Right click/hold then drag to the red arrow and release the right button.



In this new operation (7-Contour (2D)) select the geometry to open the Chain Manager. Right click Chain 1 and select Start point.



Move the start point, accept the changes then rebuild the op.





With the toolpaths displayed you can see how they look different.

Take note of the jaw position before starting a Backplot on this new op. When you start the Backplot you will notice the jaws have indexed to move this cut start point to the top. Note: You may need to check the box for **Simulate Rotary Axis** in the Backplot Options for this to display correctly.



8. Delete this extra op (7-Contout (2D)). It was just for demonstration. Resave your file.

TASK 5: POCO – PICK OFF CUT OFF

- S In this task you will pick the part off and transfer it to the sub (right) spindle.
- 1. From the menu bar select **Turning > Pickoff** found under the **Part Handling** section. This will launch the Pick off, Cut off dialog box.



2. **Pickoff.** Since we are using a single piece of stock and not barstock, set the Strategy to Pickoff. Changes can be made to the operations parameters if needed, however the events and their order cannot be altered. Users can define their own strategies if specific part handoff sequences are needed. These new strategies will appear in the Strategy list. Green check to accept and create the POCO routine.

Note: The Pickoff/Cutoff routine is not available since our job setup settings do not support that operation for this part.

Strategy:	Pickoff (Upper Stream)	Name	Pickoff - Upper Stream	
perations	Pickoff (Upper Stream) Pickoff (Lower Stream)			
Park Lowe	r Turret Under Left Spindle dles ndle - Unclamp and eject part	Comment: Park Lower Turret Un	nder Left Spindle	< >
Pickoff spi	ndle - Move to grip position ndle - Clamp and transfer stock	Turret Park Turret:	Lower Turret	~
Stock spin	dle - Unclamp ndle - Retract er Turret Home	Destination:	Lower Park Left	~
		✓ Linear		
		Movement type: X: 0.0 Y: 0.0 Z: -14.76378	Z first Offset from home position Offset from home position Offset from home position	~
<	>	B: 0.0	Axis not defined	
		Tool Call Index position:	9	

3. **Results of POCO** dialog input. Mastercam creates the handoff operations. None of these created ops can be deleted or moved. This safeguards against unintentional changes that could result in machine crashes.



4. Save your file.

TASK 6: SUB SPINDLE TURNING

- In this task you will face the part and turn the OD profile in the sub (right) spindle.
- Facing the part. Before starting the sub spindle turning make sure the red arrow is in the correct position. It should be directly below Toolpath Group-2. Then select Turning > Face...



 When the Lathe Face Properties window opens, select Axis Combination/Spindle Origin by clicking the small graphic. This will open the Axis Combination/Spindle Origin window.
 Select Lower Right as we will be using the lower turret to turn on the sub (right) spindle. Then hit the green checkmark to accept.



Mill-Turn Lesson-2 - 37

3. On the **Lathe Face Properties** window choose a tool that works on the right spindle. You must select a different tool than used on the lower turret left spindle as they will be facing opposite directions. The same lower turret tool cannot face on the left and right spindles. **Note:** Based on your actual machine, it may be necessary to change tool numbers.

2	1		^	Tool number: 4		Offsetnu	mber: 4	
T0101 R0.0313 ROUGH FACE LEFT -	T0101 R0.031 ROUGH FACE L	3 EF					Tool Angle	
80 DEG.	1			Feed rate:	0.01	in/rev) in/min	O micro-in
				Finish feed rate:	0.005	in/rev) in/min	🔿 micro-in
T0202 D0 0213	T0202 P0 031	2		Spindle speed:	200	● css	ORPM	
OD ROUGH LEFT	OD ROUGH LEF	т		Finish spindle speed:	1000	Ocss	RPM	
	9			Max. spindle speed:	7000			
T0303 R0.0156 OD FINISH LEFT	T0303 R0.015 OD FINISH LEFT	ю Г		Force tool change				
	4		~	Comment				
Show library tools	Rig	ht-click for options						^
Select library tool.		Tool <u>F</u> ilter						*
Lower Right Spindle origin:	dle Origin Machine Group-1.T	urning.RightLow	1		Sto	ck Update	To batch	f point

4. Make the following changes to the **Face Parameters** and enable the **Corner** option. Check that yours are the same.

👬 Lathe Face		×
Select Points. Use stock Finish Z. 00	Entry amount Rough stepover: 005 Finish stepover: 001 Overcut amount 00 Retract amount 0.1 Rapid retract	Tool Compensation Compensation type: Compensation direction: Left Cutter around corners: All Cutter around corners:
	Stock to leave:	Corner
-26	Cut away from center line	Filter
	Cross centerline cut	
		Chip Break
		Second Feed/Speed
	Generate toolpath	0 0 0

5. After enabling Corner, select it. Make the following changes on the Face Corner page. Hit OK to create the operation.



6. You can do a quick **backplot** again to check your toolpath. Select just the sub spindle facing op and run.



7. **Rough Milling**. We will rough this feature using a 2D High Speed Dynamic Milling operation. We will need some extra geometry for chaining in this operation. Create **Curves All Edges** just like we did on the main spindle side.



8. Select Dynamic Mill from the Milling 2D menu.

File	Home	Wireframe	Surfaces	Solids	Model Prep	Mesh	Drafting	Transform	Art	Machine	Vie	>	Milling
Contour	Drill	Dynamic M		4 	OptiRough	Area Roug	Waterline	Raster	*	Curve	Swarf Milling	Unified	Deburr
		2D					3D				Mu	Itiaxis	

9. Chain the geometry. There are multiple ways to do this, but for this exercise we will use the following options shown with black arrows. Be sure to check the From Outside radio button. When selecting the regions, we will use the Face Selection option under the Solids mode.

×



Mill-Turn Lesson-2 - 40

10. Make the following selections for **Machining Regions**, **Avoidance Regions**, and **Air Regions**. Green check to accept each set of selections and again when all selected.



Mill-Turn Lesson-2 - 41

11. Select a **1/4**" **Flat Endmill** from the library and **change the cutting length** to **0.6**". From the **Setup** menu select **Upper Right** as your axis combination and **C-Axis Face** as your **Mill Type**.

Toolpath Type Tool Holder Setup Stock Cut Parameters Corner Pretreatment	Mill type C-Axis Face Axis Substitution	Upper Left Upper Right
O Depth Cuts O Break Through Unking Parameters Entry Motion Ref. Points Arc Filter / Tolerance	Y-Axis Face Y-Axis Cross Plane Rotation	Lower Left Lower Right

12. Set your Cut Parameters as shown below.



13. Set you **Linking Parameters** and **Arc Filter/Tolerance** parameters as shown below, then green check to accept and create the operation.



Mill-Turn Lesson-2 - 43

14. **Backplot** the operation.

15. Save your file. 国

16. **Finish Mill** the pocket. Select **Milling > Contour**, then chain one of the islands. We will finish the other three islands using the **Toolpath Transform** later.

	File	Home	Wireframe	Surfaces	Solids	Model Prep	Mesh	Drafting	Transform	Art	Machin	ie Vie		Milling
C	Contour		Dynamic 1	Mill Face	* *	OptiRough	Area Roug	Waterline	Raster	*	Curve	Swarf Milling	Unified	Deburr
			2D					3D				Mu	tiaxis	
	Wirefram Mode © Cp Selectio In Selectio Branche Start/En	ne Chaining	× 30 • • *											

Mill-Turn Lesson-2 - 44

17. Use the same **1/4"** Flat Endmill for the finishing operation. From the Setup menu select **Upper Right** as your axis combination and C-Axis Face as your Mill Type.



18. Set the Lead In/Out to create a straight tangent motion.



19. Set Linking Parameters to match below, then green check to accept.



Mill-Turn Lesson-2 - 45

20. Transform the finish operation to machine the three remaining islands. Select Milling > Toolpath Transform, set Mill Type to C-Axis Face, Axis Combination to Upper Right, Transform Type to Planar Rotate, and Source Operations to Operation 18 (the finish operation we just made).



21. On the **Planar Rotate** tab set the number of instances to 3 and set the angle to 90.0. Green check to accept.

Transform Operation Parameters		×
Type and Methods Planar Rotate		
Instances # 90.0 V	Rotation view Machine Group-1.Face Mill.Right 1	



22. Backplot the Finish operation with the Transform operation.

23. Drill toolpath to countersink backside of thru hole.

File	Home	Wireframe	Surfaces	Solids	Model Prep	Mesh	Drafting	Transform	Art	Machine	View	Turning	Milling
Contour	Drill	Camic M	fill Face	* * ₽	OptiRough	Area Roug	Waterline	Raster	*	Curve	Swarf Milling	Unified	Deburr
	_	2D	•			-	3D				Mu	Itiaxis	

24. Select the outer diameter of the chamfered hole, similar to what was already done on the **Left Spindle**. Be sure to select from the part located in the **Right Spindle**.



25. Select the same tool that was used on the Left Spindle.

2D Toolpaths - Drill/Circles G83/G87 Drilling Cycle											×
🧯 💾 📆 📾											
Toolpath Type		Statu	is Tool Nu	Assembly	Tool Name	Holder Na	Diame	Corner Ra	Length	Flutes	_
Holder		1	2		0.625 5/8 X 90 Spot Drill		0.625	0.0	1.0	2	
Setup		1	3		1/2 BULL ENDMILL 0.0313 R		0.5	0.03125	1.0	4	
Cut Parameters		-	4		1/4 FLAT ENDMILL		0.25	0.0	0.6	4	

26. From the **Setup** menu select the **Upper Right** axis configuration and the **Y-Axis Face** option for **Mill Type**.



27. Set the Linking Parameters as seen below. Green check to accept.

Holder	.,,,,			Calculate incrementz Automatic linking	al values from h	ioles/lines		
Setup				Arc fit maximum rad	ius	0.5		
Cut Parameters			Output feed move			500.0		
Hole Seg	ments arameters			Clearance	2.0	Absolute Incremental		100
Ref. P Safety	omp Points y Zone	T		Use clearance of	nly at the start	Associative and end of operation	-	0
Arc Filter	/ Tolerance	8		Retract	0.1	Absolute Incremental Associative	990 910	0
		H						
ck View Settir	ngs			Top of shade	-0.5	Absolute		
ick View Settir ool ool Diameter orner Radius	ngs 0.625 5/8 X 9. ^ 0.625 0] 	Top of stock	-0.5	Absolute Incremental Associative	Star Star	0

TASK 7: MACHINE SYNCING

- In this task you will work in Mastercam Code Expert to set the machine syncs.
- Select the Main Viewsheet-1 to display both spindles. In the Operations Manager left click Machine Group -1 to select all operations. Select the Post Selected Operations button. This will generate the IOF file and open a Sync Manager window in Mastercam Code Expert.

Toolpaths
▶ × ID IX II · X II · 8 · 8 · 8 · 8 · 9 @
Image: Second

2. Code Expert. Next, we will create all the sync points. Create Sync 1 between the Retract of the 1 Lathe Face and the Approach of the 2 Lathe Rough.



Mill-Turn Lesson-2 - 49

3. Next, create Sync 2 between the 2 Lathe Rough Retract and the 3 Lathe Finish Approach.



4. Sync 3 will be between the 4 Lathe Drill Retract and the 5 G83/G87 Drilling Cycle Approach.



5. Sync 4 will go between the 6 Contour (2D) End and the 7 Turret Park End.



6. Sync 5 will be between the 14 Spindle Move End and the 15 Turret Park Lower Reference Return.



 Sync 6 will be the final sync located between the 16 Lathe Face performed by the lower turret and the 17 2D High Speed (2D Dynamic Mill) which will be machine by the upper turret.



8. Save you work. Saving from the Code Expert Sync Manager also writes back to the Mastercam file. The colored light at the top of the Operations Manager will indicate whether the file has been saved or not.

millturn lesson 2_2023-1.iof* 🥥 × Start Page	Not saved.
millturn lesson 2_2023-1.iof 🤤 🗙 Start Page	Saved.

TASK 8: MACHINE SIMULATION:

I In this task you will run a complete machine simulation to check for potential problems.

1. In Code Expert, from the **Sync Manager** tab click on the **Launch Icon** to launch machine simulation.

File	Home View	Sync Manag	jer			
Post Stop	Launch	Renumber	Reset All Values	 ✓ Approach ✓ Retract ✓ Link 	 ✓ Stock ✓ Fixtures ✓ Component 	Selected Operations Only Expanded Operations Only
Post	Simulation 🖬	Synchronization	Tokens		Displa	ау Гы

2. **Play** the machine simulation.



3. Adjust the simulation speed as needed. When finished, **Exit** and return to Code Expert to post the NC code.

		11			
Performance Precision	14 14 AA 🕨 M	A 144 14	Ø D	Slow — Fast	\times

TASK 9: POSTING THE CODE:

- ➔ In this task you use code expert to post the NC program.
- 1. Before we post the code, we have a few extra settings to consider. Each operation has an independent **Coolant Strategy** selection as well as Y-axis offsets and Reference Positions if desired to set. Access from the drop-down arrow for each operation.

perations	•
Machi	ne Group-1 🔍
Jpper	Lower
🖂 🔁 2 Lathe Rough 🗹 📜 🙀 🗭 📄 📨	🔺 🔁 📙 1 Lathe Face 🔻 📜 🏠 🗭 📟
Coolant	IEG.
Coolant strategy for an operation	Off 👻 🗸
Operation	
Y-axis offset for lathe operations	0
Reference Position	
Reference position that defines where this opera	tion begins. Upper X Only Return 🔹 🁌 📄 🛛 👄
	G.

2. You can also enter more information under the drop-down for Machine Group-1.

UU.	Machine Group-1 V	
Upper	Feed Maximum feedrate for polar interpolation (G112)	100
T1: OD ROUGH LEFT - 80 DEG. Sync 1 ♥ Approach (X1.1580, Z0.2100) ♥ Motion	Header Job number Programmer name	
Sync 2 ♥ → We Retract ♥ → Hend	Tailstock Stop program to load tailstock	2

3. Save your file.

4. **Post** the code by pressing the **G1 button** located under the **Sync Manager** tab. Code Expert will open a new tab with the NC code.



5. By default, the code is displayed in single stream. Under the **NC Functions** tab, to switch to **Multi-Stream** mode. You may be prompted to select a file name for multi-stream view.

File	Home View NC Fun	tions			
Go To	Insert Block Numbers inser Remove Block Numbers Rem Remove Spaces in Rem Editing	t Block Skip ove Block Skip ove Comments Communications 🕫 Syncs	Last Mark First Previo	bus Next Last Single Multi-Street Display	am Align Syncs NC Configuration Utilities
milltur	m lesson 2.NC × millturn lesson	2_2023-1.iof 🥥 Start Page			
	1 8				
	2 00000 (UPPER TURRET)			
	3 (millturn lesson 2_	🔤 Create Multi-Stream View	×		
	4 (GENERIC FANUC MILL		~		
	5 (DATE: MONDAY, 30	NC Configuration:			
	6 (MCAM FILE - C:\Us		/	2\Mastercam Files\milltur	n lesson 2_2023.mcs
	7 (NC FILE - C:\User	Generic Fanuc Mill-Turn LTZ (1 NC stream file)	-	lesson 2.NC)	
	8 (JOB NUMBER -)	Files to view:			
	10 (TO01001 OD POUGH	militum losson 2 NC	Proute		
	11 (T002002 0.625 5/	mindrin resson zinc	Diowse		
	12 (T003003 1/2 BUL		Select		
	13 (T004004 1/4 FLA		1		
	14 G28 UO. VO.				
	15 G28 W0.				
	16				
	17 N100	Help	OK Cancel		
1	18 (OPERATION # 2)				

Mill-Turn Lesson-2 - 55

6. You will see all the sync points you created. Hovering over one sync point will highlight it in both streams if they are both visible on-screen.

File		Home View NC Fu	nctions										
Go To	🖁 ir 🛃 R 📑 R	nsert Block Numbers 🕌 Ins Remove Block Numbers 🌺 Re Lemove Spaces 💁 Re Editing	ert Block Skip Send File move Block Skip Send move Comments Receive Communications 5	First Previous	Next Last	Mark	First	Previous Tools	Next Last	Single Stream	Multi-Stream Display	Align Syncs	NC Configuration Utilities
milltu	rn les	sson 2.NC × millturn less	on 2_2023-1.iof 🔵 Start Page										
	1 2	% 00000 (UPPER TURRE	т)										
	3	(millturn lesson 2_ (GENERIC FANUC MILL	🚰 Create Multi-Stream View					×					
	5	(DATE: MONDAY, 30 (MCAM FILE - C:\Us	NC Configuration:					1	2\Mastercan	Files	millturn	lesson	2_2023.mcs
	7	(NC FILE - C:\User	Generic Fanuc Mill-Turn LTZ (1	NC stream file)	/	•		1	lesson 2.NC)			
	9	(PROGRAMMER -)	Files to view:		×								
	10	(T001001 OD ROUGH	millturn lesson 2.NC				Brow	/se					
	11	(T002002 0.625 5/ (T003003 1/2 BUL					Sele	ct					
	13	(T004004 1/4 FLA					1						
	14	G28 UO. VO.											
	15	G28 W0.											
	17	N100	Help			OK	Can	cel					
	18	(OPERATION # 2)					-						

7. Your code is now ready to run. If you have communications set up, hit send to send the NC file to your machine.



8. This concludes Mill-Turn Lesson 2.