

# MASTERCAM

Mechanical Workshop Practice 4

**MECHANICAL ENGINEERING DEPARTMENT** 

# BASIC MILL

# MASTERCAM TUTORIAL 1



## ALL RIGHTS RESERVED

No part of the publication of this book may be republished, stored for production, or converted into any form or by any means, whether by electronic means, photographs, recordings and so on without the prior written permission of Sultan Salahuddin Abdul Aziz Shah.

## MECHANICAL WORKSHOP PRACTICE 4 – MASTERCAM BASIC MILL TUTORIAL I

EDITOR-IN-CHIEF: Mohd Nasir Bin Kamaruddin

EDITORS BY TOPIC Dr. Mohd Shahrom Bin Ismail Nazratulhuda Binti Awang @ Hashim

AUTHORS: Mohd Nasir Bin Kamaruddin Ts. Salhana Binti Sahidin@Salehudin Nazratulhuda Binti Awang @ Hashim Dr. Mohd Shahrom Bin Ismail

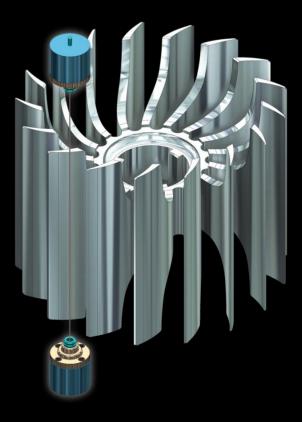
NO e-ISBN: 978-967-0032-24-5

First Edition 2022

Published by:

## **UNIT PENERBITAN**

Politeknik Sultan Salahuddin Abdul Aziz Shah Persiaran Usahawan, Seksyen UI, 40150 Shah Alam Selangor Telephone No.: +603 5163 4000 Fax No.: +603 5569 1903



# BIODATA



#### MOHD NASIR BIN KAMARUDDIN

Serving as a Programme Leader. Acting as a Curriculum Coordinator of PSA Certified Center, Department of Polytechnic and Community College Education Branch (JPPKK). Acquired special approval as a Certified Curriculum Facilitator from the Skills Development Department (JPK). Experienced in mapping polytechnic and community college syllabus based on NOSS Ability Structure.



#### NAZRATULHUDA BINTI AWANG @ HASHIM

As early as 2008, he served as a lecturer in the field of engineering, particularly packaging, at several polytechnics and is currently the Head of the Mechanical Engineering Department, Polytechnic of Sultan Salahuddin Abdul Aziz Shah. She is actively involved in journal and book editorial for PSA.



#### TS. SALHANA BINTI SAHIDIN@SAELHUDIN

Serves as a Head of Strategic Management Center, Sultan Salahuddin Abdul Aziz Shah Polytechnic (PSA). SKM MECHANICAL DRAUGHTMANSHIP holder. Experienced in managing programme implementation under Department of Skills Development from 2020 until present.



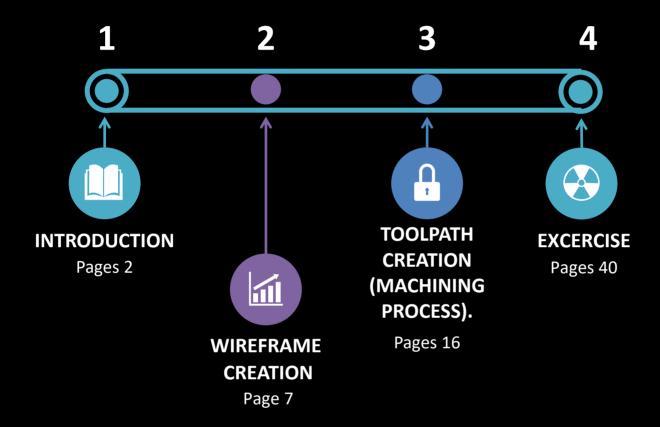
#### **DR. MOHD SHAHROM BIN ISMAIL**

He obtained his MEng. (Distinction) in Mechanical Engineering in 2010 from University of Strathclyde and PhD in mechanical engineering from Universiti Teknikal Malaysia Melaka. He is currently a senior lecturer in Politeknik Sultan Salahuddin Abdul Aziz Shah and is responsible for various national initiatives in higher education that focuses on the area of research, publication, strategic collaboration and human capital development.



Alhamdulillah, all praise and thanks to Allah <sup>®</sup>for His mercy and grace to all of us as the people of Prophet Muhammad .<sup>®</sup> Mechanical Workshop Practice 4 – Mastercam Basic Mill Tutorial I can be produced according to the set plan. This book is published to guide student in DJJ40142 Mechcanical Workshop Practice 4 (CAM) course. A wreath of acknowledgment for the Mechanical Engineering Department management and the PSA Executive Committee who are the backbone of the success of producing this tutorial book. It is hoped that the publication of this book can be a source of inspiration and reference for the Polytechnic those implement this Mastercam software in this course..

# CONTENTS





#### CONTENTS

TOPIC	PAGES
APPRECIATION	
ABSTRACT	
INTRODUCTION. 1.1 Starting MasterCAM Software 1.2 GUI – Graphical User Interface 1.3 Manager Panels	2
WIREFRAME CREATION. 2.1 Setting Up GUI 2.2 Rectangle	7
2.3 Circle	
2.4 Chamfer TOOLPATH CREATION (MACHINING PROCESS).	16
3.1 Setup Sheet	
3.2 Machining & Stock	
3.3 Face Mill	
3.4 Circle Mill	
3.5 Spot Drill	
3.6 Chamfer	
3.7 Post / G-CODE	

## EXCERCISE.

40

4.0 Excercise 1

#### INTRODUCTION

This chapter has been developed by Nazratulhuda Binti Awang @ Hashim

#### 1.1 Starting MasterCam

- 1.1.1 For Windows 7
  - Select the Start button
  - Select All programs and click on Mastercam 2022

#### 1.1.2 For Windows 8

- Select the Start button
- Click on the drop down arrow to open Apps
- Find and click on Mastercam 2022

#### 1.1.3 For Windows 10

- Select the Start button
- Click on the drop down arrow to open Apps
- Find and click on Mastercam 2022
- To start the software from desktop, click on the shortcut icon as shown



C C - 1 C - C	Neth Dutting Transform Art Mathine View	Mastercan Design 2022	- & X Standard - My Matteram 😇 🚱 A
*·	Z 0.0 - X Non-Associative B & Acc Center Points	Constant of the second s	
Toolpaths ▶ ★ ★   1 ⊨ 1x 당·   종 등 왕 여 1 ⊨ 2   9 음 응 ☆   ▼ ★ 다 수   % 연 년 · 코 양			
			•
			ŏ
$\bigcirc$			
8		€ Free NCG G Regin (NCG) C Itemetic (NCG) C View Ø Guide C guine	Ů
		Detet Entities     Analyze Distance     Analyze Entity Reporter	e T
			2 4 6 6
	•		
	тор		10 Hotek
Toolpaths Solids Planes Levels Recent Functions	Viewsheet 1 +	SECTION VIEW: OFF SELECTED ENTITIES: 0 X: -18.487167 11: 26.79220 Z: 0.0000	0 3D CPLANE: TPLANE: WCS: 🕀 🕀 🕘 🥥 🧔

## 1.2 GUI – Graphical User Interface

1. Quick Acces Toolbar

QAT Contains a fully customizable set of functions that can be quickly accesed by the user.

#### 2. Backstage (FILE)

Allows you to manage files. You can insert information about files, start new file, open an existing one or merge files together. You can also save, convert or print files as well as access the help resources.

3. Tabs

Contain all the functionality within Mastercam.

#### 4. Ribbon

Display the commands available for a selected tab

5. Selection Bar

Allows you to set the AutoCursor modes and to switch between wireframe or solid selections.

6. Quick Mass Buttons

Lets you select all entities of a specific type. Clicking on the left side of the button or right side of the button toggles between select all or only.

#### 7. Right Click Menu

Right click menu allows quick access to function such as zoom, graphic views or recent functions used.

8. Toolpath/Solid/Planes Manager

List the history of the toolpath operation and solids

#### 9. Graphics Window

Workspace area in Mastercam where the geometry is displayed

#### 10. Scale

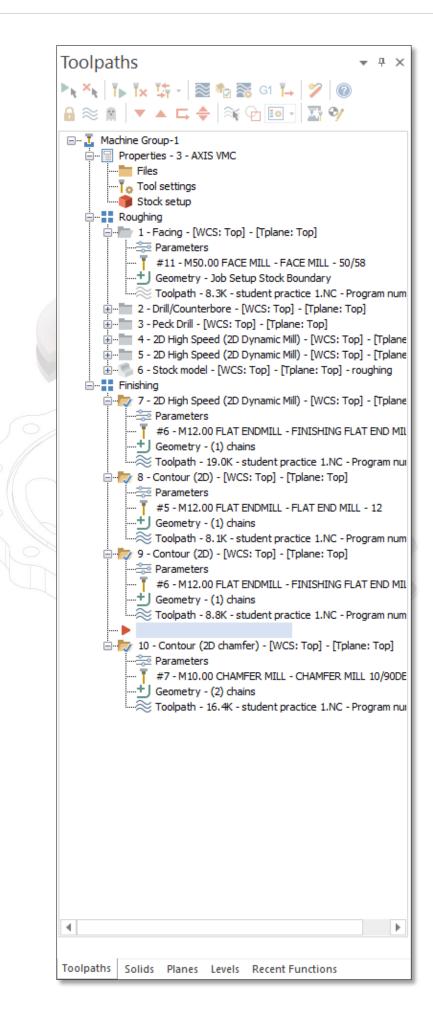
Show you a scale of the object on the screen.

#### 11. WCS: TOP T/Cplane

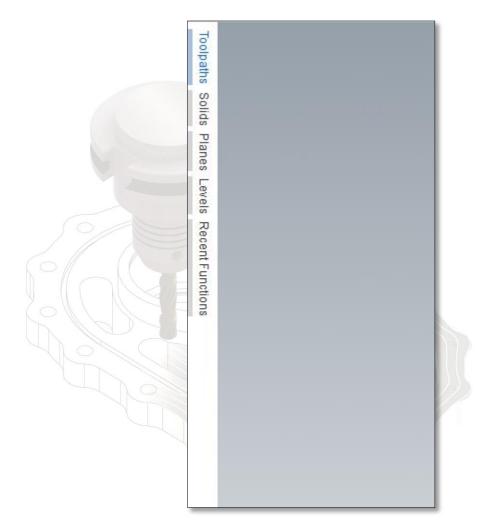
Display the current WCS and T/Cplane information

#### **1.3 Manager Panels**

The Toolpath Manager display all the operations for current part. You can manage any operation as shown in figure.



The Toolpath Manager, Solid Manager or Planes Manager can be hidden to gain more space in the graphics area for creating geometry. Use auto hide icon to close all Toolpaths, solid, Planes and Levels Manager panels. The panels will be hidden to the left of the graphics window as shown or at the bottom of the manager as shown previously.



To un-hide them, click on one of the managers to open it and then click again on the auto hide icon a shown. Selecting the close icon, will close the manager panel.

#### WIREFRAME CREATION

This chapter has been developed by Ts Salhana Binti Sahidin@Salehudin

#### 2.1 Setting UP GUI

First step, you must set the scale for your drawing. This is most important step before starting your drawing. Go to File and configurations.

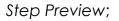


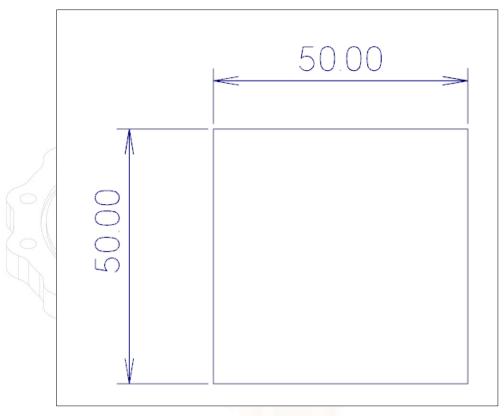
#### Then, select the prefer units, either inch or metric.

System Configuration		×
Analyze CAD Chaining Colors Communications Converters Default Machines Dimensions and Notes Files On-Screen Controls Post Dialog Defaults Prining Reports Screen Selection Shading Simulation Solids Spin Controls Stat / Exit Tolerances Displath Manager Toolpath Manager	Number of places after decimal for analyze       N.123 ~         Analyze Measurement Options	
Current:	c:\users\nasir\onedrive\document\mcamxm.config <metric> <startup></startup></metric>	2

## 2.2 Rectangle

In this step, you will learn how to create rectangle given the width, the height and the anchor position. You will create the 50 mm by 50 mm rectangle with the center anchor in the orgin.





Create a 50 mm by 50 mm Rectangle

	<b>*</b> • 🔒	1	(24 ∓						
File	Home	Wireframe	Surfaces	Solids	Model Prep	Mesh	Drafting	Transform	Art
<b>₽</b> oint Position •	+ Bolt Circle	+ Line	ine Parallel ine Perpendicula ine Closest +	(	ircle	Points angent e Edge Point	Spline Manual *	Rectangle	A Greate Bou etters E
Point	ts	Li	nes		Arcs		Splines		

In Rectangle panel, enter the Width and Height and enable Anchor to center as shown.

Rectangle	<b>ч</b> ×
(?)	o 😒 💿
Basic	
Points	۲
1 2	
Dimensions	۲
Width: 50.0	- ‡ 🔒
Height: 50.0	- 🗘 🔒
Settings	۲
Anchor to center	

Note: Don't select that create surface. Anchor to center sets the base point on the rectangle to its center and draws the rectangle outward from the center

inter a width and height, or select the position of the corner.	🔓 🏨 AutoCursor 🗸 📩 🍾 🧐 🦷	9 9 🔮 🕸 + 🖩 + 🥵 🐿 🔗 🦄	
г <i>–</i> –			
			i
I			I.
			1
			· ·
1			1
i I			i
			1
			i
I			1
			1
			i
1			1
			1

Select the position of the base point shown. A preview of the wireframe should look as shown. The wireframe should appear in cyan blue color which is the color for the live entities. While rectangle is live, you can adjust the dimensions or select a new base point. Select the OK button to exit the Rectangle command.



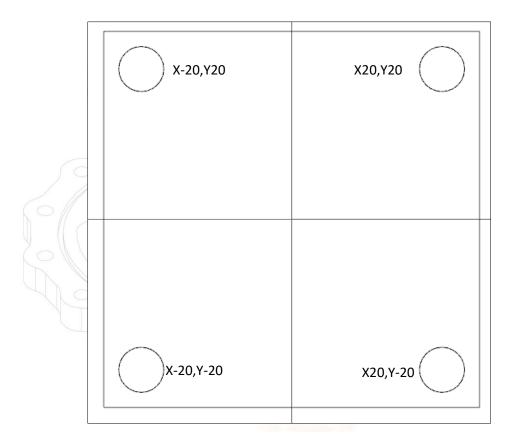
While creating wireframe, you can undo the last step using the Undo icon ?. You can undo as many step as need. You also can redo by using the Redo icon ?. To delete unwanted geometry, selec the geometry first and then press Delete from the keyboard or right click on mouse then click delete entities.

		1
	* · · · · · · · · ·	
	🚳 - 🇮 🍡 👫 3D Z 0.0 - 💐 1	
	Zoom Window	
	🔎 Unzoom 80%	
	Ø Dynamic Rotation	
	Fit	
	Top (WCS)	
	Front (WCS)	
	🗊 Right (WCS)	
	🗊 Isometric (WCS)	
	GView	
	👰 Quick Cplane	
(	× Delete Entities	
	Analyze Distance	
	Analyze Entity Properties	
	-	

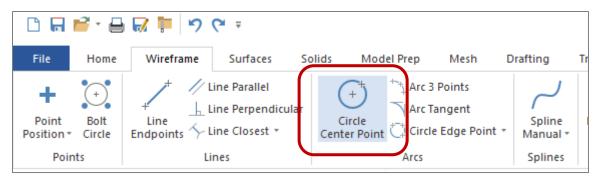
To zoom, move the cursor in the center of the geometry and scroll the mouse wheel.

#### 2.3 Circle

In this step, you will make circles for which you know the diameter and the coordinates. To use Circle, you need to know the measurement of the center point and the radius or the diameter of the circle. To complete this step, you must know the Coordinate System that specifies each point by a pair of numerical coordinates.



From the Arcs group, select Circle Center Point.



Enter a Radius of 3.0 mm the panel as shown. You can lock the value by click on the locker icon. So, it will lock for making another three circles. Select the AutoCursor icon from the selection toolbar and the field where you can type the coordinates will open at the upper left side of the graphics windows as shown

Circle Center Point		
(2)		
Basic		
Entity	$\overline{\mathbf{O}}$	
Method:      Manual     O Tangent		
Center Point	$\odot$	
Reselect		
Size	۲	
Radius: 3.0	<b>- ↓ </b>	
Diameter: 6.0	<b>→ \$ @</b>	
Settings	$\overline{\mathbf{O}}$	
Create surface		

Type 20,20 as shown. When you fill the coordinates, the first value is represent x value/axis and the second value represent y value/axis. Key in the coordinate another three postion with this value; 20,-20, -20,20 and -20,-20. Once complete, choose the OK button to exit the command.

Another method, you also can use mirror function. Go to transform tab. Click on mirror icon.

File	Home	Wire	frame	Surfa	ices	Solids	Mod	el Prep	Mesh	Drafting	Transf	orm
Dynamic	Translate	Translate to Plane	Notate	<b>↓</b> Project	Move to Origin	Ц Mirror	C↔  Foll	<b> →</b> Entity	Chains	Geometry	lar Array	Stre
			Positi	on				O	ffset	Layou	t	

Then select the entities to mirror on X axis.

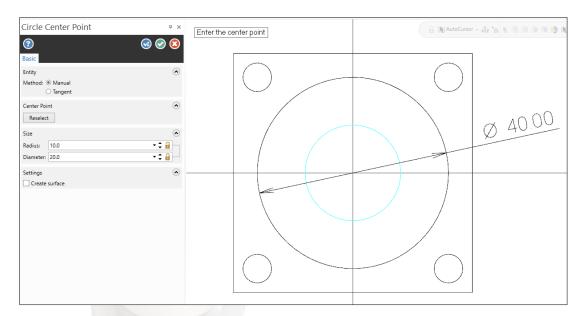
Mirror 4 ×	
? Solution	
Basic Advanced	
Entity	
Method:  © Copy O Move Join	•
Selection  Reselect	Ŭ
Axis	
X axis:	
Y offset: 0.0 ▼ ♦	
OYaxis: X offset: 0.0 ▼	
O X and Y axes:	
O Angle: 0.0 → 🗘 🕟	
O Vector:	
Annotation 📀	$\bigcirc$
Circle Start Position	

Do the same step for another two circles on Y axis. You also can use rotate function if the angle each circle are same. To do this, go to transform tab and click the rotate icon. Select entities to rotate.

Rotate + ×	
⑦	
Basic Advanced	
Entity	 
Method:   Copy	
O Move O Join	(+)
Selection	
Reselect	
Rotation Center Point	
Reselect	
Instances	
Number: 4	
Angle: 360.0 -	
Distance: O Angle between (1) Total sweep	
Method:   Rotate	
○ Translate	$\frown$
Remove	
Reset	
Circle Start Position	

Insert the data number is 4 (circles), angle 360° and distance total sweep.

Next, create another circle center point for diameter 40mm and 20mm at origin point.



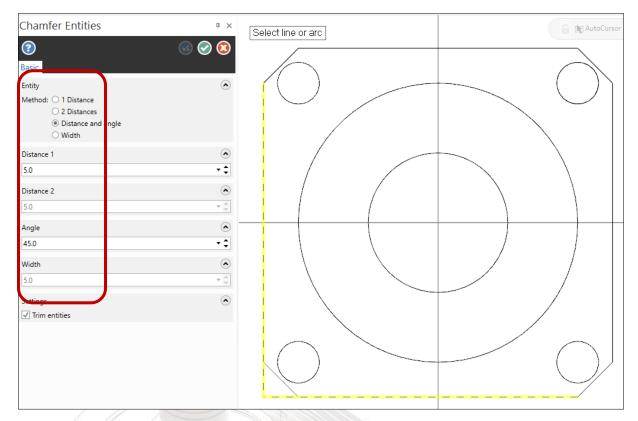
While the circle is live, cyan color, the circle diameter can be modified. When you enter to finish the circle, you cannot edit the circle. Either you undo or delete and make a new circle.

#### 2.4 Chamfer

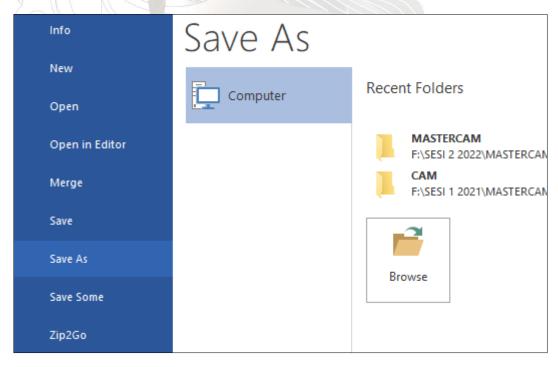
In this section, you will create 45° chamfers at each corners of the rectangle. First, go to wireframe tab and then click the chamfer icon.



Choose the distance and angle method. Put the distance 5.0 mm and the angle is 45°.



Finish the step by save the file at file tab and click save as and browse the location to save it.



#### **TOOLPATH CREATION (MACHINING PROCESS)**

This chapter has been developed by Mohd Nasir Bin Kamaruddin

#### 3.1 Setup Sheet

This is important step you must plan before running the toolpath procdure. When you get the design/drawing from customer, you must find out and plan;

- ✓ Type of Machine (depend on your machine brand)
- ✓ Material Selection
- ✓ Type of Machining/Toolpath
- ✓ Type of Tool and diameter
- ✓ Facing, Milling and Finishing procedure
- ✓ Post Processor for Machining

For this basic tutorial we will use

- ✓ Default CNC Machine
- ✓ Aluminum Material
- ✓ Face Mill, Circle Mill, Drill, Contour and finishing Toolpath
- ✓ Face Mill Flat End Mill and Dril Cutting Tools

#### 3.2 Machining & Stock

This step, you will learn how to setup the maching and stock / material. For this education purpose we will use default machine. Go to machine tab and click Mill by default as shown.

File Home Wireframe Surfaces	Solids Mod	el Prep Mesh	Drafting Trans	form Art	Machine View
Image: Second	Control Mach Definition Defini			Simulate Gene	1 Create
Default chine Type	ol	b Setup	Simulator	Fa Po	st Setup S
Manage List	<b>▼</b> ₽ ×				
Machine Group-1     Properties - Mill Default MM     Toolpath Group-1     M					

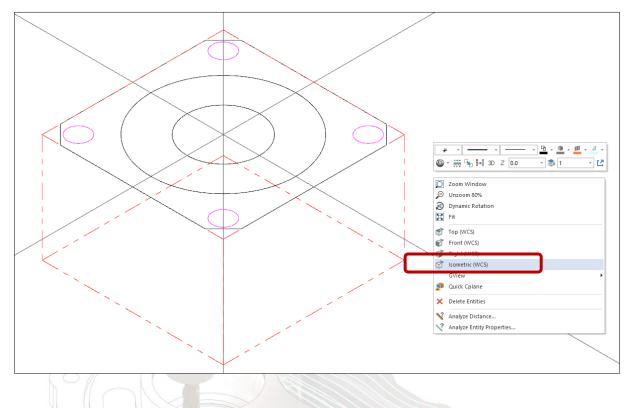
Next, expand the properties and click tool setting.

Machin	e Group Properties		
Files	Tool Settings Stock S	etup	
	Default program number Feed Calculation From tool From material	0	Toolpath Configuration Assign tool numbers sequentially Wam of duplicate tool numbers
	<ul> <li>From defaults</li> <li>User defined</li> </ul>	5000.0	Use tool's step, peck, coolant Search tool library when entering a tool number
	Spindle speed Feed rate <b>Retract rate</b>	5000.0 100.0 150.0	Advanced options
	Plunge rate	25.0	Clearance height Retract height Feed plane
	Minimum arc feed	125.0	Sequence number

You can set program number. This function is to make sure the machine recall the tool setting memory by reading the program number. Basically, you can active the toolpath configuration by pick the assing tool number and warn of duplicate tool numbers. Now, we go to stock setup tab. You will see the shape option and the picture of your stock location. Usually, for basic machining you will learn rectangular and cylindrical shape of stock. Next, click the All Entitites button and the software will automatically measure the length of X and Y axis. You must put the height of Z axis. Now, key in the height of Z 30 mm.

Machine Group Propertie	es	$\times$
Files Tool Settings S	tock Setup	
Shape Shape Rectangular Cylindrical Solid/Mesh File	Iane Top Axis O X O Y O Z C	
- File		
<ul> <li>✓ Display</li> <li>✓ Fit screen</li> <li>● Wireframe</li> <li>○ Shaded</li> <li>Stock Origin         <ul> <li>In view</li> <li>coordinates</li> <li>X</li> <li>0.0</li> <li>Y</li> <li>0.0</li> </ul> </li> </ul>	Y X 50.0 50.0 Z 30 0	
Z 0.0	Select comers       Bounding box       NCI extents         All Surfaces       All Solids       All Entities       Unselect All	I

Right click your mouse and click the isometric view to view the stock that you have set. You will see the stock as a shown.



#### 3.3 Face Mill

This section you will start the first machinig process setup (toolpath). Before we machining the stock to produce the shape of customer need. We will start with Face Mill first. It is to make sure the stoct in good figure and measurement. This processs will clearn the dirt/oil in your surface stock.

At the toolpath tab, go to 2D option and choose Face Mill.

File	Home	Wireframe	Surfaces	Solids
	(mm	ē		<b>,</b>
Contour	Dril	I Dynam	ic Face	
		2D		
Toolpath	าร			-
► <sub>k</sub> × <sub>k</sub>   ī	Tx ⊈‡	- 🛛 🗞 🕯	😸 G1 🏹 🛛 🌮	2
$\mathbf{R} \approx \mathbf{R}$	<b>•</b>	<b>L 🔶</b> 🔍	G 💿 - 🗵	3 0/

You will see wireframe chaining popup. For facing process you don't need to click the chain at any entities of your stock. Sotware will automatically detect the top of your shape stock. It will read as a face mill location.

Wireframe Chaining	×
Mode Cplane 💿 3D	
Selection Method	
Selection	
Branches	
Start/End	
	]

Then clik the Ok button. You will see the 2D Toolpath setting box.

2D Toolpaths - Facing	· · · · · ·	>
🕴 🔚 🖬 🖻 🥗		
Toolpath Type Tool Holder Cut Parameters Cut Parameters Linking Parameters Home / Ref. Points Arc Filter / Tolerance Planes Coolant Canned Text Misc Values	Contour Pocket Facing Slot mill Model Chamfer	Chain geometry (0)  Solid model Side clearance 0.0
Axis Control Axis Combination Rotary Axis Control		Avoidance model
Quick View Settings Tool 20. FLAT E Tool Diameter 20 Corner Radius 0 Feed Rate 50 Spindle Speed 3500 Coolant Off Tool Length 0 Length 0ffset 1 Diameter Off 1 Cplane / Tpl Top Axis Combin Default (1)		
<ul> <li>✓ = edited</li> <li>⊘ = disabled</li> </ul>	Generate toolpath	<ul><li>✓ × ⊕ ?</li></ul>

Click the tool list. You will see the box as shown.

Y   🛃 🖬   📭 🧉						
Tool Holder		Status	Tool Number	Assembly Na	Tool Name	I
Cut Parameters						
interim Linking Parameters						
Arc Filter / Tolerance Planes						
Coolant Canned Text Misc Values						
Axis Control						
Rotary Axis Control	<					>
				Ri	ght-click for op	tions

Go to select library tool. You will see the list of all type of tools. As you can see the figure, it show you 280 tools appear on the list. It will difficult you to choose the right tool. So, you must filter it by clicking the filter option.

Us	ers\Public\Do	ocu\Mill_mm.too	oldb 🖻				
	Tool Number	Assembly Na	Tool Name	Holder Name	Diameter	Corner Rad ^	
	1		NC SPOT DRILL - 6		6.0	0.0	
	2		NC SPOT DRILL - 8		8.0	0.0	
	3		NC SPOT DRILL - 10		10.0	0.0	
	4		NC SPOT DRILL - 12		12.0	0.0	Filter
	5		NC SPOT DRILL - 16		16.0	0.0	Filter Active
	6		NC SPOT DRILL - 20		20.0	0.0	280 of 280 tools
	7		HSS/TIN DRILL 8xD		2.0	0.0	200 01 200 (0018
	8		HSS/TIN DRILL 8xD		2.5	0.0	D: 1 1
	9		HSS/TIN DRILL 8xD		3.0	0.0	Display mode
	10		SOLID CARBIDE DRI		3.0	0.0	
	11		HSS/TIN DRILL 8xD		3.3	0.0	<ul> <li>Assemblies</li> </ul>
	12		SOLID CARBIDE DRI		3.4	0.0	Both
	13		HSS/TIN DRILL 8xD		3.5	0.0	
	14		SOLID CARBIDE DRI		3.7	0.0	
	15		SOLID CARBIDE DRI		4.0	0.0	
	16		HSS/TIN DRILL 8xD		4.0	0.0	

Click the filter and click the none button to clear all tool selection. Now, only select the face Face Mill tool only as shown. Click OK button.

		1		-		V	
Tool	l List Filter						,
T	ool Types						Tool Diameter
							Ignore V
				Fa	ce mill		Radius Type
	2200.3022			U			None Corner Full
							<ul> <li>Tool Material</li> </ul>
		Þ	P		U		HSS Ceramic
		9					🗹 Carbide 🗹 User Def 1
	U.						☑ Ti Coated ☑ User Def 2
		-	M	M	H.		All None Copy job setup matl
	0	Ø	ų		9		
			All	Non	е		
	Opera	ition maskin	g	Unit mas	king		
	No o	peration ma	isking 🖂	No unit	masking	$\sim$	
	Reset all						🖌 🗸 🖌 🎇 🖓 🖓

The update tools list will shown you the face mill tool only.

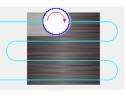
		- C:\Users\Publ	lic\Documents\Shared	Mastercam 2022	?\Mill\Tools	\Mill_mm.to
		Assembly Na	Tool Name	Holder Name	Diameter	Corner Radi
. I		Assembly Na		Holder Mallie		
21	247		FACE MILL - 42/50		42.0	0.0
21	248		FACE MILL - 50/58		50.0	0.0
23	249		FACE MILL - 55/63		55.0	0.0
23	250		FACE MILL - 63/71		63.0	0.0
22	251		FACE MILL - 72/80		72.0	0.0
21	252		FACE MILL - 80/88		80.0	0.0
8	253		FACE MILL - 92/100		92.0	0.0
23	254		FACE MILL - 100/108		100.0	0.0
23	255		FACE MILL - 117/125		117.0	0.0
22	256		FACE MILL - 125/133		125.0	0.0

Now select the Face mill 50 mm diameter. You can decide which are better diameter. It depend of your both tool availability and also the size of stock. In general, if the size of the tool diameter is big, it will speed up you maching process but it will less the quality of your stock surface.

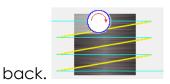
2D Toolpaths - Facing		
🎙 🔚 👪 🖻 🖷		
Toolpath Type		Tool diameter: <mark>50.0</mark>
Holder	Tool Number Tool Name Diameter Corner Radius	Corner radius: 0.0
	1 FACE MILL 50.0 0.0	
Cut Parameters		Tool name: FACE MILL - 50/58
Depth Cuts		Tool name: FACE MILL - 50/56
Linking Parameters		Tool #: <mark>1 Length</mark> c
Home / Nei. Points		
Arc Filter / Tolerance		Head #: 0 Diameter of
Planes		
Coolant		
Canned Text		

The box will appear as shown above. Clik the Cut Paremeters and choose the cutting method. There are 4 type of cutting method you can choose accordingly. The function of theese cutting method are;

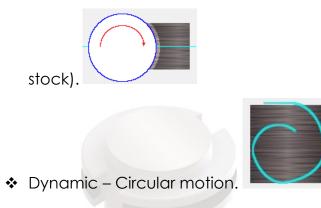
Zigzag – back and forth cutting method



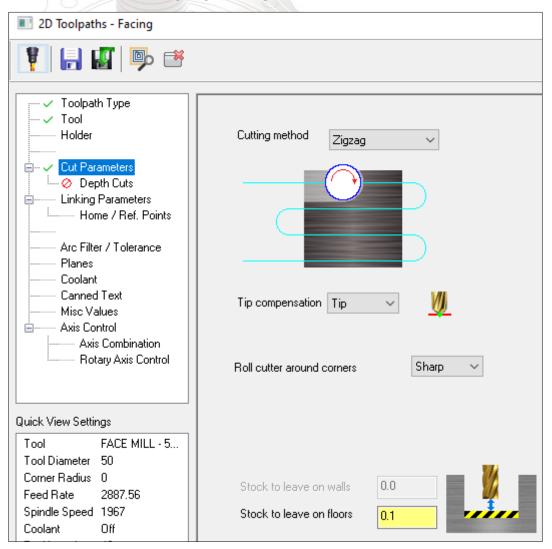
One way – one way cutting only. The cutting tool will step up when go



One pass – one step cutting only (suitable for diameter tool bigger than



In this lesson, we will use Zigzag cutting method.



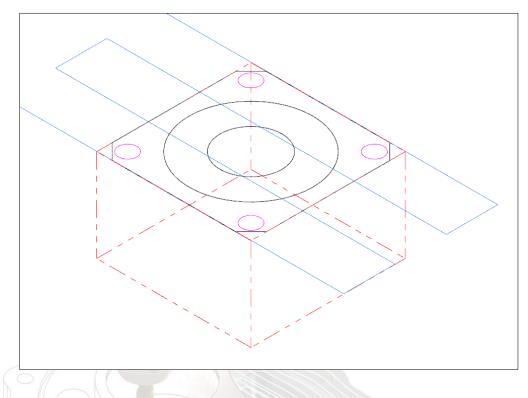
Leave stock on floor 0.1 mm for last process/finishing. Go to linking parameters and use the software setting. This step is very important to make sure a large effect on the final toolpath. Here the cutting depth and repositioning height a set. There are five major setting for this section;

- Clearance sets the height of the tool moves, if not enabled, than the Retract value will be used.
- Retract sets the height from the tool moves up to before the next tool pass.
- Feed Plane sets the height the tool rapids to before changing to the plunge rate
- ✓ Top of Stock sets the initial height of material in the Z axis
- ✓ Depth sets the final machining height in Z axis

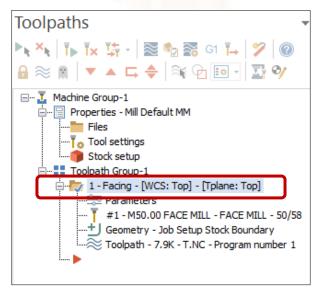
We can use either absolute or incremental coordination systems. Absolute coordination for the value stated is relative to the origin. While incremental coordination re relative to other parameters or chained geometry.

	Arc fit maximum radius	12.0 13000.0
	Clearance 50.0	<ul> <li>Absolute</li> <li>Incremental</li> <li>Associative</li> </ul>
	Use clearance only at the start and	end of operation
	Retract 25.0	<ul> <li>Absolute</li> <li>Incremental</li> <li>Associative</li> </ul>
	Feed plane 10.0	<ul> <li>Absolute</li> <li>Incremental</li> <li>Associative</li> </ul>
	Top of stock 0.0	<ul> <li>Absolute</li> <li>Incremental</li> <li>Associative</li> </ul>
	Depth	<ul> <li>Absolute</li> <li>Incremental</li> <li>Associative</li> </ul>

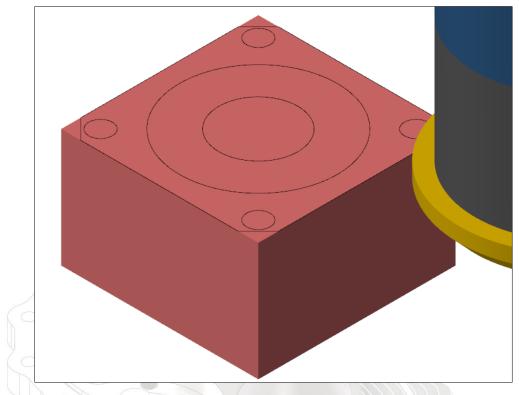
Set both top of stock and depth as 0 mm. Now click ok. The result will shown as figure



The blue trails/line mean the feed/mill process using Zigzag method. We can use backplot operation store to see details movement for cutting process. Software will animate the movement of cutter through blue trails. From this function, we can detect error in the program. Before we use the backplot operation, make sure you already sellect which operation you need to run by thick the left folder (green tick) at toolpath box.



If you want to see how all operations run, you can select icon sellect all operations . You also can simulate the operation by verify selected operation. Click the icon .

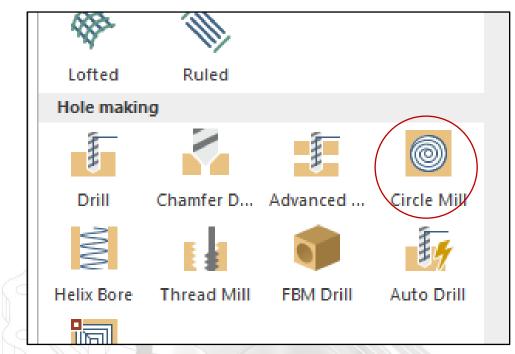


This function will simulate in three dimension (3D) movement /animation. At the left side of simulation screen will shown machining process info.

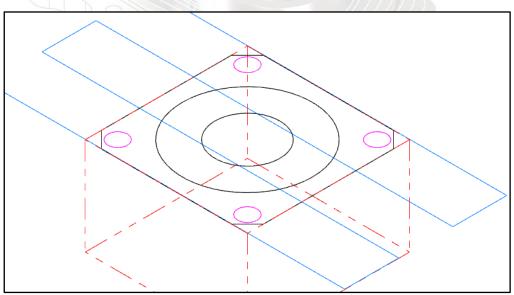
M	love Info	≁ ₽ ×
4	Move Info	
	Move ID	7 of 11
	Elapsed Time	7.57s
	Machine	5_5AXGEN_VMCTTAE
	Move Type	Feed Linear Move
	Operation Name	Facing
	Operation Nu	ID: 1 (1 of 1)
	Tool Number	#1 - FACE MILL - 50/
	Tool Orientation	0.000; 0.000; 1.000
	<b>Tool Tip Position</b>	-55.000; -8.333; 0.100
	Axis Values	
-	Toolpath Info	
	Feed Length	549.996
	Feed Time	12.82s
	Rapid Length	89.800
	Rapid Time	0.43s
	Total Length	639.796
		12.25

#### 3.4 Cirlce Mill and Hole

This section you will learn how to make both two circles mills and holes. At the toolpath tab, go to 2D option and choose Cirlce Mill.



You will see a toolpath display. Press Alt + T to to hide it. To unhide, press it again.



Toolpath Hole Defini	tion	Ψ ×
Selection Advanced		
Features		۲
*/ Type Arc 1	Diameter 40.0	
	140.0	

In top view, click the big circle first. Then click ok.

20	Testestes	Cont	- 6400
20	Toolpaths -	CIrci	e IVIIII

🎙   🔒 👪   🖻 🥶		
Toolpath Type	Status Tool Number Assembly Na Tool Name I	Tool diameter: <mark>50.0</mark> Corner radius: <mark>0.0</mark>
Stock Cut Parameters Cut Parameters Sinch Finishing Simulations Simulation Si		Tool name: FACE MIL Tool #: 1 Head #: 0
Hole Segments Linking Parameters Home / Ref. Points Safety Zone Quick View Settings Tool FACE MILL - 5 Tool Diameter 50	<      Right-click for options      Select library tool      Filter Active      Filter	RCTF     Feed rate: 3.58125     FPT: 0.0003 Plunge rate: 3.58125     Force tool change

Select library tool and filter Endmill1 flat,

Тос	ol List Filter					
	Tool Types			~		
D.S.						
	Endmill1	Flat				
			<b>O</b>		- m	_

Select tool size small than 20mm. We will use same tool size for another 20 mm cricle diameter. No, we choose 12mm diameter tool size.

₩, Т	Tool Selection - C:\Users\Public\Documents\Shared Mastercam 2			
C:\U:	C:\Users\Public\Docu\Mill_mm.tooldb			
	Tool Number	Assembly Na	Tool Name	
	213		FLAT END MILL - 3	
23	214		FLAT END MILL - 4	
83	215		FLAT END MILL - 5	
83	216		FLAT END MILL - 6	
23	217		FLAT END MILL - 8	
23	218		FLAT END MILL - 10	
23	219		FLAT END MILL - 12	
23	220		FLAT END MILL - 14	
83	221		FLAT END MILL - 16	
83	222		FLAT END MILL - 18	
22	223		FLAT END MILL - 20	

The choosen tool will appear at screen.

🖪 2D Toolpaths - Circle Mill						
¥   🔒 👪   🖻 🐣						
Toolpath Type		1		1		Tool diameter: 12.0
Holder		Tool Number	Tool Name	Diameter	Corner Radius	Corner radius: 0.0
Stock	8	2	FLAT END	12.0	0.0	
Cut Parameters	8	1	FACE MILL	50.0	0.0	Tool name: FLAT END MIL
Roughing ⊡ Finishing						Tool #: 2
·····⊘ Transitions ····⊘ Depth Cuts						Head #: 0
Break Through						
Limits						
Hole Segments						
Linking Parameters						RCTF S
Home / Ref. Points						

Go to Cut Parameter tab and set the value stock to leave on walls and floor at 0.1 mm. Go to roughing tab and activate it.

Stepover —	<mark>50.0</mark> %	6.0			
🗹 Helical Entry			$\sim$		
Minimum radius	<mark>10.0</mark> %	1.2			
Maximum radius	<mark>45.0</mark> %	5.4			
XY clearance		0.2	- UC	<b>WARRENT</b>	
Z clearance		2.0			
Plunge angle		3.0			
If helix fails					
O Plunge	🖲 Skip				

Depth cuts			
Max rough step:	12.0		
Finish			
Number of cuts:	0		
Step:	0.5		
- Override Feed Speed			
Eed rate	2992.5376		
Spindle speed	8992		
🗹 Keep tool down		Tapered walls	
Subprogram		Taper angle	0.0
Absolute O In	cremental		

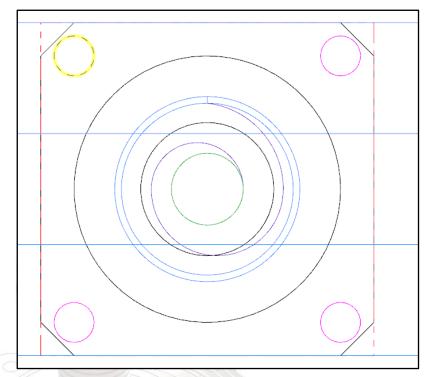
Activate the depth cuts of rough at 12 step with keep tool down.

Go to linking parameter and set value as figure shown.

	Arc fit maximum radius	12.0
	Output feed move	13000.0
	Clearance 125.0	Absolute
		Incremental
	Use clearance only at the start and	Associative l end of operation
	Retract 10.0	Absolute
		Incremental
<u> </u>		<ul> <li>Associative</li> </ul>
	Feed plane 5.0	<ul> <li>Absolute</li> </ul>
		Incremental
		<ul> <li>Associative</li> </ul>
	Top of stock 0.0	Absolute
		Incremental
		<ul> <li>Associative</li> </ul>
	Depth	Absolute
		<ul> <li>Incremental</li> </ul>
		<ul> <li>Associative</li> </ul>

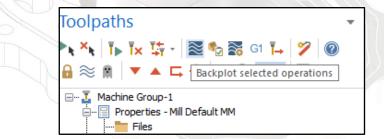
For begineer, we will use absolute coordinate. Put the depth value at -6.0 mm and click ok.

You will show the cutting toolpath as shown.

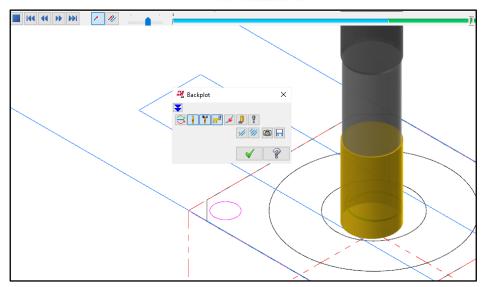


Next step is to backplot the toolpath. This backplot shows the path of tools.

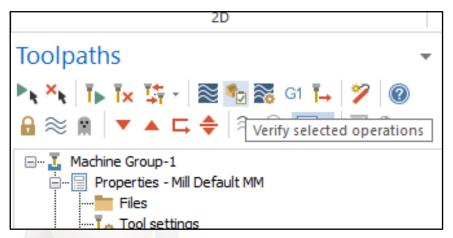
This function will let you spot errors in the program before machining process. Go to backplot icon.



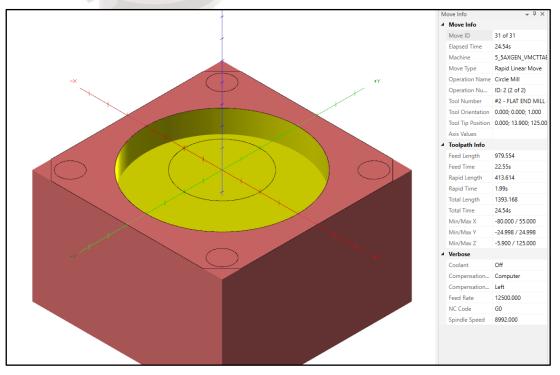
Play the backplot operation.



Final step for this circle mill process, simulate the toolpath in verify. This function is to shows the path of the tools to cut the stock or material.



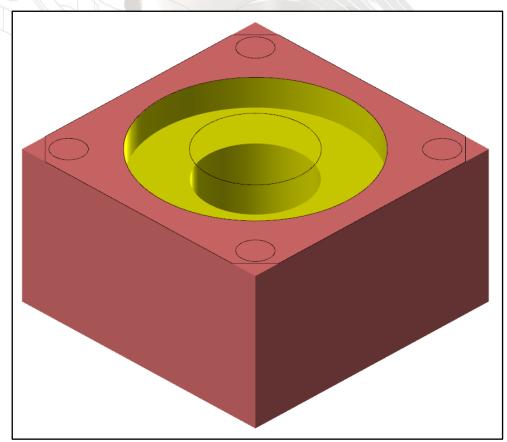
In verify mood, you will see a detail information about milling process such as feed lenght, feed time and total time.



Now, we go to second circle. Same step are applied for this small circle. We will use same toolpath type, cut parameter, roughing and finishing. New setup applied on linking parameter only.

	Calculate incremental values from hole	s/lines
	Automatic linking	12.0
	Output feed move	13000.0
	Clearance 125.0	Absolute
<b>T</b>		<ul> <li>Incremental</li> <li>Associative</li> </ul>
	Use clearance only at the start an	d end of operation
	Retract 10.0	Absolute
		Associative <sup>(0)</sup>
	Feed plane 5.0	
↓		Incremental Associative
	Top of stock	Absolute
		<ul> <li>Incremental</li> <li>Associative</li> </ul>
r i i i i i i i i i i i i i i i i i i i	Depth	Absolute
		<ul> <li>Incremental</li> <li>Associative</li> </ul>
	Calculate depth from top of stock	
Check Collisions		

When the previous depth is -6.0mm, so the new top of stock is also same (original coordinate). Thats mean, the feeding process will start at absolute coordinate z: -6.0mm. The new depth z is -24mm.



In this section, drill operation is needed. We will use spot drill for 6mm holes. Clik drill.

Contour Drill Dynamic Face	<ul> <li>↓</li> <li>↓</li> <li>OptiRough</li> </ul>	Pocket SD	Parallel →	Curve Swarf Milli	Unified Parallel	-         Stock         Stock         Stock           -         -         Shading Display Model         Stock
Toolpath Hole Definition	<del>.</del> т ×	Select one or more e	entities to add t	to or remove from the	Features list.	
Selection Advanced		- [Ctrl+click] to select	all matching ra	adius solid features. hing radius solid featu		its or AutoCursor position ctor as the initial selection
Features	•	- [Ctrl+Double-click]	to select all soli	id holes of the same ty		
*/ Type Diameter	·	<ul> <li>Click on a selected</li> </ul>	solid feature's	arrow to change direc	tion.	
Arc 1         6.0           Arc 2         6.0           Arc 3         6.0           Arc 4         6.0						
Depth Filters						
<ul> <li>Use highest Z depth</li> <li>Use lowest Z depth</li> <li>Off</li> </ul>						
Sort	۲				/	
Selected Order	~					
Selected Sort Insert point: O Top of list	۲		$\overbrace{+}{}$			
2D Sort Rotary Sort	•		$\checkmark$			

Click the center of 4 holes and 4 arc measurement will display at left display.

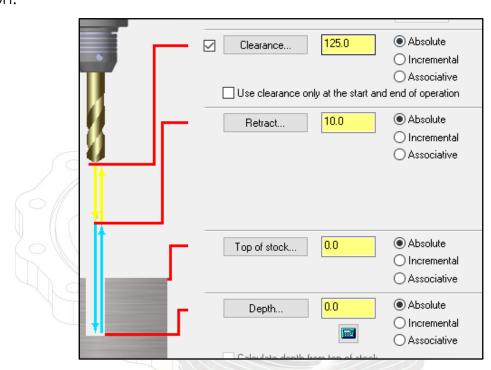
Tool List Filter Tool Types Tool Diameter ~ 6.0 Equal I Radius Type U m 🗹 None Corner 🗹 Full 13 Drill Tool Material ų D 🗹 HSS 🗹 Ceramic 🗹 Carbide 🗹 User Def 1 ? V U 🗹 Ti Coated User Def 2

In toolpath setup, we will use new tool type, drill.

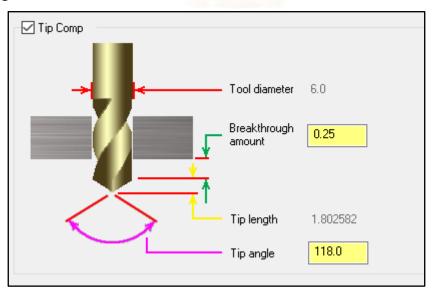
Change Tool Diameter to Equal 6.0. Selected desired tool drill.

₩, т	ool Selection	- C:\Users\Publ	ic\Documents\Shared N	Aastercam 2022	\Mill\Tools\	Mill_mm.tooldb
C:\U	sers\Public\Do	cu\Mill_mm.toc	oldb 🗃			
	Tool Number	Assembly Na	Tool Name	Holder Name	Diameter	Corner Radius
22	29		SOLID CARBIDE DRI		6.0	0.0
2	30		HSS/TIN DRILL 8xD		6.0	0.0

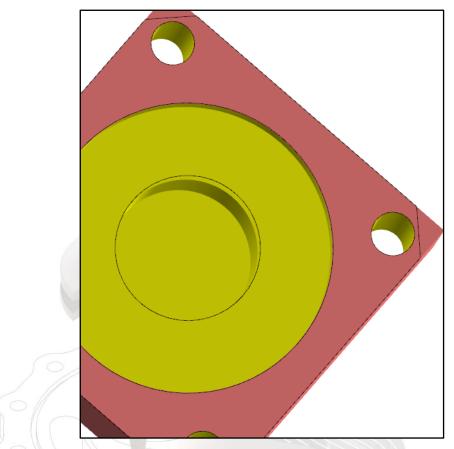
Choose Linking Parameter and enable the clearance. Click the calculator icon.



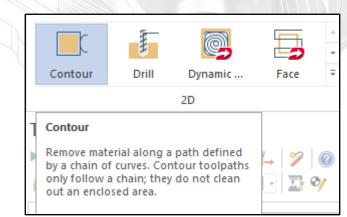
To generate 1.0 chamfer, key in -30 in depth box. Activate the tip comp for breakthrough.



Simulate by verify the operation

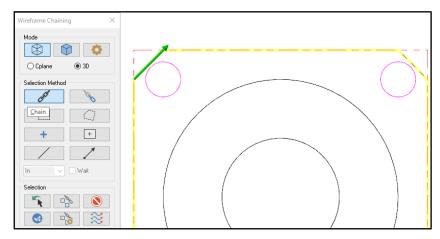


Last operation is to cut 4 corner of stock. We will use contour function to finished it.

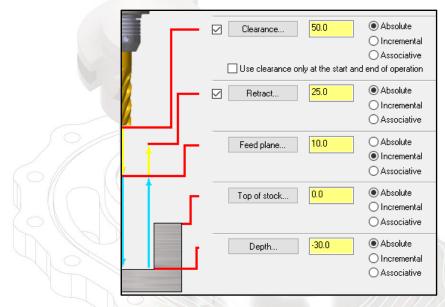


Use chain selected method and click outline of stock. You will see green arrow appear. You can change direction of arrow by click the reverse icon

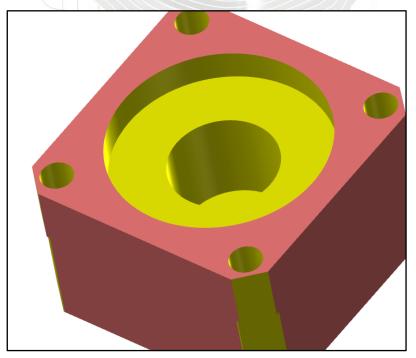




By using same flat endmill tool and set the linking parameter as figure shown.



The finished stock will be shown as figure.

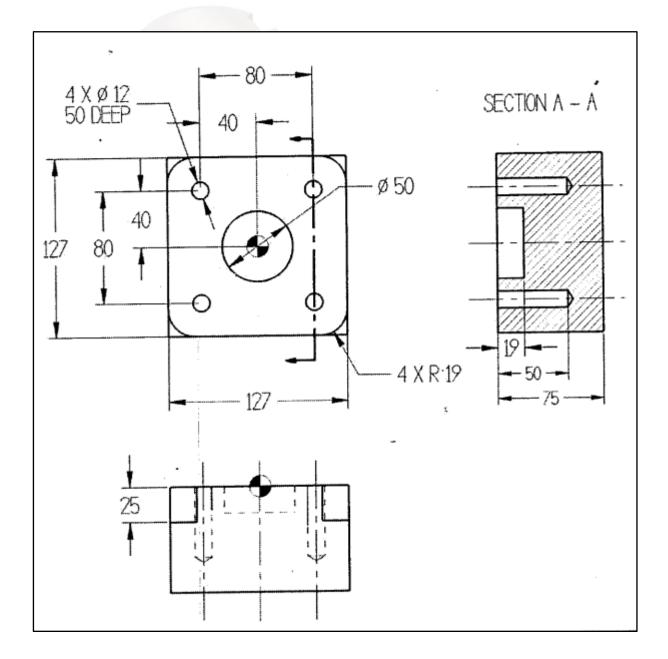


## EXCERCISE

This chapter has been developed by Dr. Mohd Shahrom Bin Ismail

Use the commands from the wireframe tab to create the geometry as follows;

- ✤ Rectangle
- Fillet Entities
- Circle Center Point
- ✤ Trim



Instruction;

- ✓ Use 12mm Flat Endmill
- ✓ Set Max rough step to 6mm
- $\checkmark~$  For drill the holes, use a 20mm NC Spot Drill
- $\checkmark~$  Set the cycle to Drill/counterbore and set Dwell of 1.0 second







**ISSUED:** 



Politeknik Sultan Salahuddin Abdul Aziz Shah Persiaran Usahawan, Seksyen UI, 40150 Shah Alam, Selangor Darul Ehsan Tel : 03-51634000 https://psa.mypolycc.edu.my/

