POLITEKNIK MALAYSIA

SULTAN SALAHUDDIN ABDUL AZIZ SHAH

MASTERCAM

TUTORIAL 1

Mastercam UNIVERSITY

Mechanical Workshop Practice 4

MECHANICAL ENGINEERING DEPARTMENT

BASIC MILL

MASTERCAM TUTORIAL 1

BASIC MILL

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.

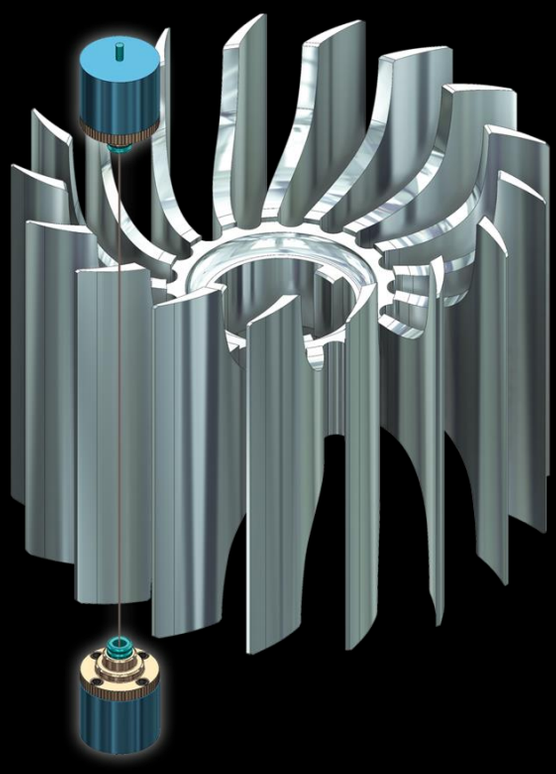
MECHANICAL WORKSHOP PRACTICE 4 – MASTERCAM BASIC MILL TUTORIAL 1

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

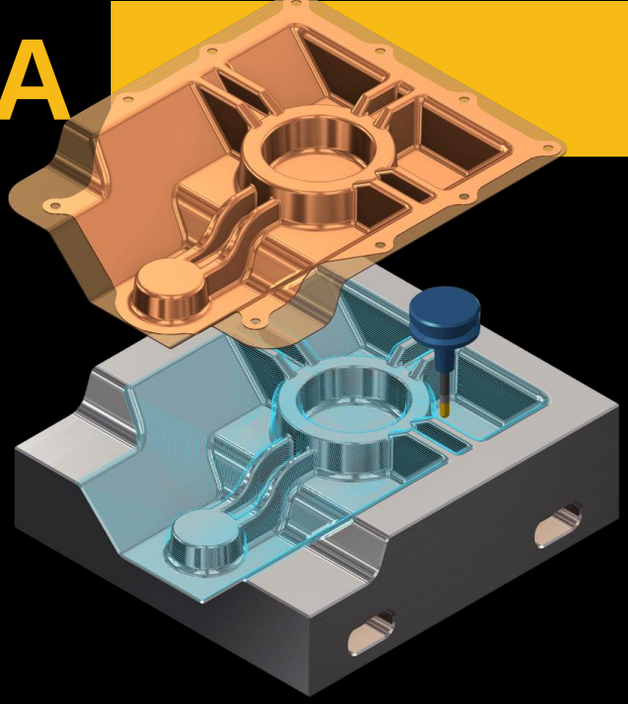


First Edition 2022

Published by:

UNIT PENERBITAN
Politeknik Sultan Salahuddin Abdul Aziz Shah Persiaran Usahawan,
Seksyen U1, 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.

PREFACE

Alhamdulillah, all praise and thanks to Allah ﷻ for His mercy and grace to all of us as the people of Prophet Muhammad ﷺ. Mechanical Workshop Practice 4 – Mastercam Basic Mill Tutorial 1 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..

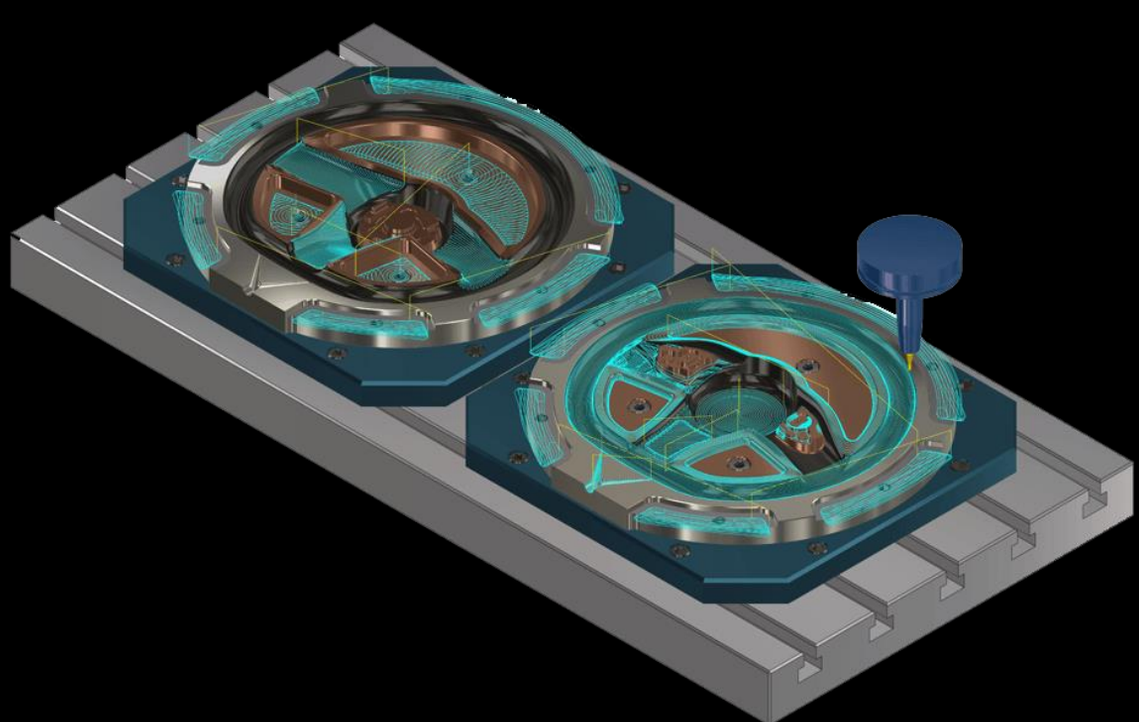
CONTENTS

1 **INTRODUCTION**
Pages 2

2 **WIREFRAME CREATION**
Page 7

3 **TOOLPATH CREATION (MACHINING PROCESS).**
Pages 16

4 **EXCERCISE**
Pages 40



CONTENTS

TOPIC	PAGES
APPRECIATION	
ABSTRACT	
INTRODUCTION.	**2**
1.1 Starting MasterCAM Software	
1.2 GUI – Graphical User Interface	
1.3 Manager Panels	
WIREFRAME CREATION.	**7**
2.1 Setting Up GUI	
2.2 Rectangle	
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

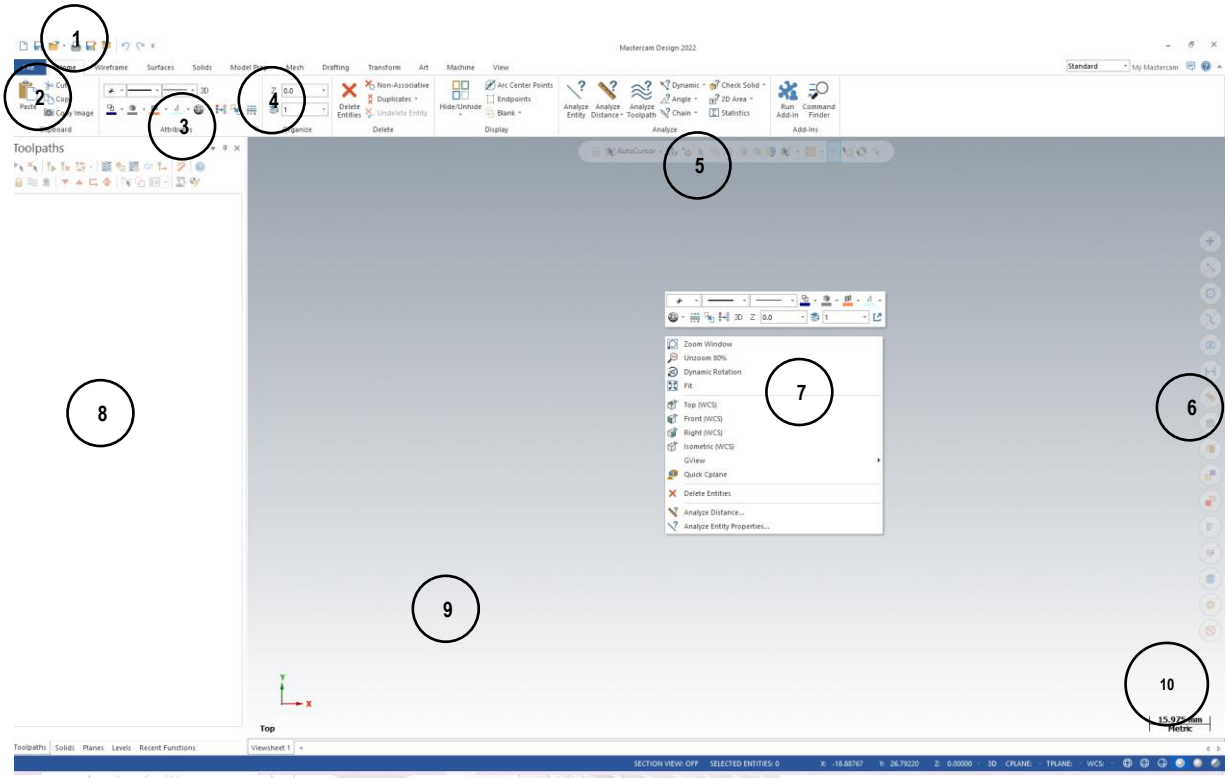
1.1.2 For Windows 8

- 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

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

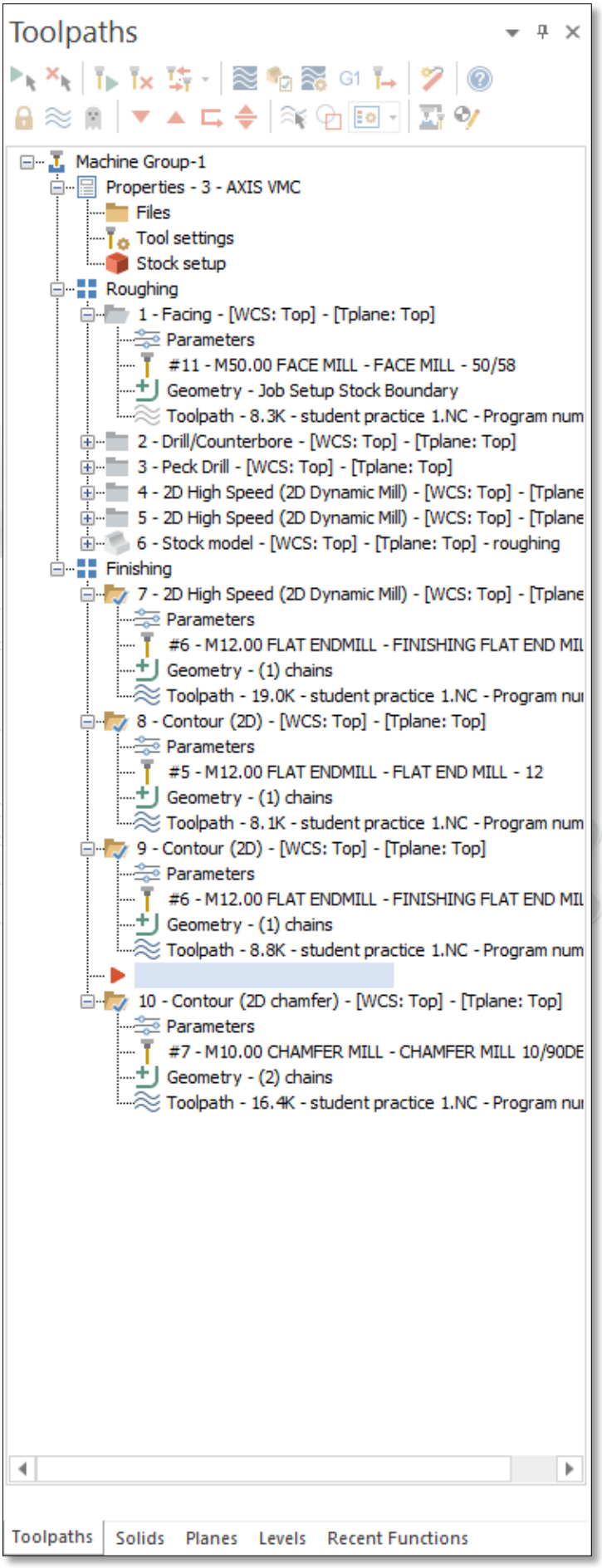
Show you a scale of the object on the screen.

11. WCS: TOP T/Cplane

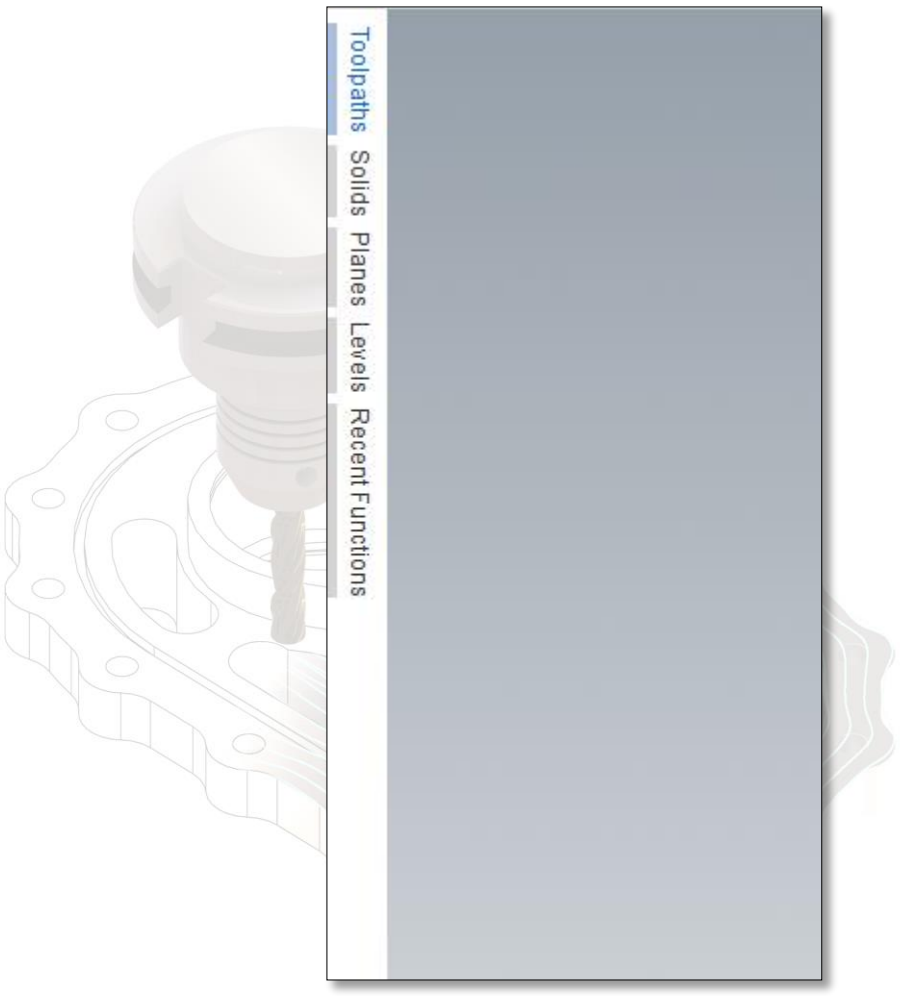
Display the current WCS and T/Cplane information

1.3 Manager Panels

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


Toolpaths
Machine Group-1
Properties - 3 - AXIS VMC
Files
Tool settings
Stock setup
Roughing
1 - Facing - [WCS: Top] - [Tplane: Top]
Parameters
#11 - M50.00 FACE MILL - FACE MILL - 50/58
Geometry - Job Setup Stock Boundary
Toolpath - 8.3K - student practice 1.NC - Program num
2 - Drill/Counterbore - [WCS: Top] - [Tplane: Top]
3 - Peck Drill - [WCS: Top] - [Tplane: Top]
4 - 2D High Speed (2D Dynamic Mill) - [WCS: Top] - [Tplane
5 - 2D High Speed (2D Dynamic Mill) - [WCS: Top] - [Tplane
6 - Stock model - [WCS: Top] - [Tplane: Top] - roughing
Finishing
7 - 2D High Speed (2D Dynamic Mill) - [WCS: Top] - [Tplane
Parameters
#6 - M12.00 FLAT ENDMILL - FINISHING FLAT END MIL
Geometry - (1) chains
Toolpath - 19.0K - student practice 1.NC - Program nu
8 - Contour (2D) - [WCS: Top] - [Tplane: Top]
Parameters
#5 - M12.00 FLAT ENDMILL - FLAT END MILL - 12
Geometry - (1) chains
Toolpath - 8.1K - student practice 1.NC - Program num
9 - Contour (2D) - [WCS: Top] - [Tplane: Top]
Parameters
#6 - M12.00 FLAT ENDMILL - FINISHING FLAT END MIL
Geometry - (1) chains
Toolpath - 8.8K - student practice 1.NC - Program num
10 - Contour (2D chamfer) - [WCS: Top] - [Tplane: Top]
Parameters
#7 - M10.00 CHAMFER MILL - CHAMFER MILL 10/90DE
Geometry - (2) chains
Toolpath - 16.4K - student practice 1.NC - Program nu
Toolpaths
Solids
Planes
Levels
Recent Functions

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.



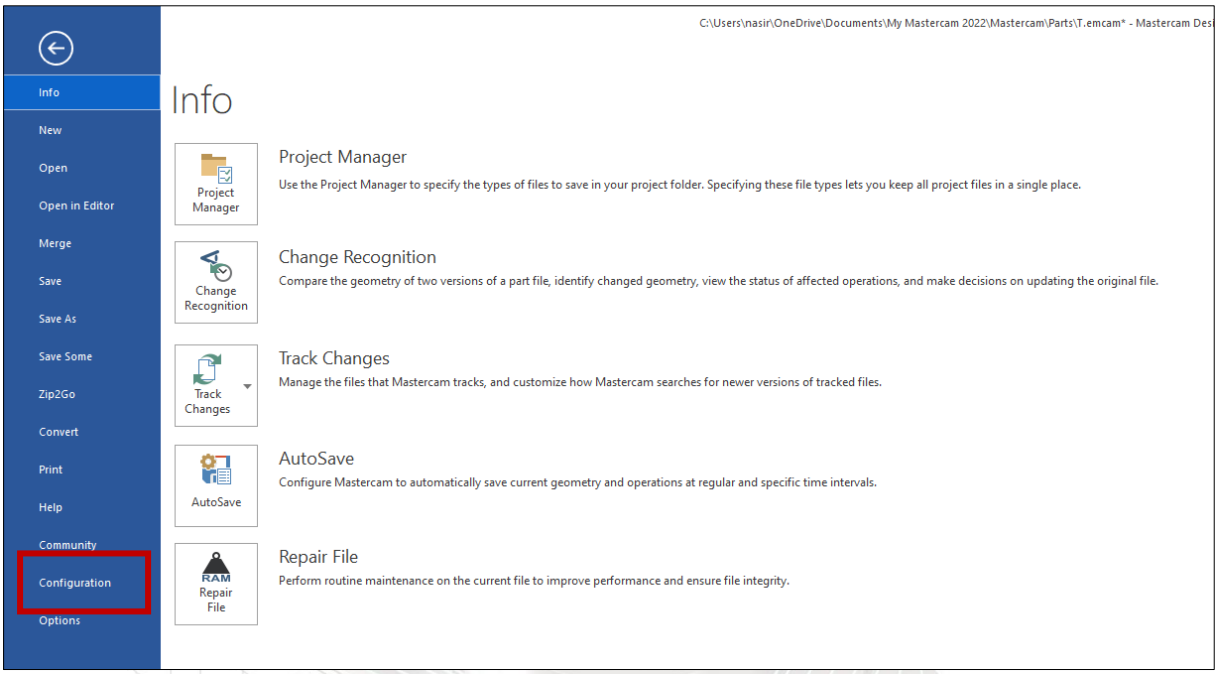
To un-hide them, click on one of the managers to open it and then click again on the auto hide icon a shown. Selecting the close icon, will close the manager panel.

WIREFRAME CREATION

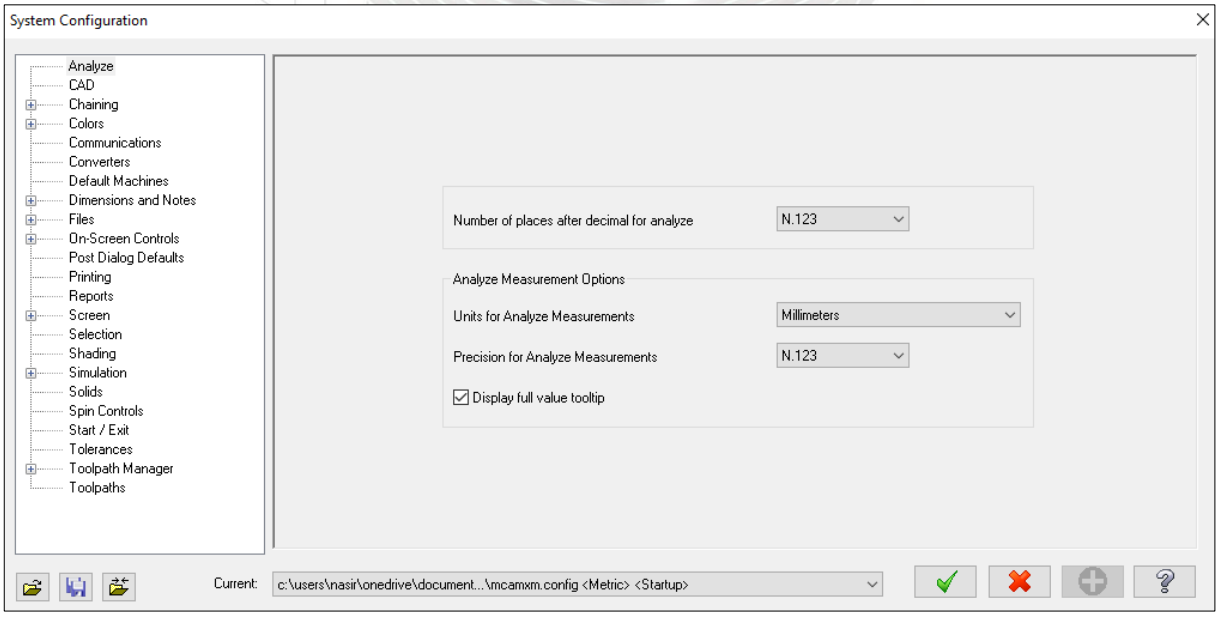
This chapter has been developed by Ts Salhana Binti Sahidin@Salehudin

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.



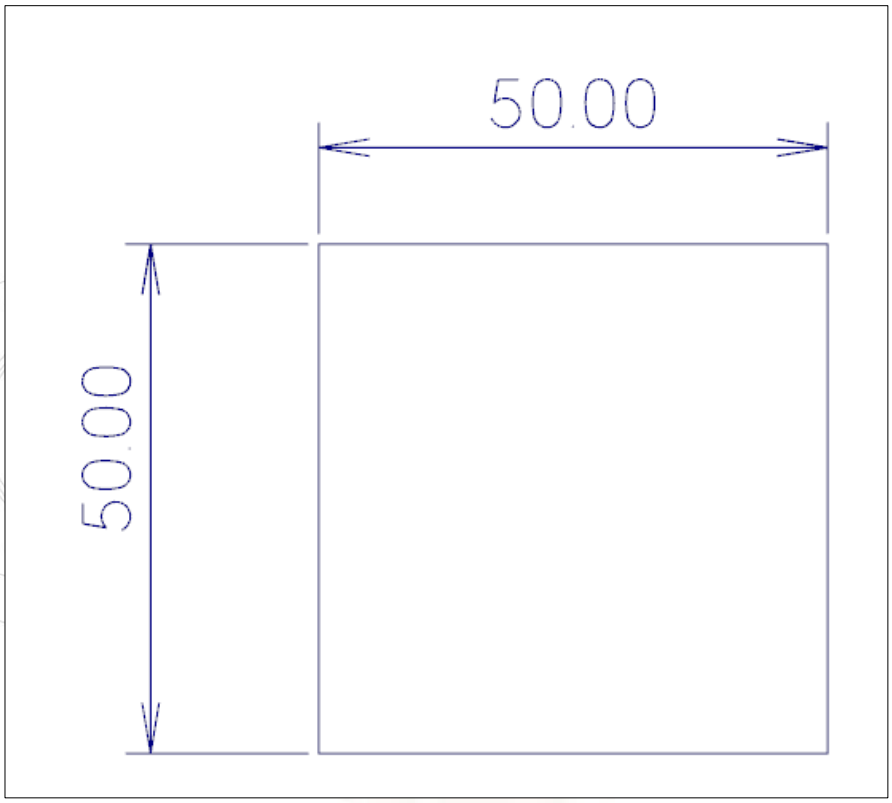
Then, select the prefer units, either inch or metric.



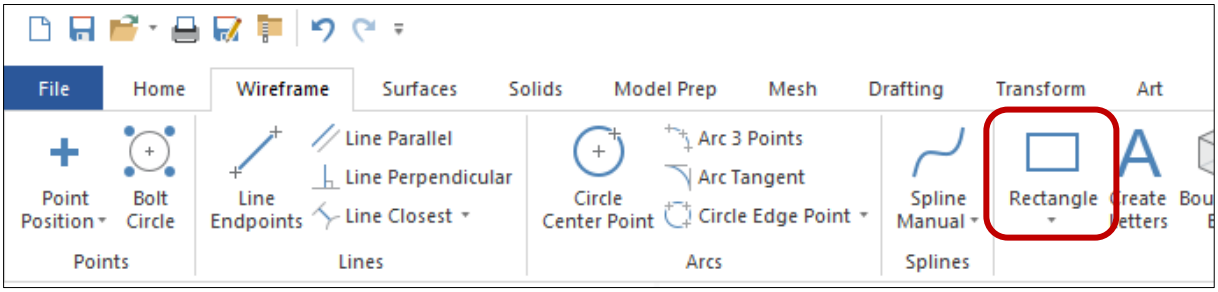
2.2 Rectangle

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.

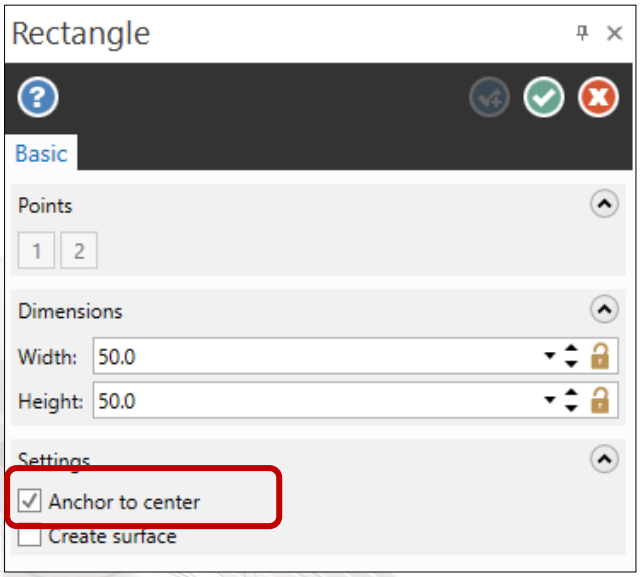
Step Preview;



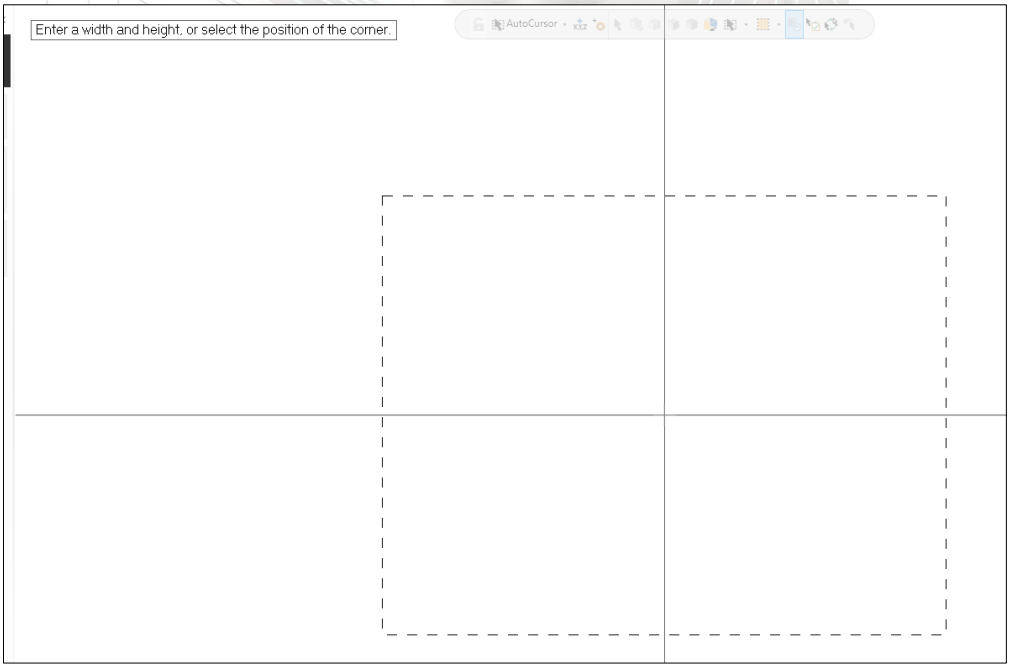
Create a *50 mm* by *50 mm* Rectangle



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

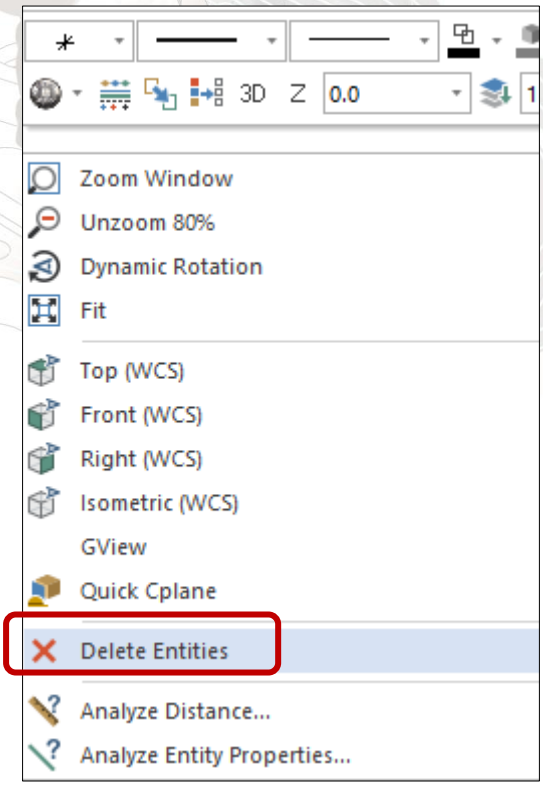


Note: Don't select that create surface. Anchor to center sets the base point on the rectangle to its center and draws the rectangle outward from the center



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.

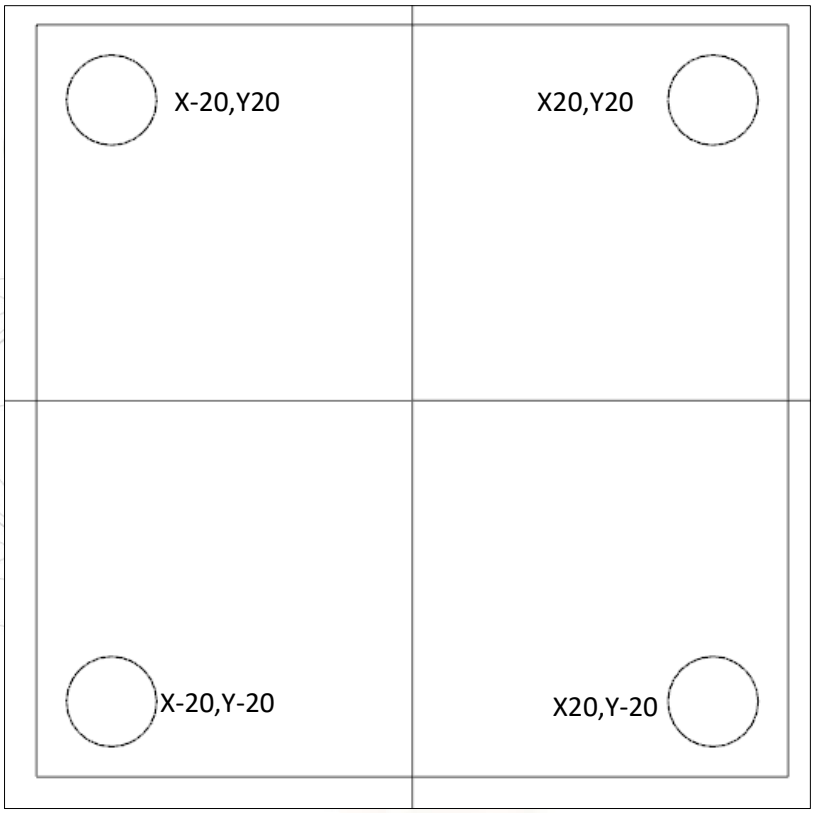
While creating wireframe, you can undo the last step using the Undo icon . You can undo as many step as need. You also can redo by using the Redo icon . To delete unwanted geometry, selec the geometry first and then press Delete from the keyboard or right click on mouse then click delete entities .



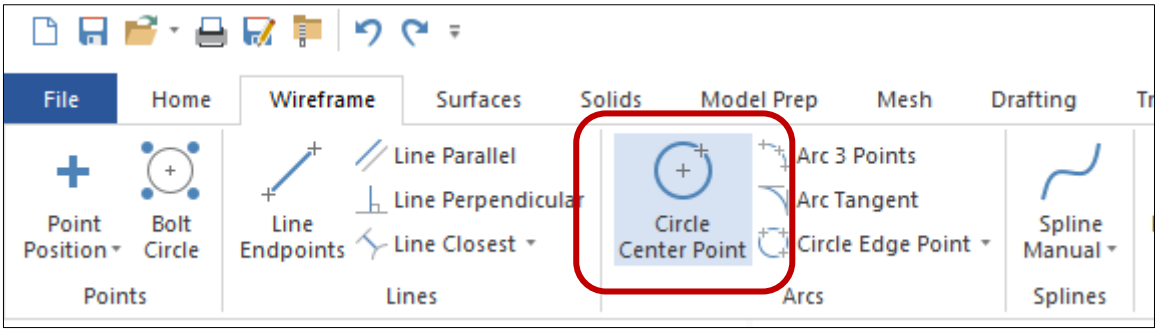
To zoom, move the cursor in the center of the geometry and scroll the mouse wheel.

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.

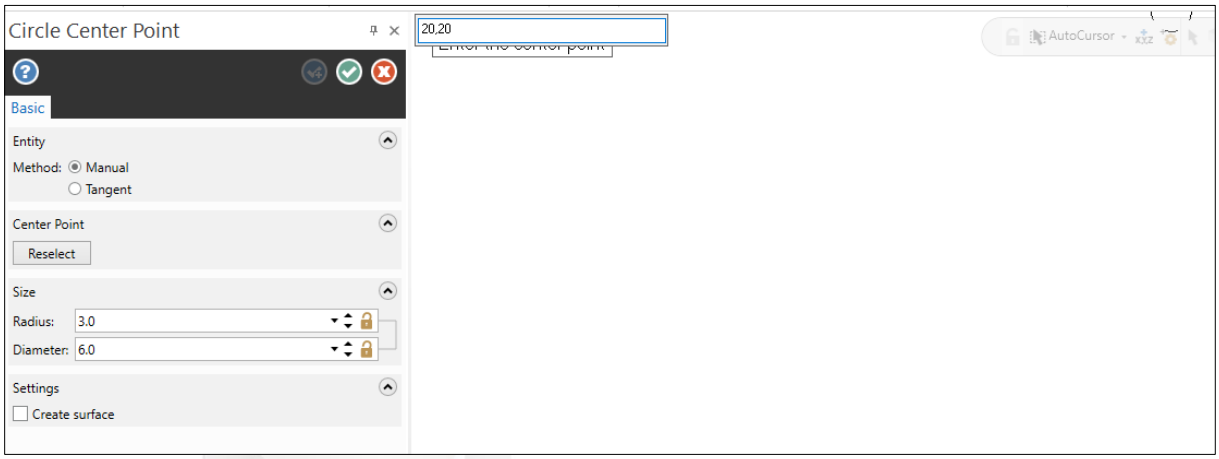


From the Arcs group, select Circle Center Point.



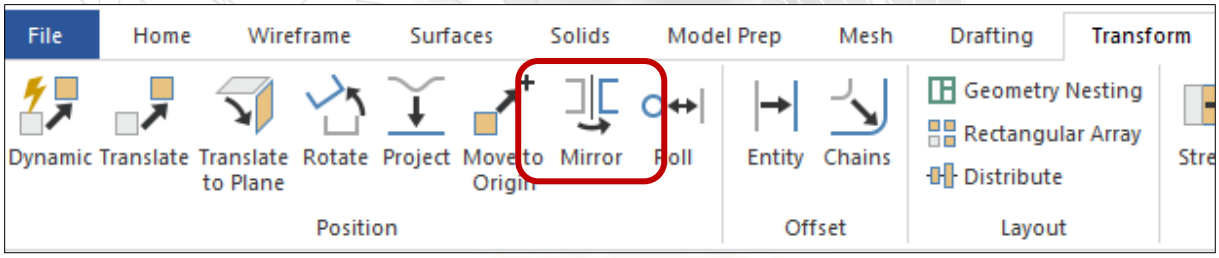
Enter a Radius of 3.0 mm the panel as shown. You can lock the value by click on the locker icon. So, it will lock for making another three circles. Select the AutoCursor icon from the selection toolbar and the field where you can type

the coordinates will open at the upper left side of the graphics windows as shown

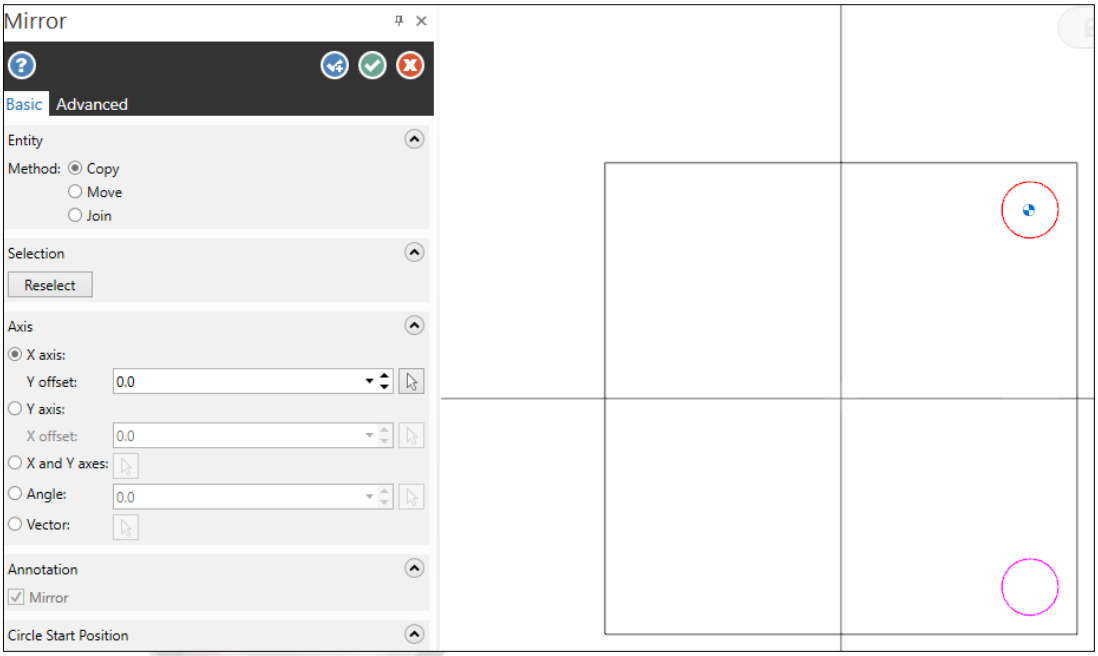


Type 20,20 as shown. When you fill the coordinates, the first value is represent x value/axis and the second value represent y value/axis. Key in the coordinate another three postion with this value; 20,-20, -20,20 and -20,-20. Once complete, choose the OK button to exit the command.

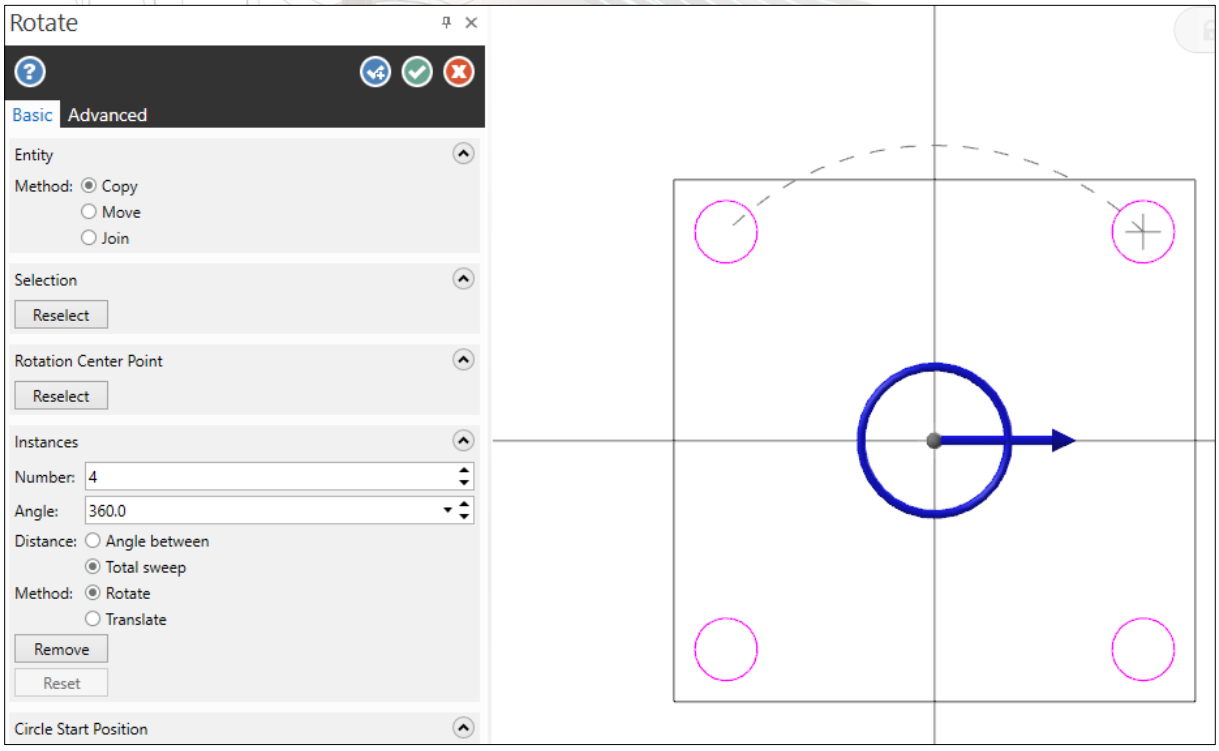
Another method, you also can use mirror function. Go to transform tab. Click on mirror icon.



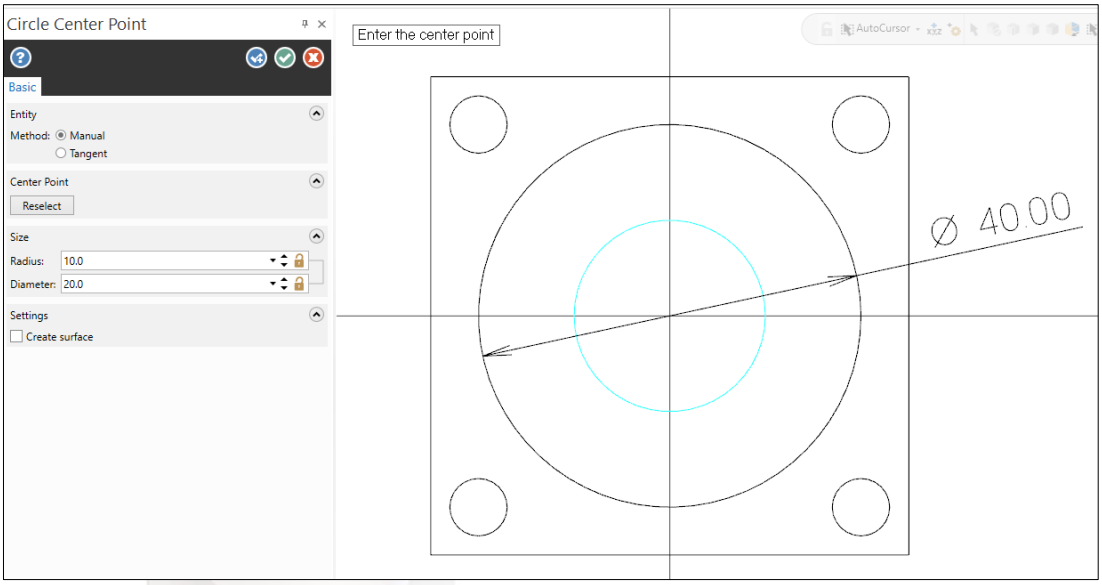
Then select the entities to mirror on X axis.



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.



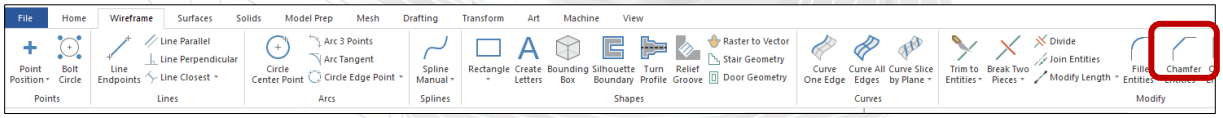
Insert the data number is 4 (circles), angle 360° and distance total sweep. Next, create another circle center point for diameter 40mm and 20mm at origin point.



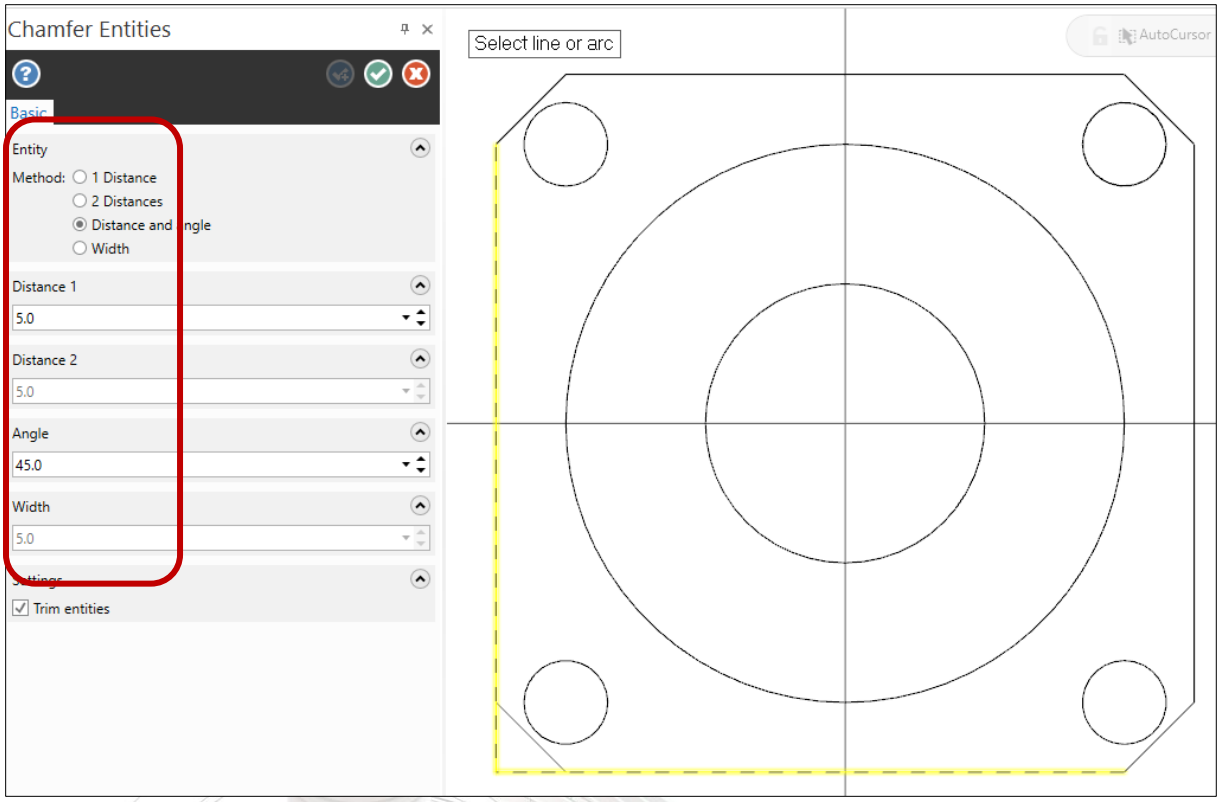
While the circle is live, cyan color, the circle diameter can be modified. When you enter to finish the circle, you cannot edit the circle. Either you undo or delete and make a new circle.

2.4 Chamfer

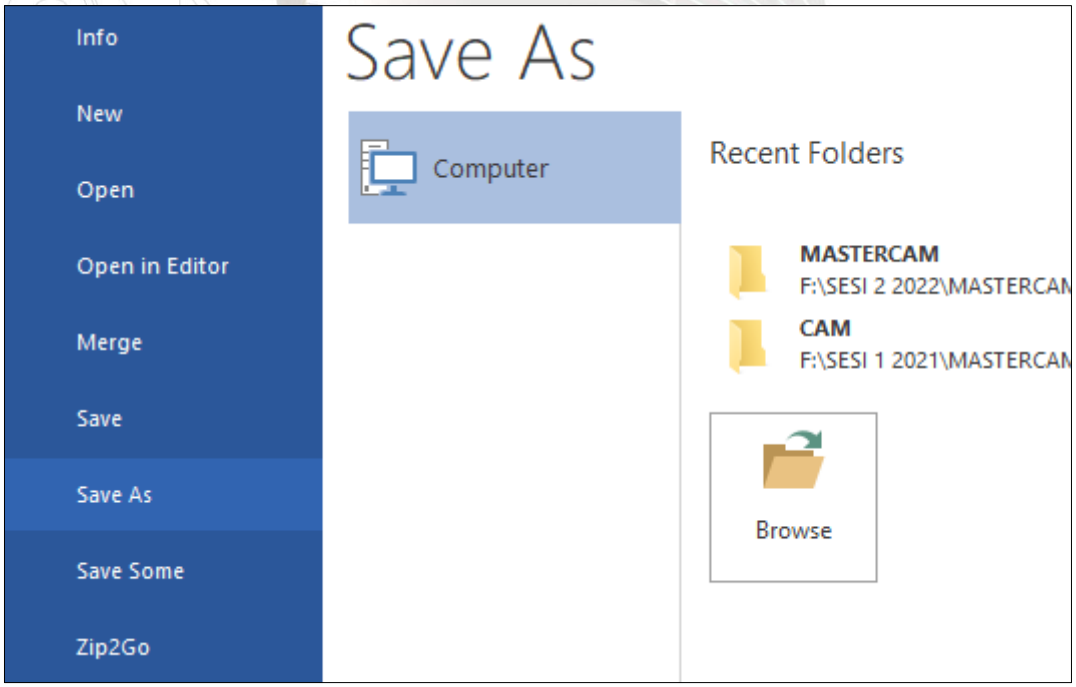
In this section, you will create 45° chamfers at each corners of the rectangle. First, go to wireframe tab and then click the chamfer icon.



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;

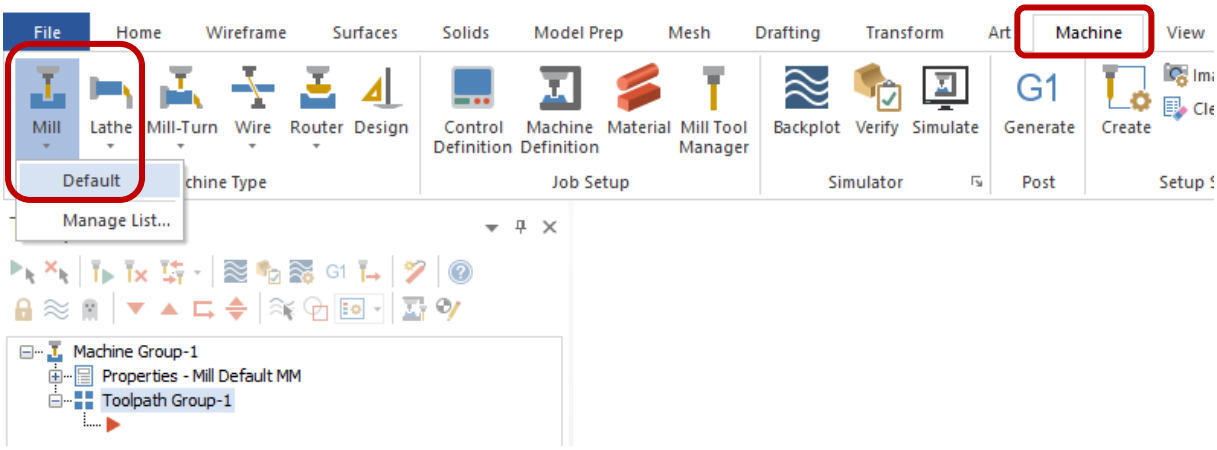
- ✓ Type of Machine (depend on your machine brand)
- ✓ Material Selection
- ✓ Type of Machining/Toolpath
- ✓ Type of Tool and diameter
- ✓ Facing, Milling and Finishing procedure
- ✓ Post Processor for Machining

For this basic tutorial we will use

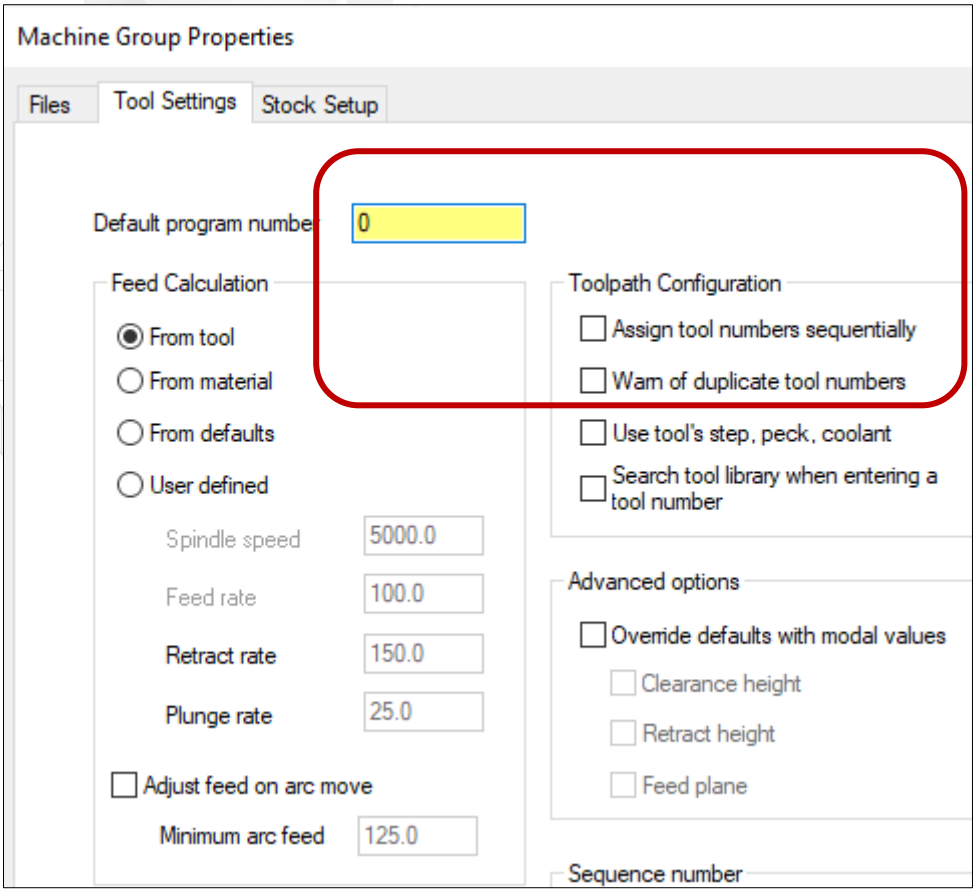
- ✓ Default CNC Machine
- ✓ Aluminum Material
- ✓ Face Mill, Circle Mill, Drill, Contour and finishing Toolpath
- ✓ Face Mill Flat End Mill and Dril Cutting Tools

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.

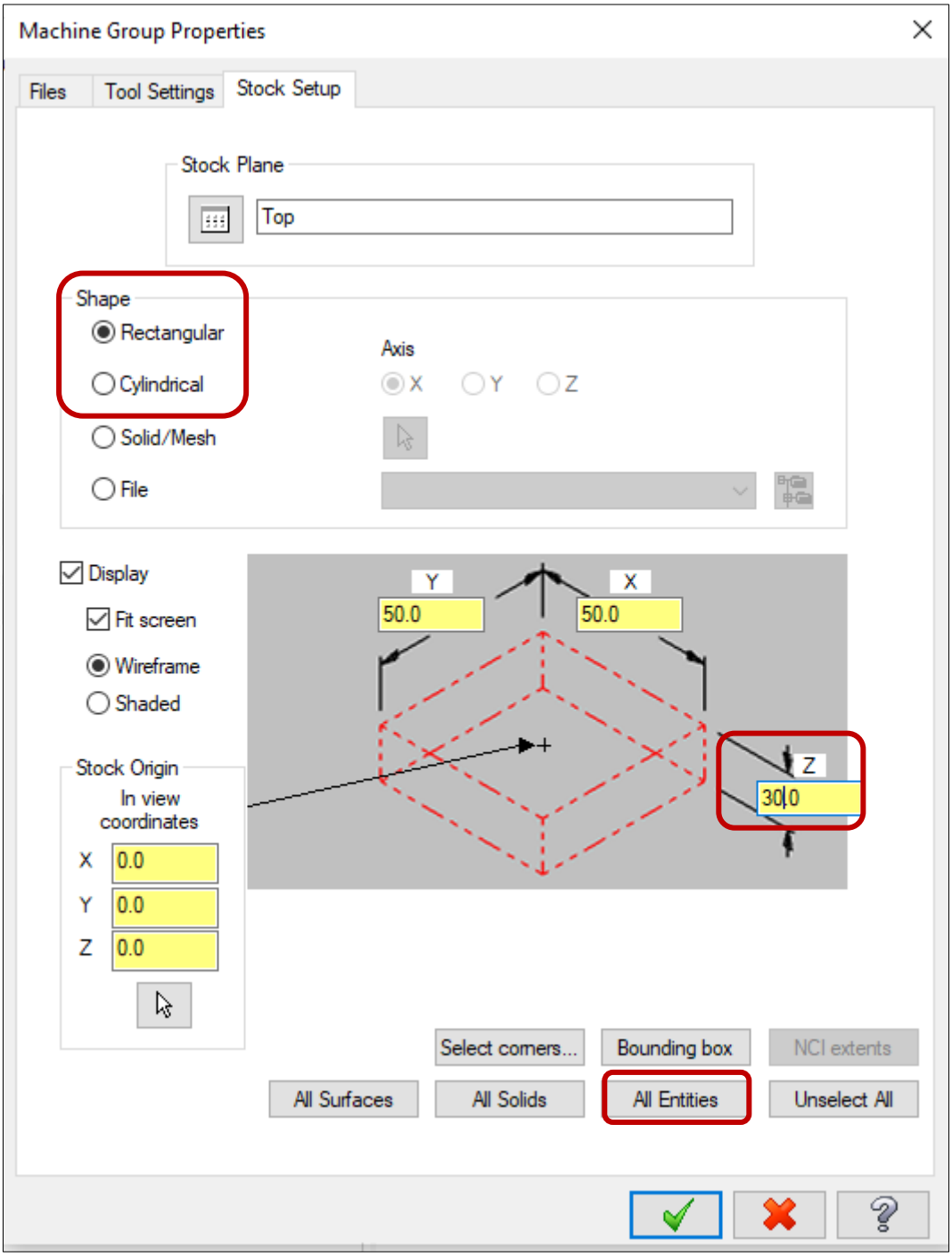


Next, expand the properties and click tool setting.

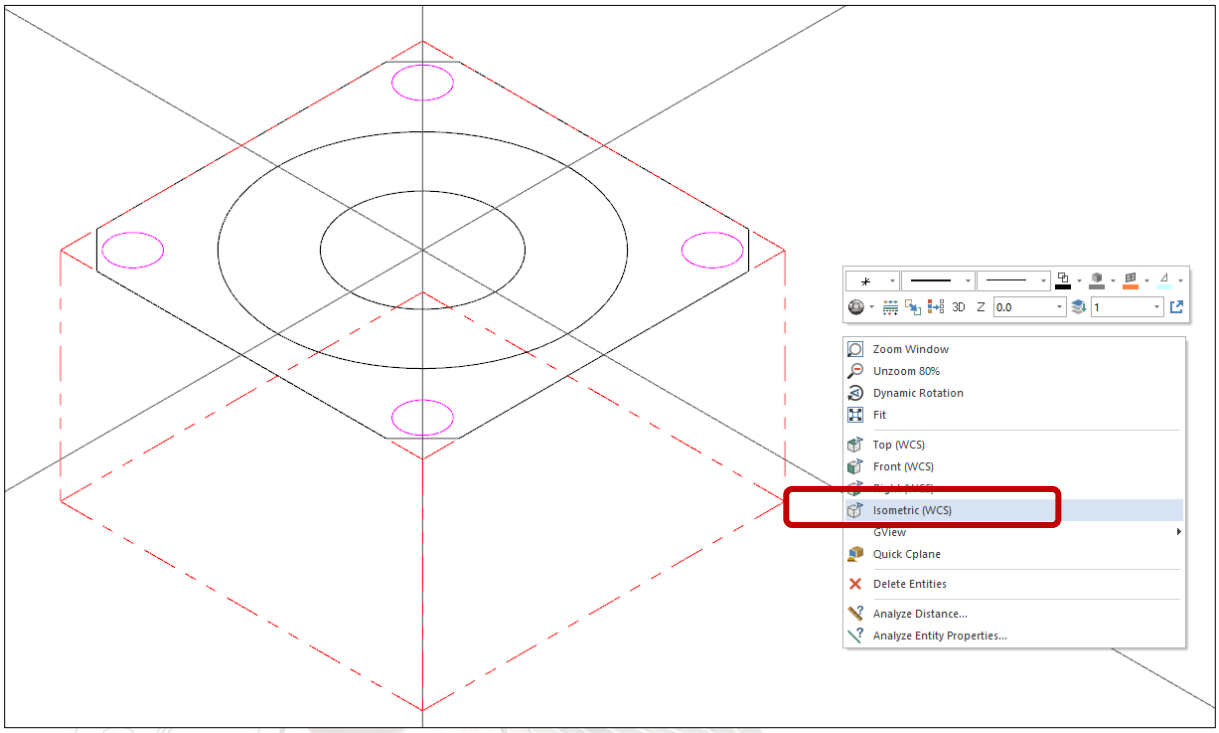


You can set program number. This function is to make sure the machine recall the tool setting memory by reading the program number. Basically, you can active the toolpath configuration by pick the assing tool number and warn of duplicate tool numbers.

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.



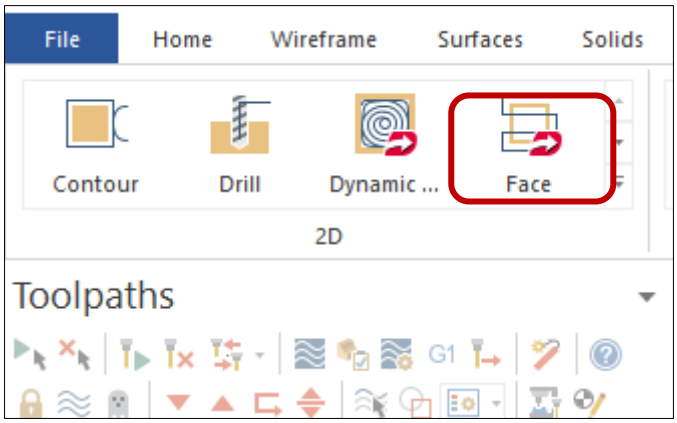
Right click your mouse and click the isometric view to view the stock that you have set. You will see the stock as a shown.



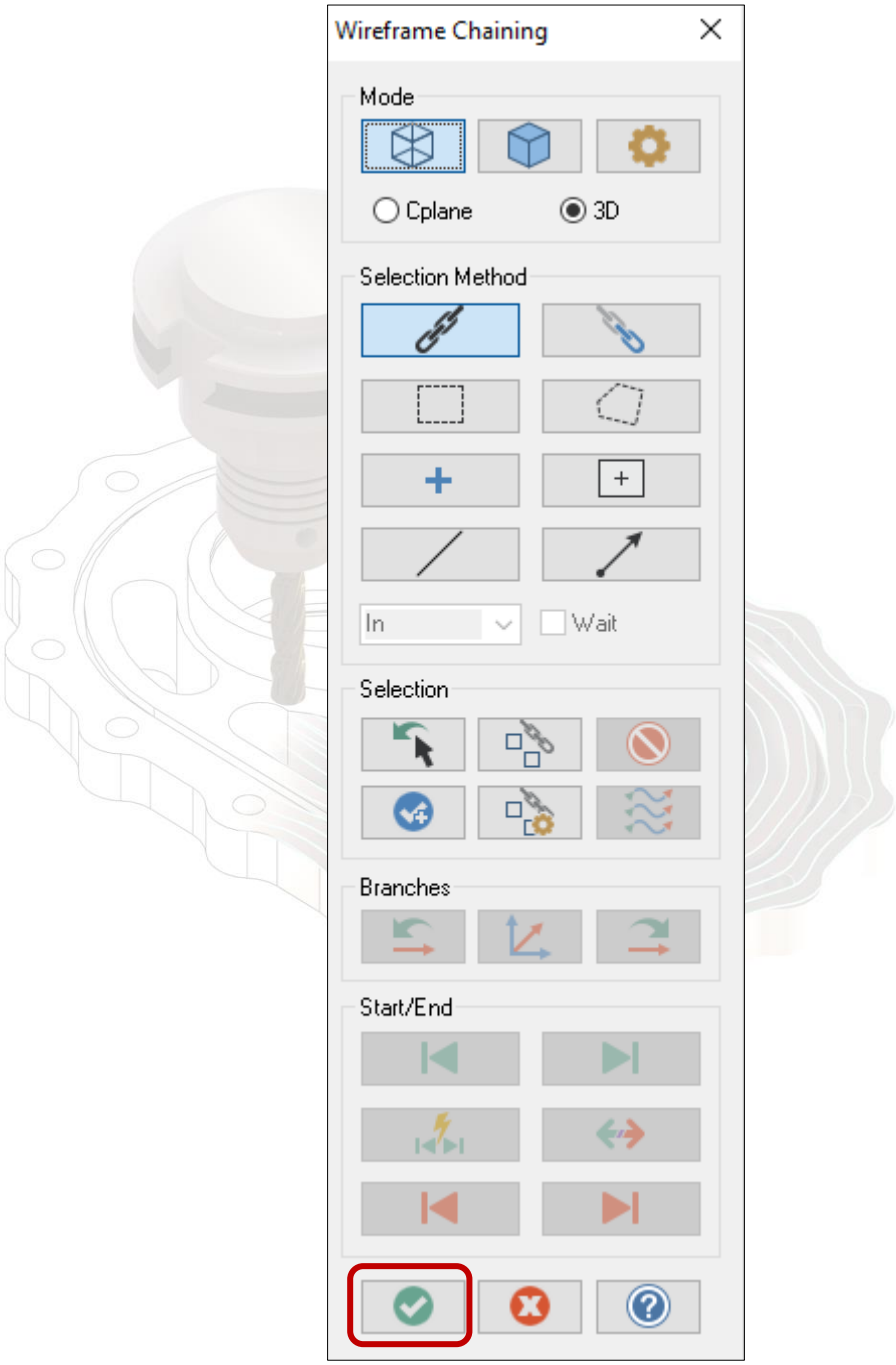
3.3 Face Mill

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.

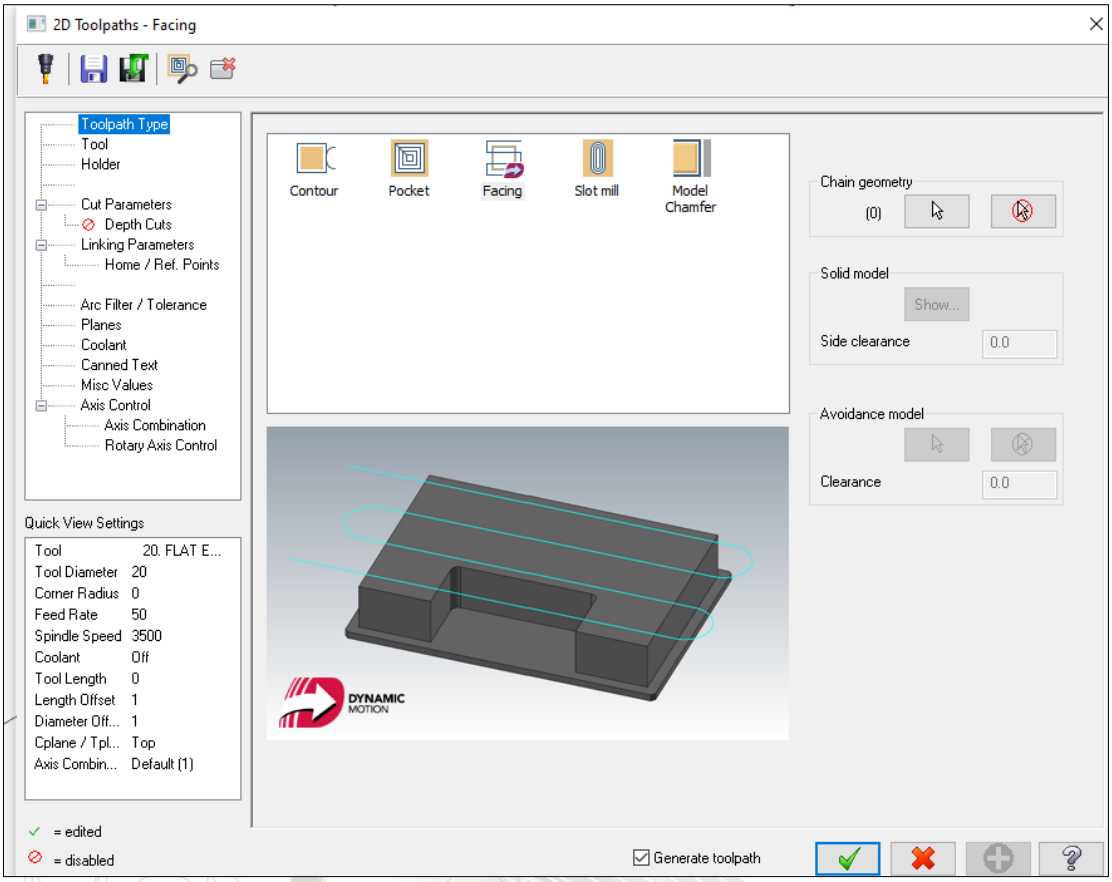
At the toolpath tab, go to 2D option and choose Face Mill.



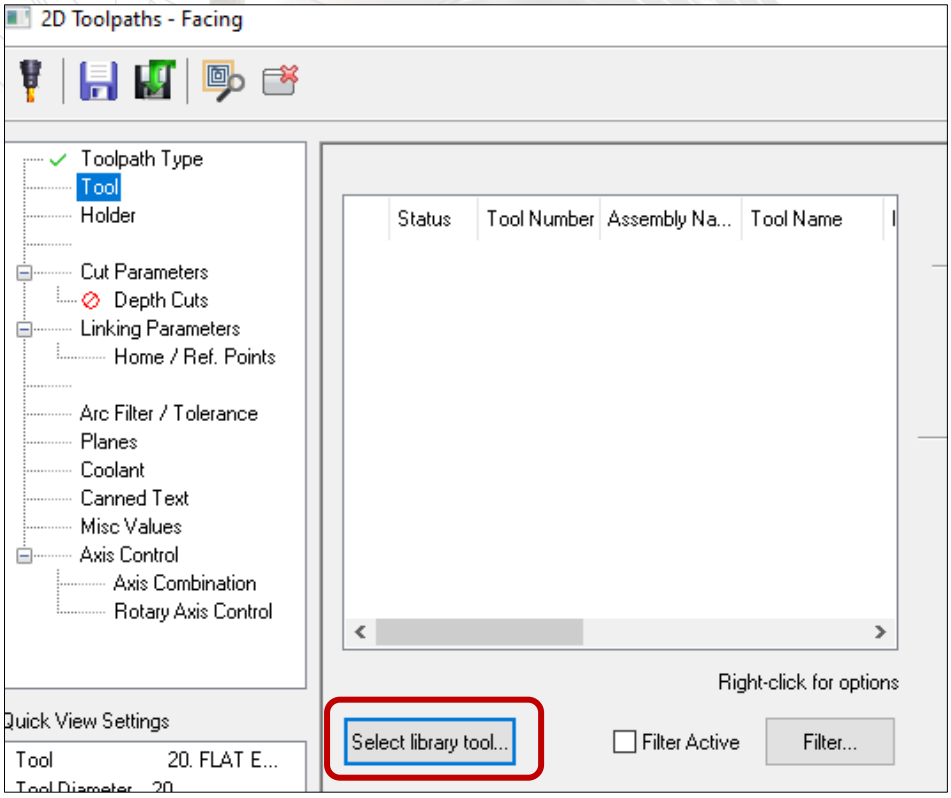
You will see wireframe chaining popup. For facing process you don't need to click the chain at any enttites of your stock. Sotware will automatically detect the top of your shape stock. It will read as a face mill location.



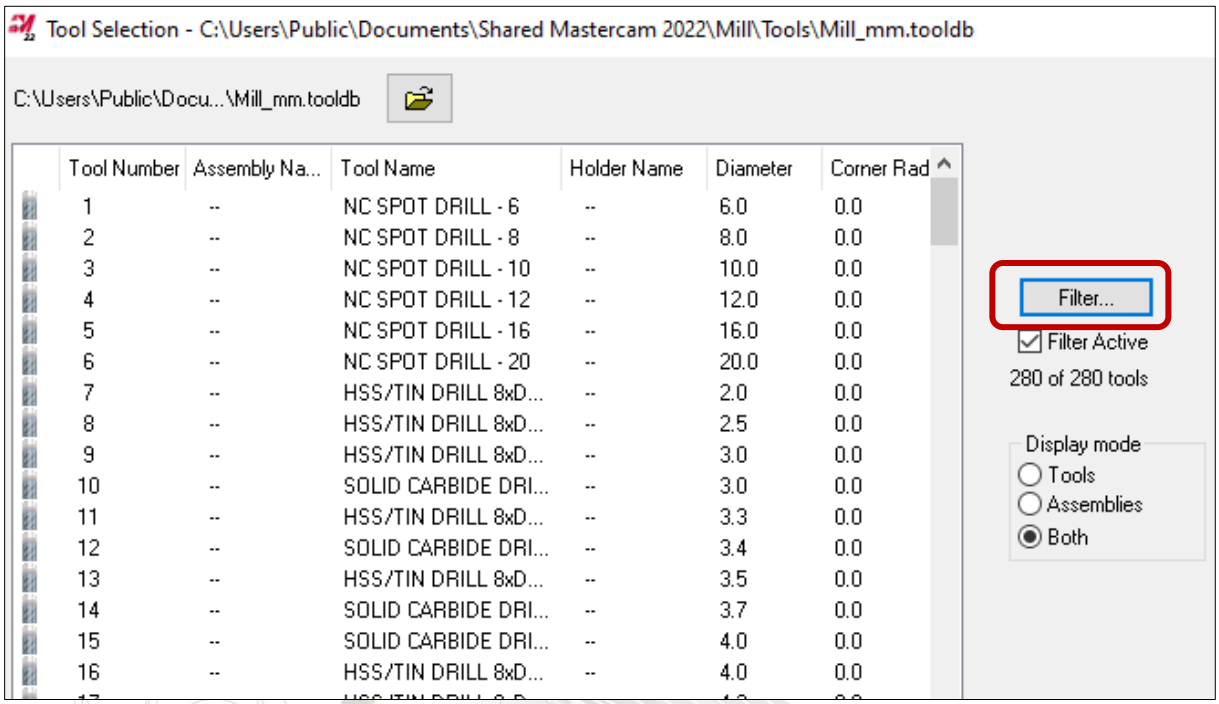
Then clik the Ok button. You will see the 2D Toolpath setting box.



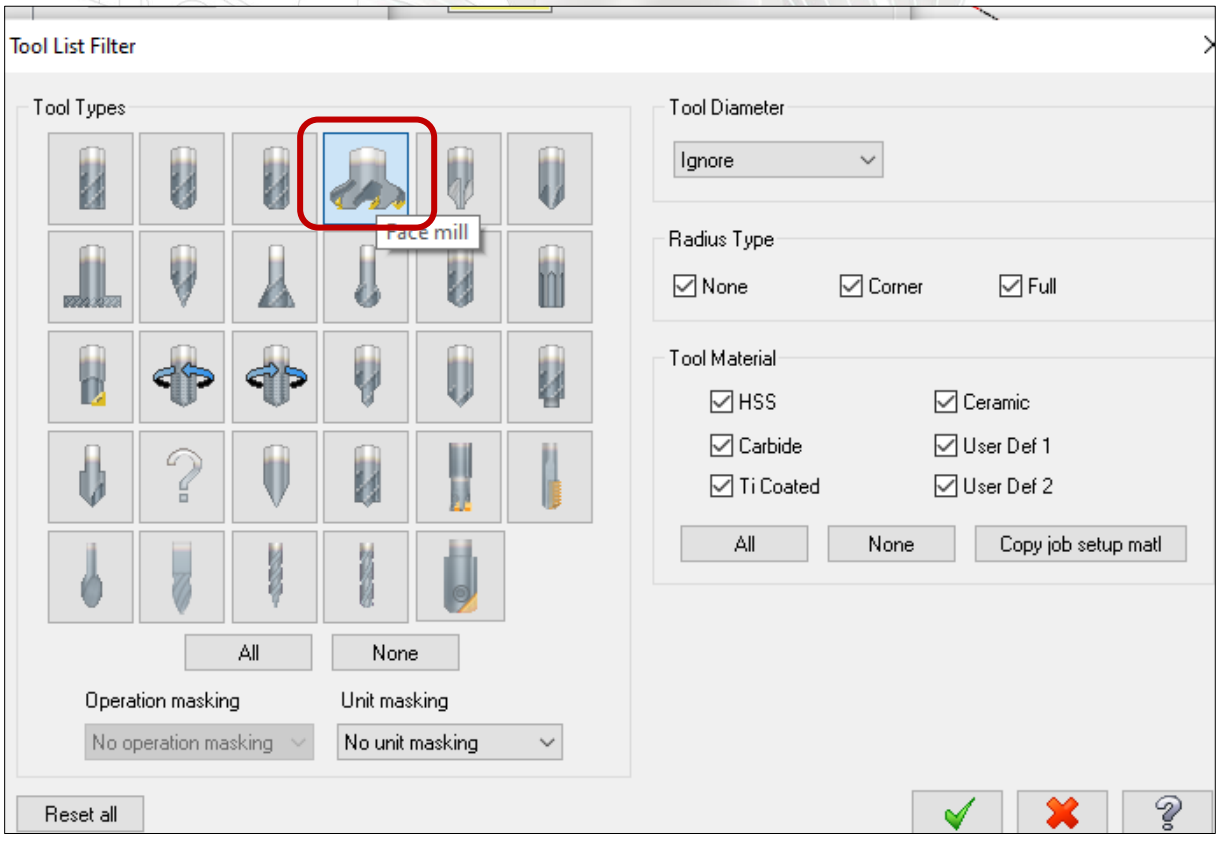
Click the tool list. You will see the box as shown.



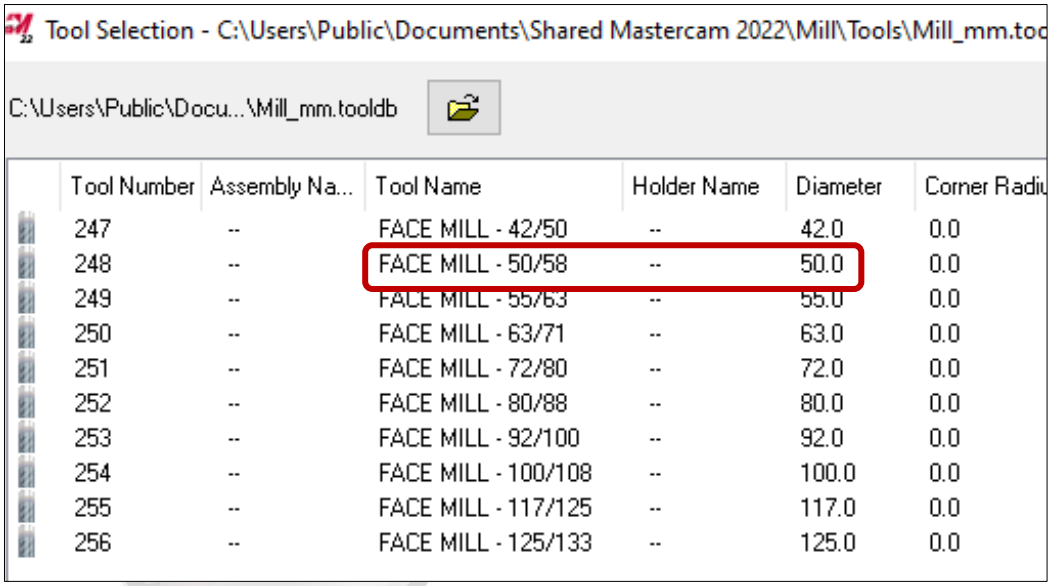
Go to select library tool. You will see the list of all type of tools. As you can see the figure, it show you 280 tools appear on the list. It will difficult you to choose the right tool. So, you must filter it by clicking the filter option.



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.

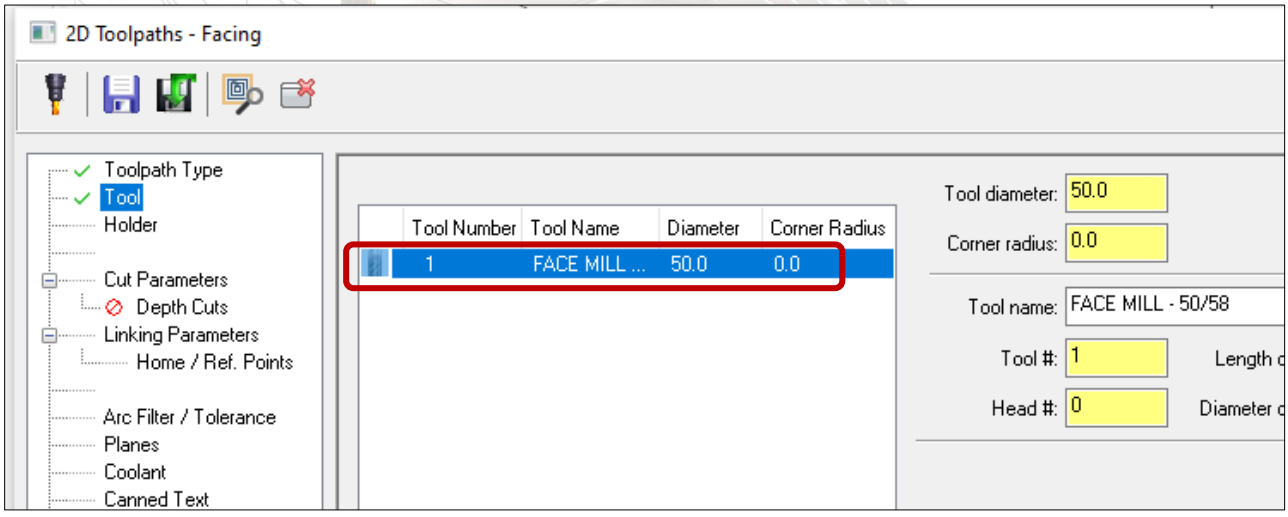


The update tools list will shown you the face mill tool only.

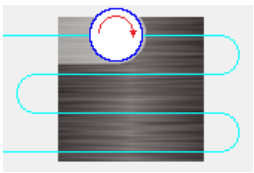


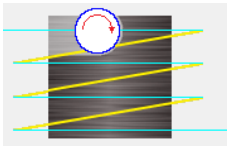
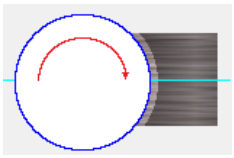
Tool Number	Assembly Na...	Tool Name	Holder Name	Diameter	Corner Radiu
247	--	FACE MILL - 42/50	--	42.0	0.0
248	--	FACE MILL - 50/58	--	50.0	0.0
249	--	FACE MILL - 55/63	--	55.0	0.0
250	--	FACE MILL - 63/71	--	63.0	0.0
251	--	FACE MILL - 72/80	--	72.0	0.0
252	--	FACE MILL - 80/88	--	80.0	0.0
253	--	FACE MILL - 92/100	--	92.0	0.0
254	--	FACE MILL - 100/108	--	100.0	0.0
255	--	FACE MILL - 117/125	--	117.0	0.0
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.

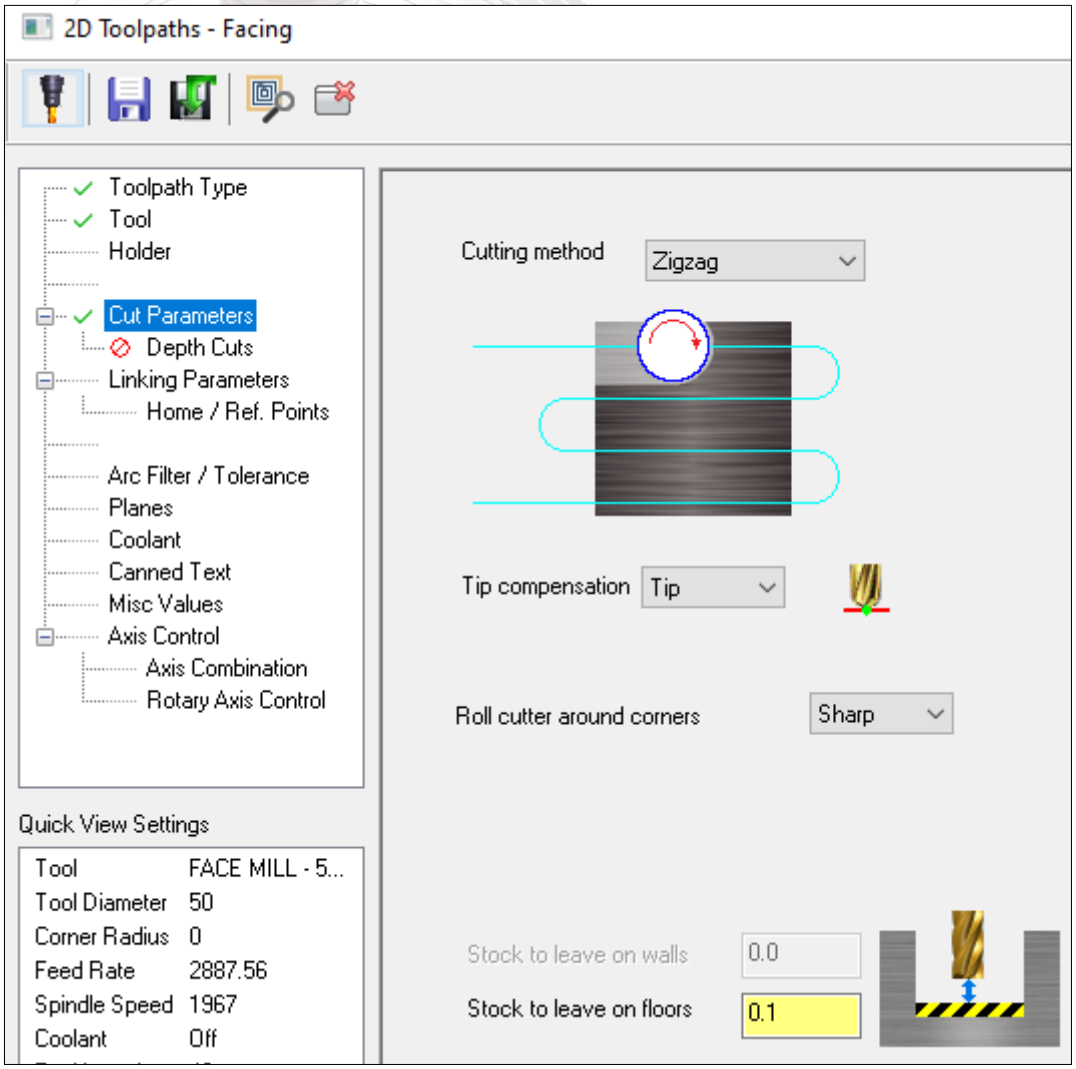


The box will appear as shown above. Clik the Cut Paremeters and choose the cutting method. There are 4 type of cutting method you can choose accordingly. The functon of theese cutting method are;

- Zigzag – back and forth cutting method 

- One way – one way cutting only. The cutting tool will step up when go back. 
- One pass – one step cutting only (suitable for diameter tool bigger than stock). 
- Dynamic – Circular motion.

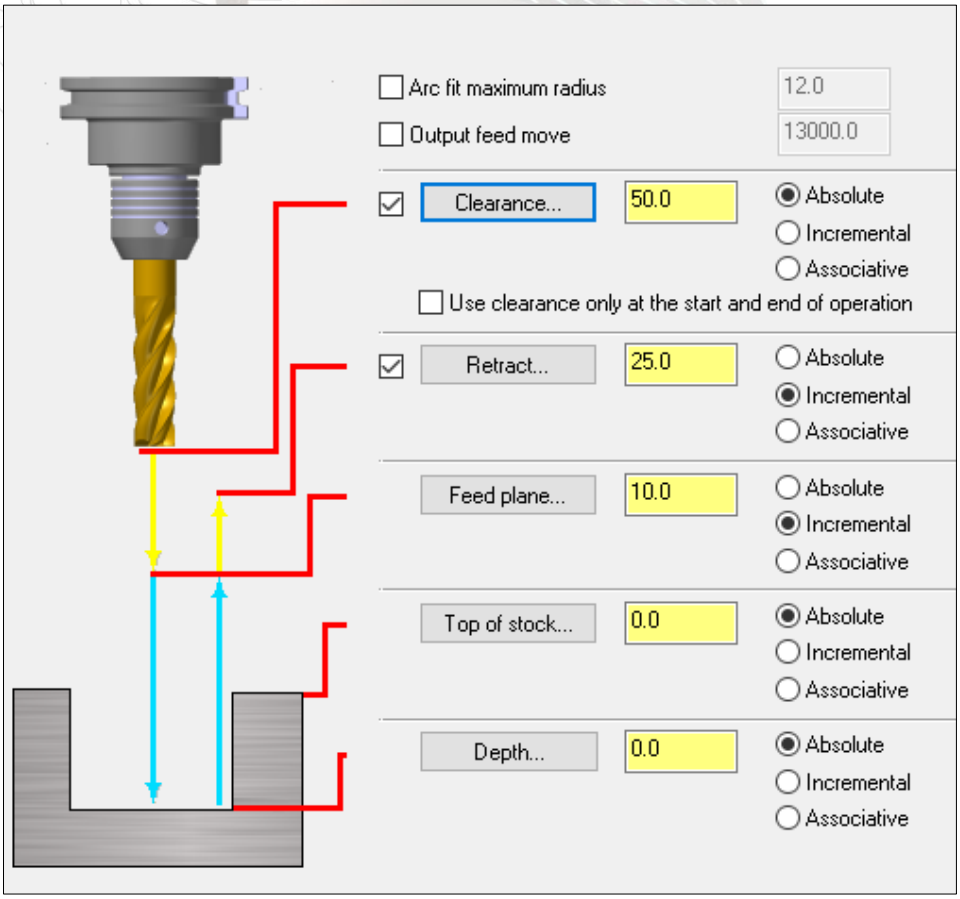
In this lesson, we will use Zigzag cutting method.



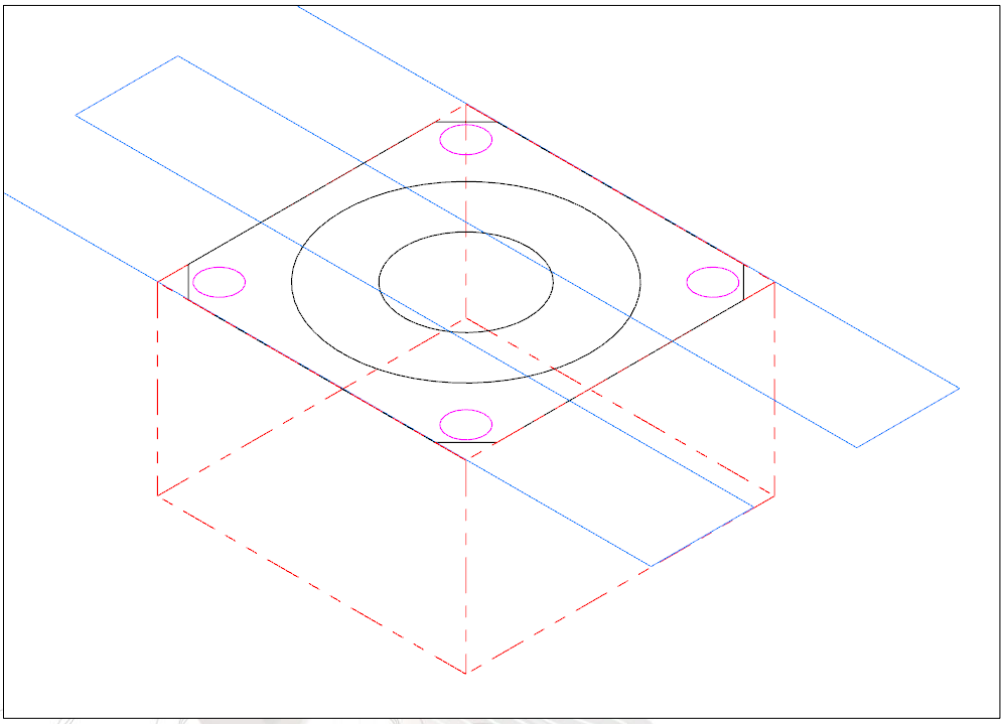
Leave stock on floor 0.1 mm for last process/finishing. Go to linking parameters and use the software setting. This step is very important to make sure a large effect on the final toolpath. Here the cutting depth and repostioning height a set. There are five major setting for this section;

- ✓ 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

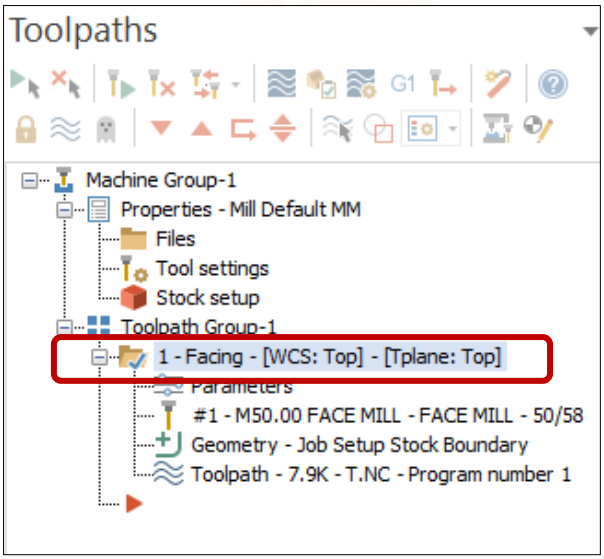
We can use either absolute or incremental coordination systems. Absolute coordination for the value stated is relative to the origin. While incremental coordination re relative to other parameters or chained geometry.



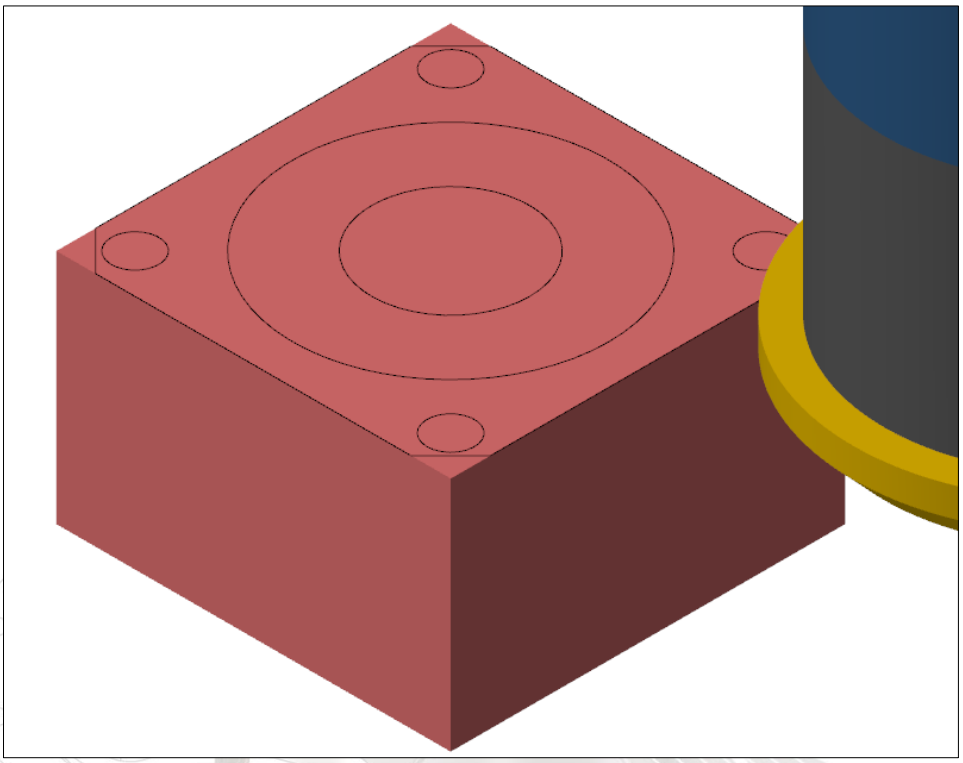
Set both top of stock and depth as 0 mm. Now click ok. The result will shown as figure



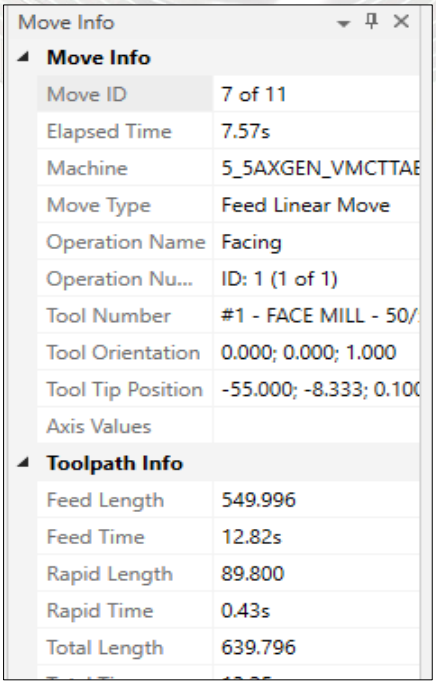
The blue trails/line mean the feed/mill process using Zigzag method. We can use backplot operation to see details movement for cutting process. Software will animate the movement of cutter through blue trails. From this function, we can detect error in the program. Before we use the backplot operation, 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 .

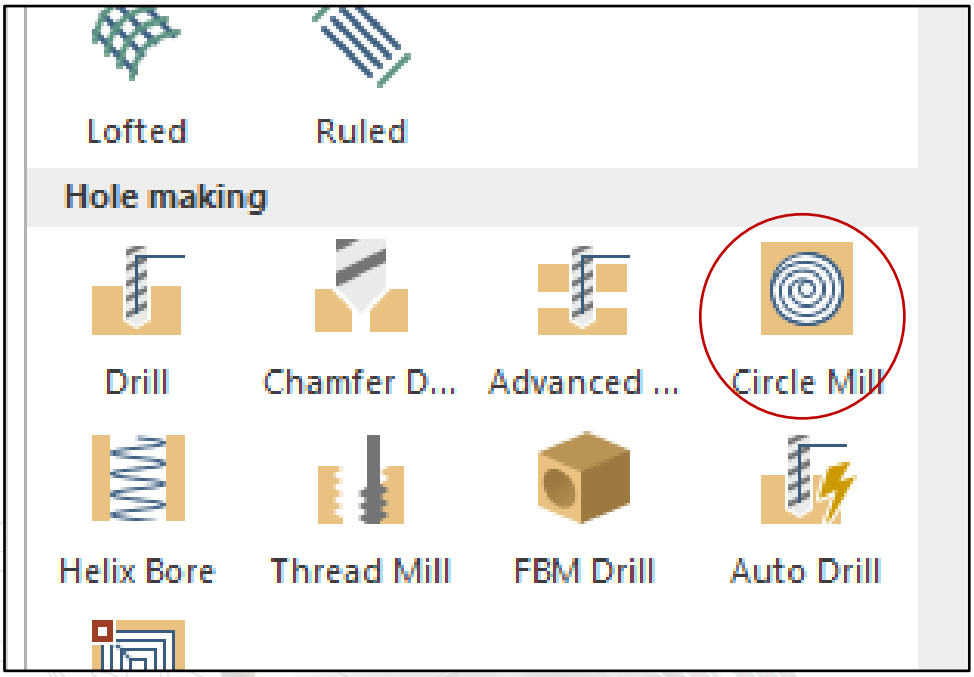


This function will simulate in three dimension (3D) movement /animation. At the left side of simulation screen will shown machining process info.

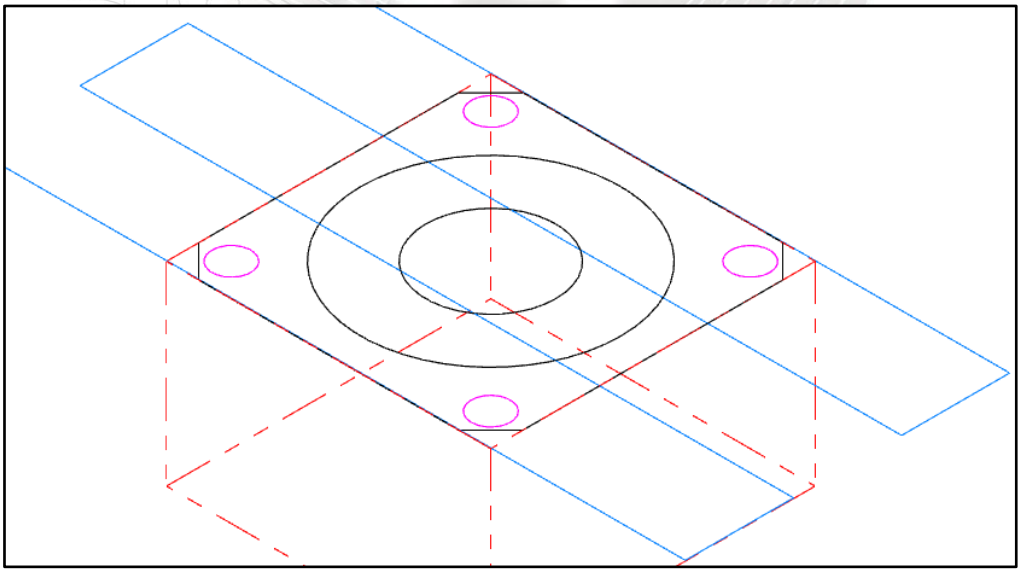


3.4 Cirlce Mill and Hole

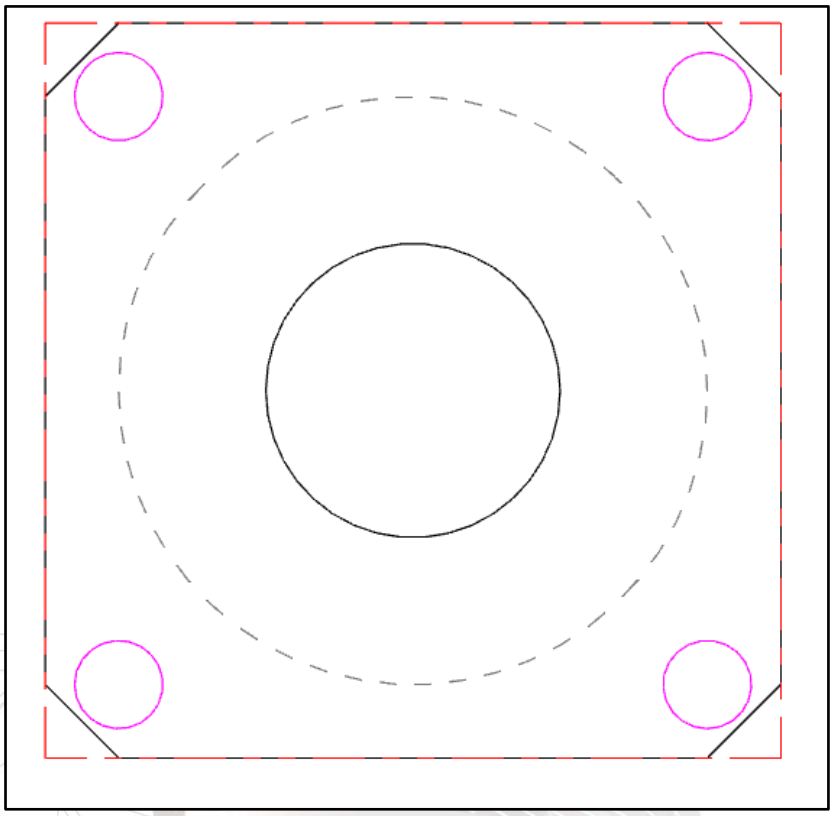
This section you will learn how to make both two circles mills and holes. At the toolpath tab, go to 2D option and choose Cirlce Mill.

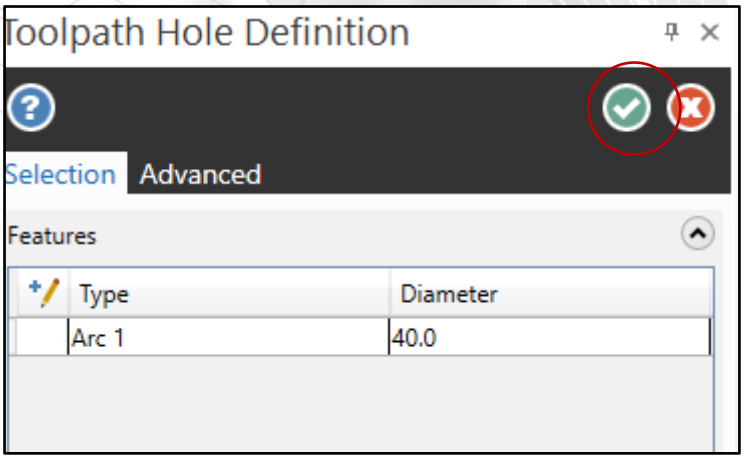


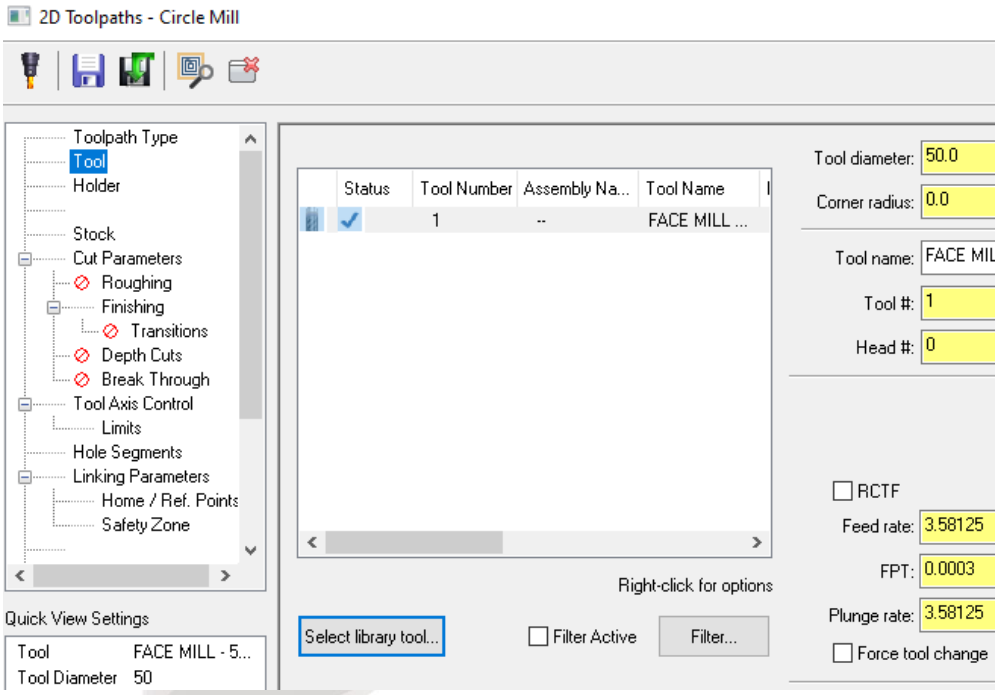
You will see a toolpath display. Press Alt + T to to hide it. To unhide, press it again.



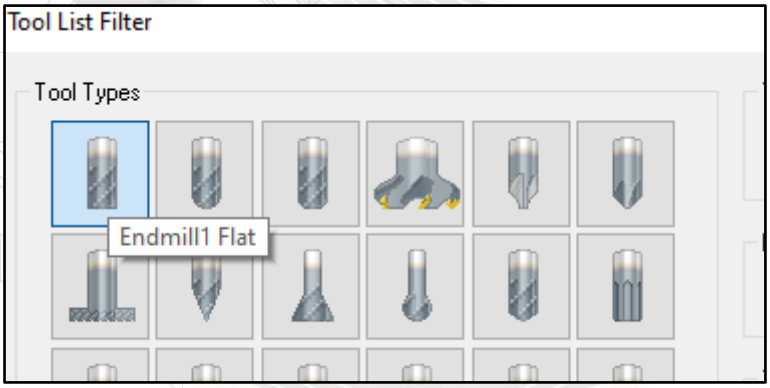
In top view, click the big circle first. Then click ok.



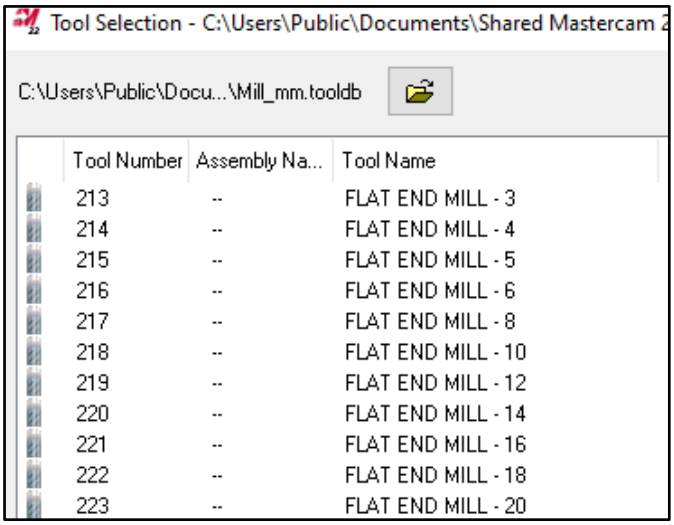




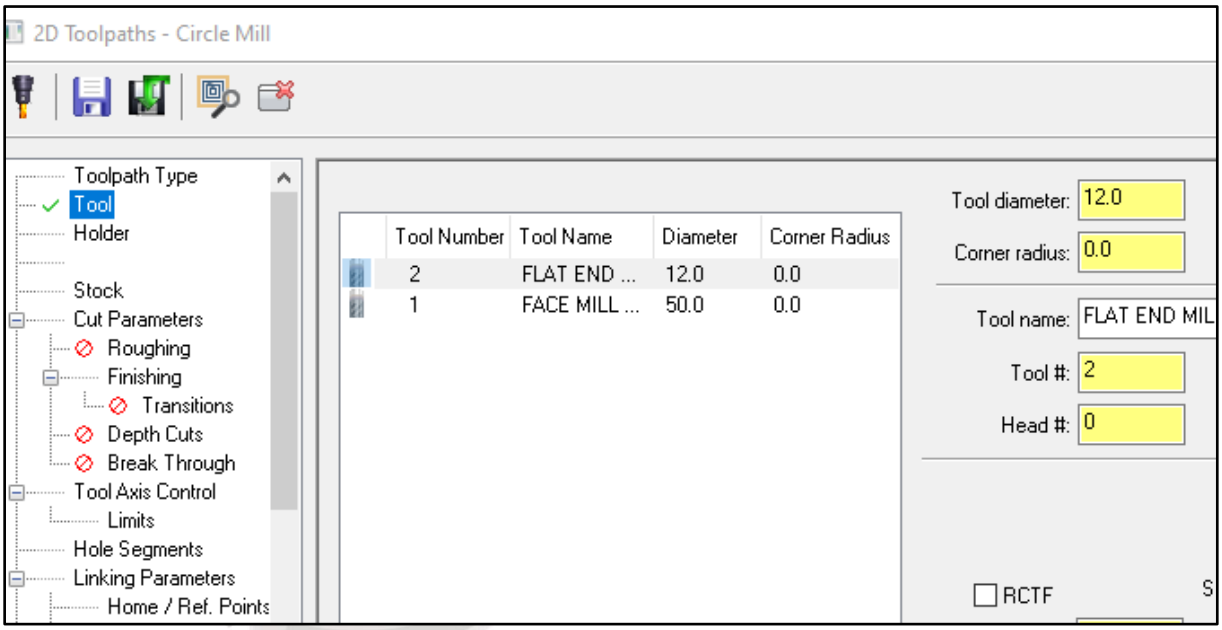
Select library tool and filter Endmill1 flat,



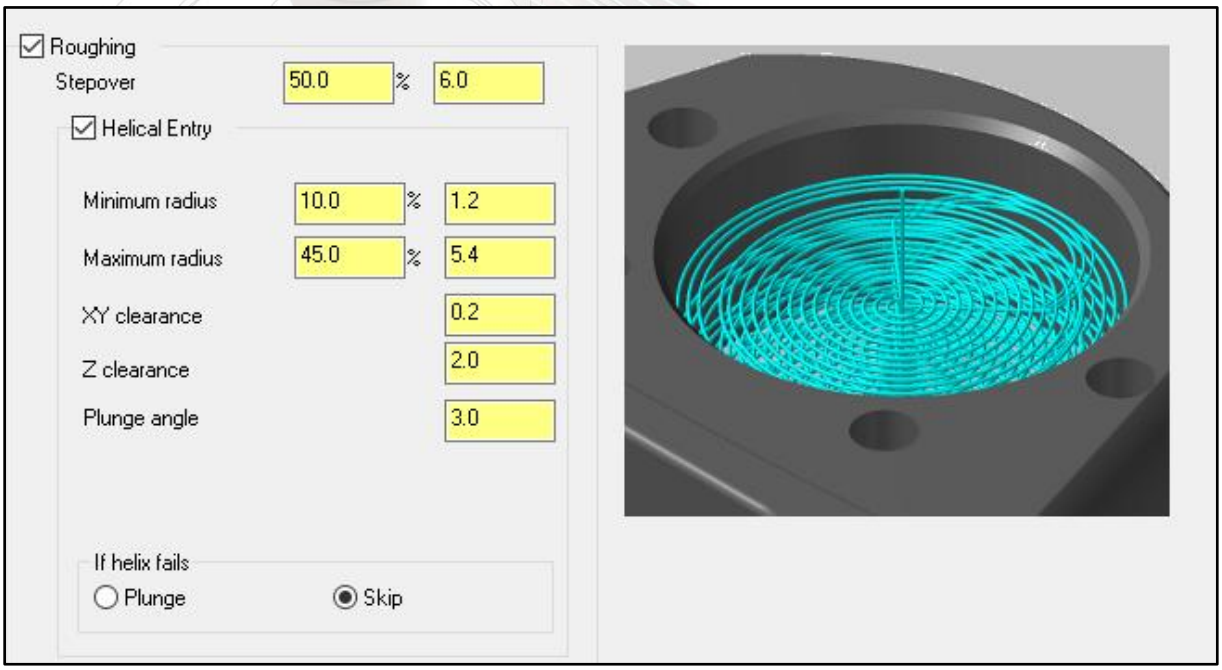
Select tool size small than 20mm. We will use same tool size for another 20 mm cricle diameter. No, we choose 12mm diameter tool size.



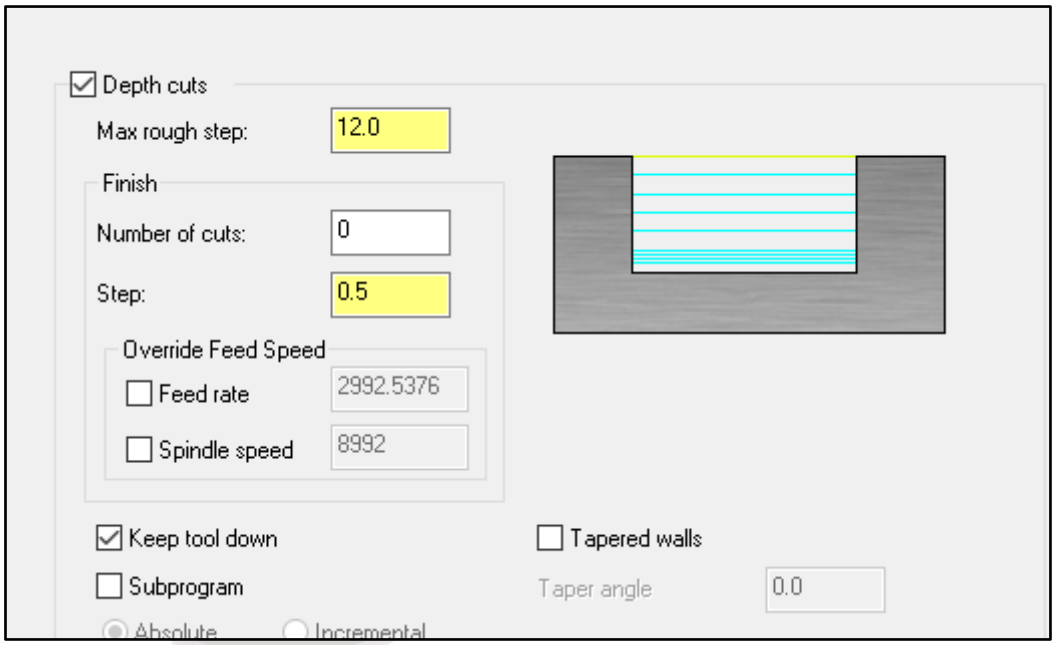
The choosen tool will appear at screen.



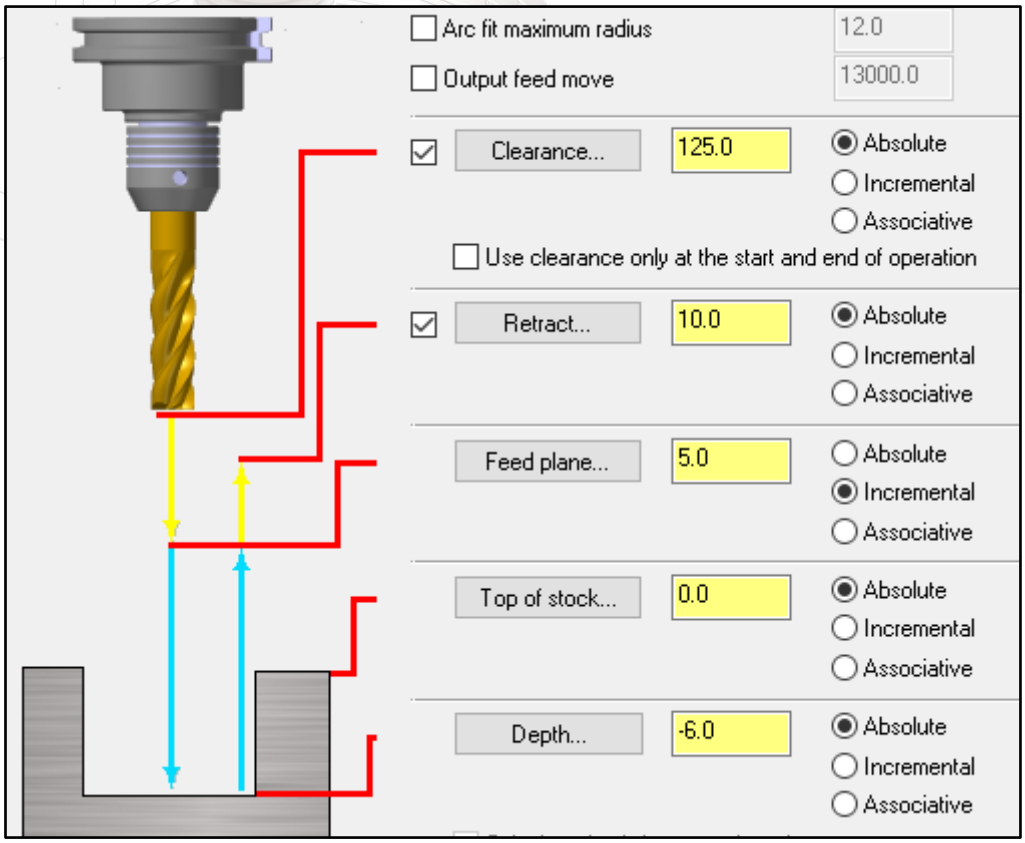
Go to Cut Parameter tab and set the value stock to leave on walls and floor at 0.1 mm. Go to roughing tab and activate it.



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

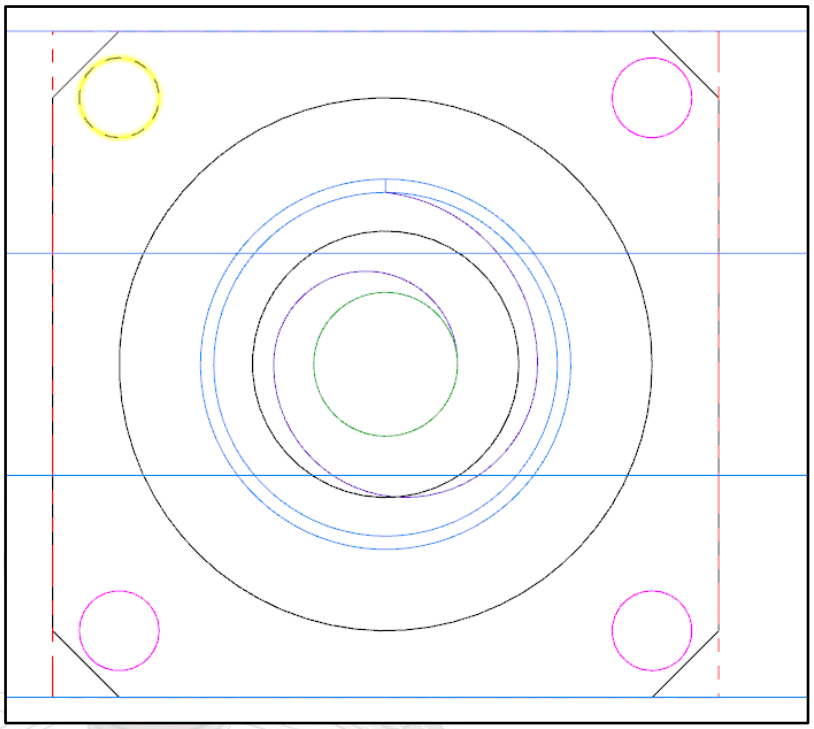


Go to linking parameter and set value as figure shown.

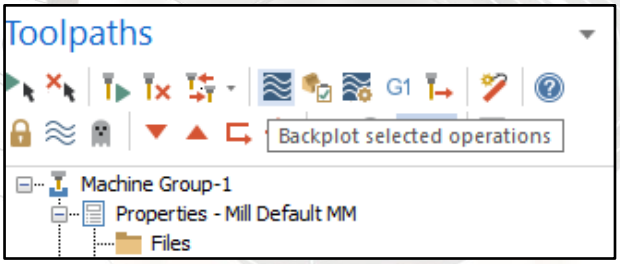


For begineer, we will use absolute coordinate. Put the depth value at -6.0 mm and click ok.

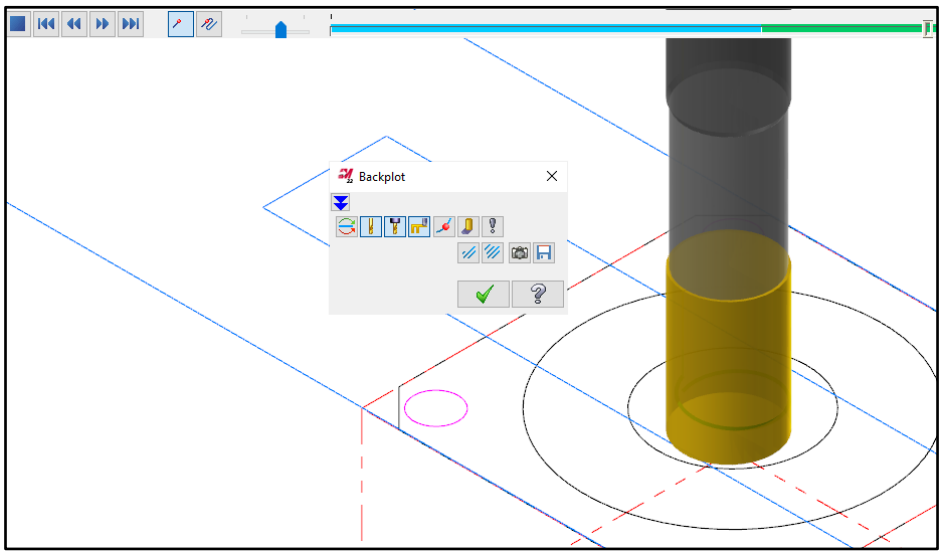
You will show the cutting toolpath as shown.



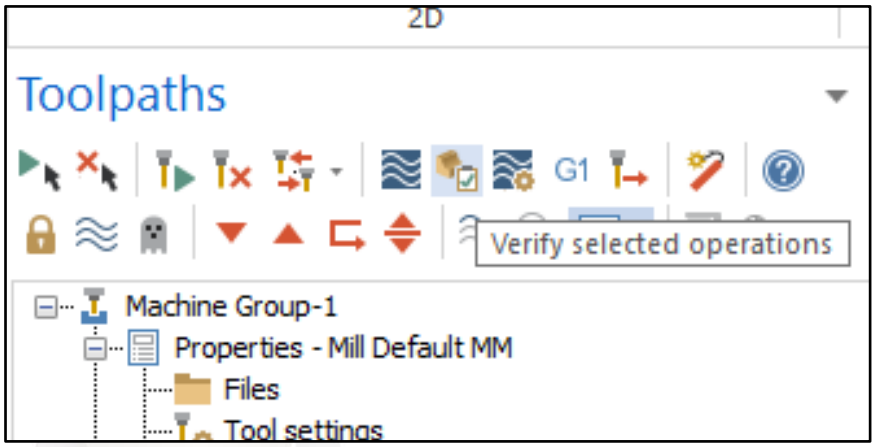
Next step is to backplot the toolpath. This backplot shows the path of tools. This function will let you spot errors in the program before machining process. Go to backplot icon.



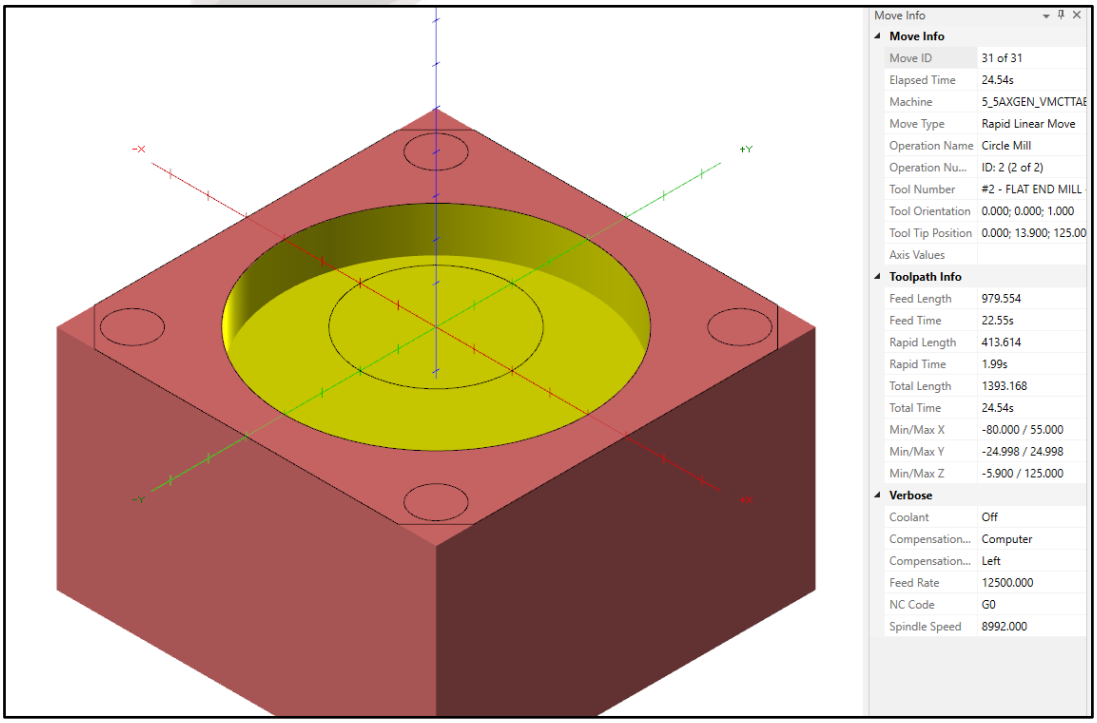
Play the backplot operation.



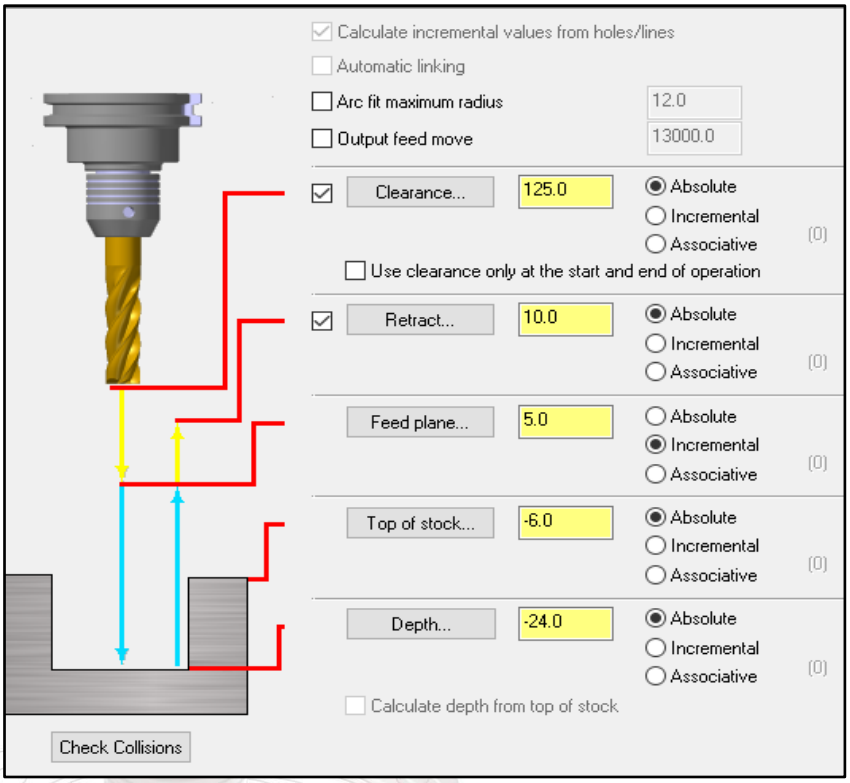
Final step for this circle mill process, simulate the toolpath in verify. This function is to shows the path of the tools to cut the stock or material.



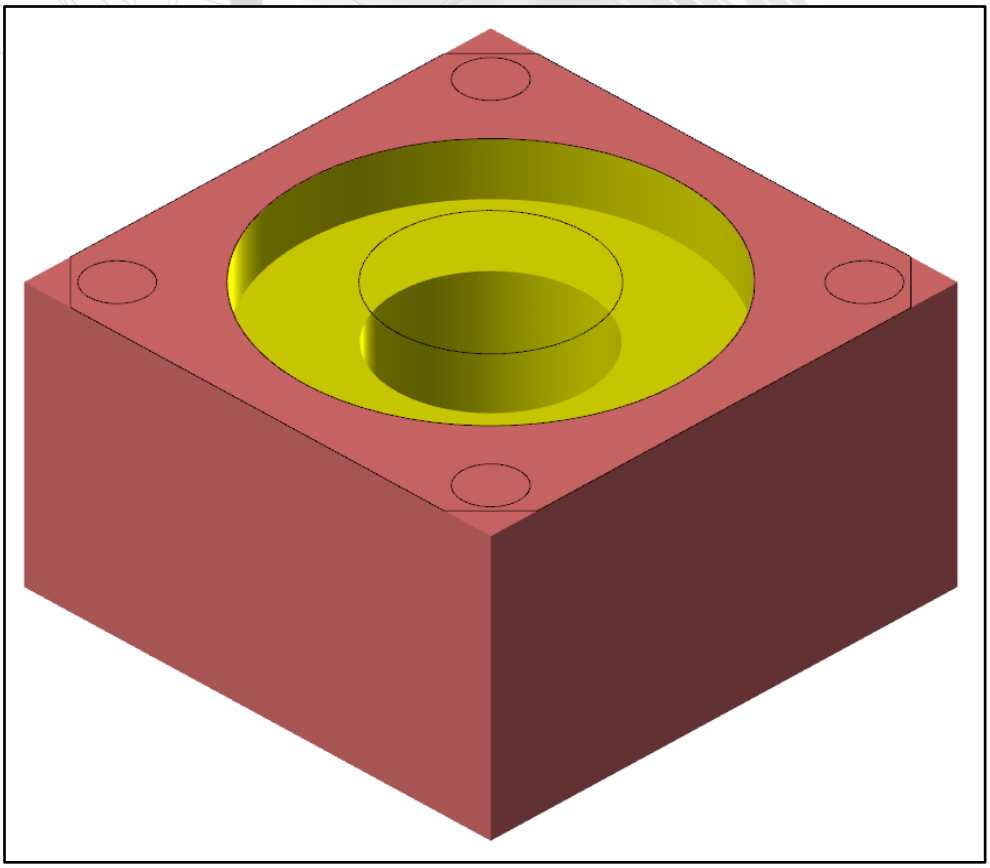
In verify mood, you will see a detail information about milling process such as feed lenght, feed time and total time.



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.

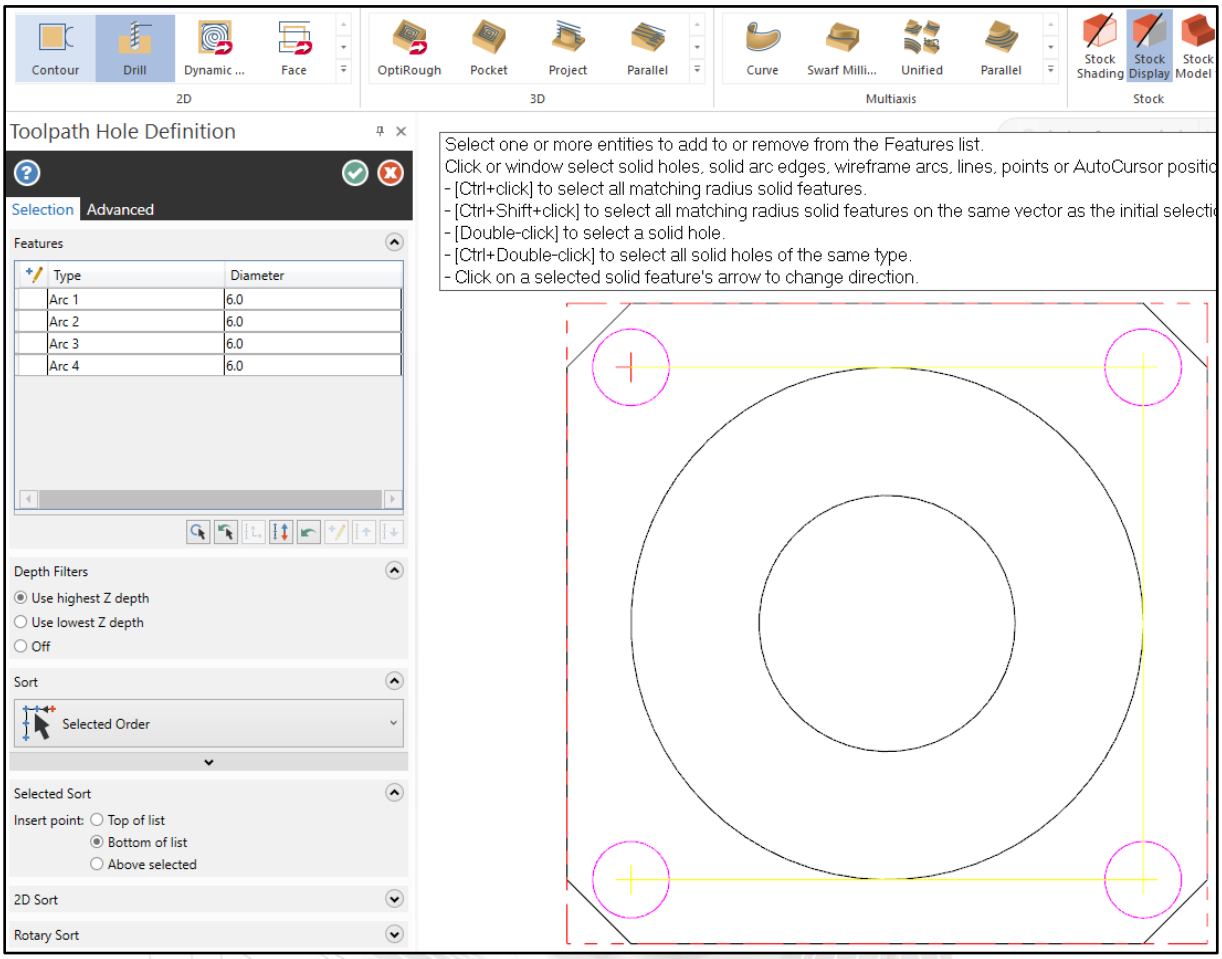


When the previous depth is -6.0mm, so the new top of stock is also same (original coordinate). Thats mean, the feeding process will start at absolute coordinate z: -6.0mm. The new depth z is -24mm.



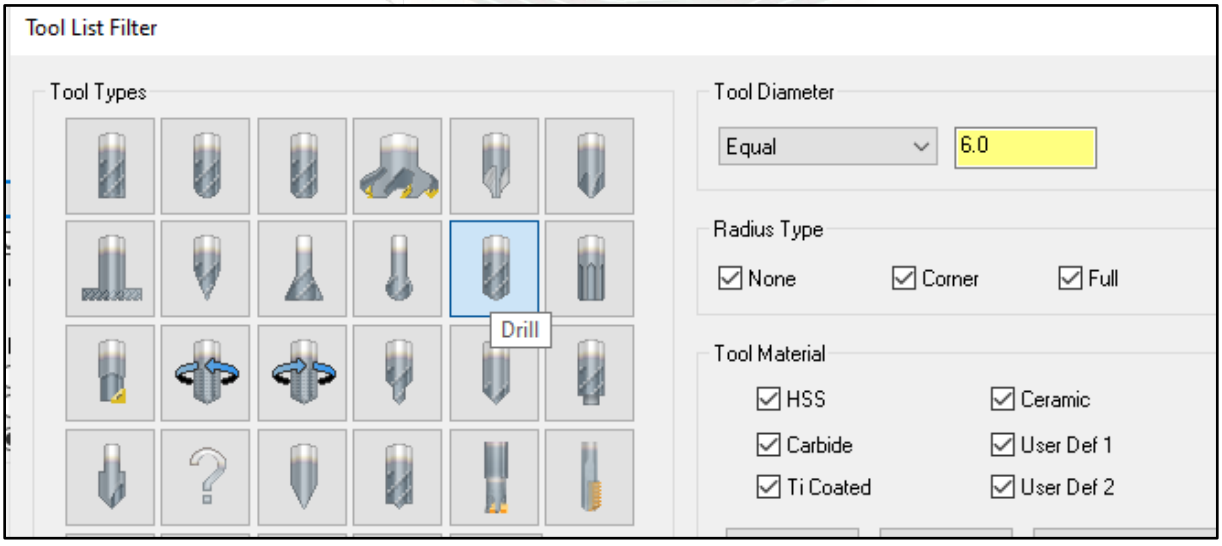
In this section, drill operation is needed. We will use spot drill for 6mm holes.

Clik drill.

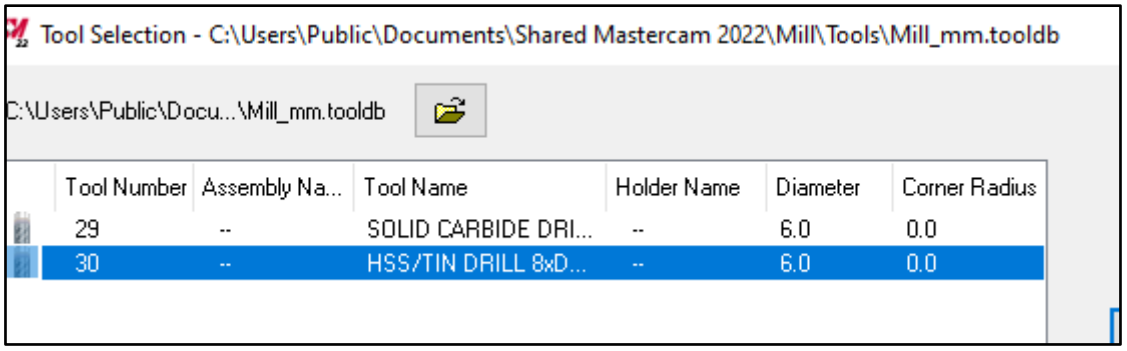


Click the center of 4 holes and 4 arc measurement will display at left display.

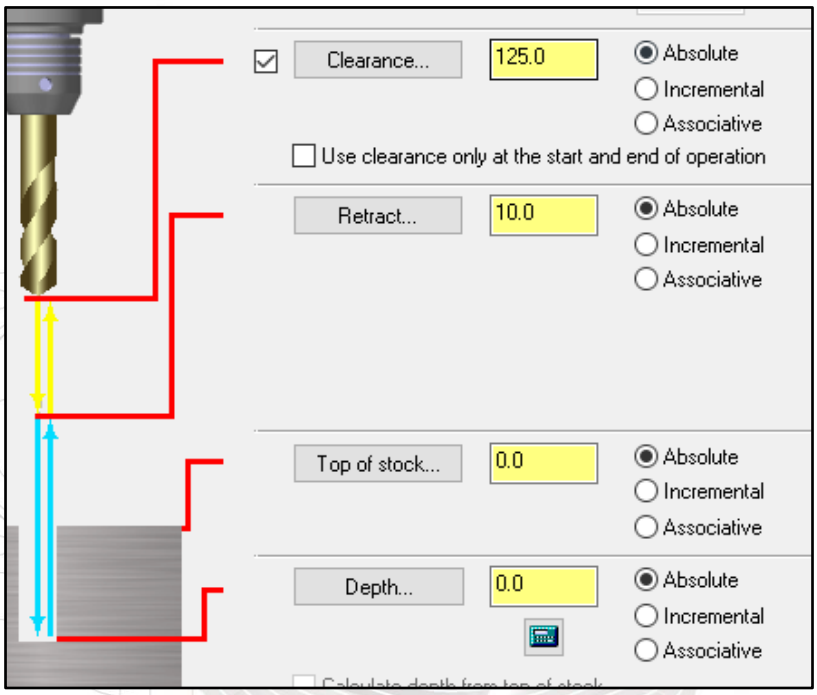
In toolpath setup, we will use new tool type, drill.



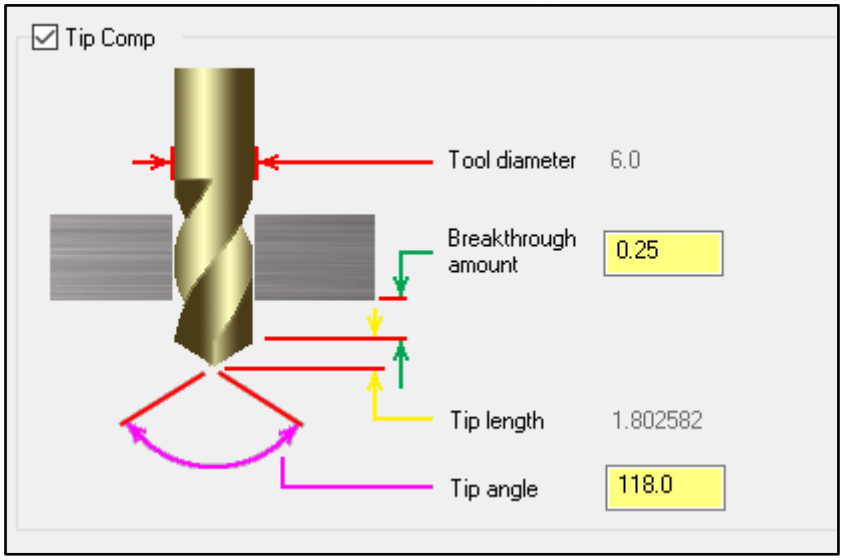
Change Tool Diameter to Equal 6.0. Selected desired tool drill.



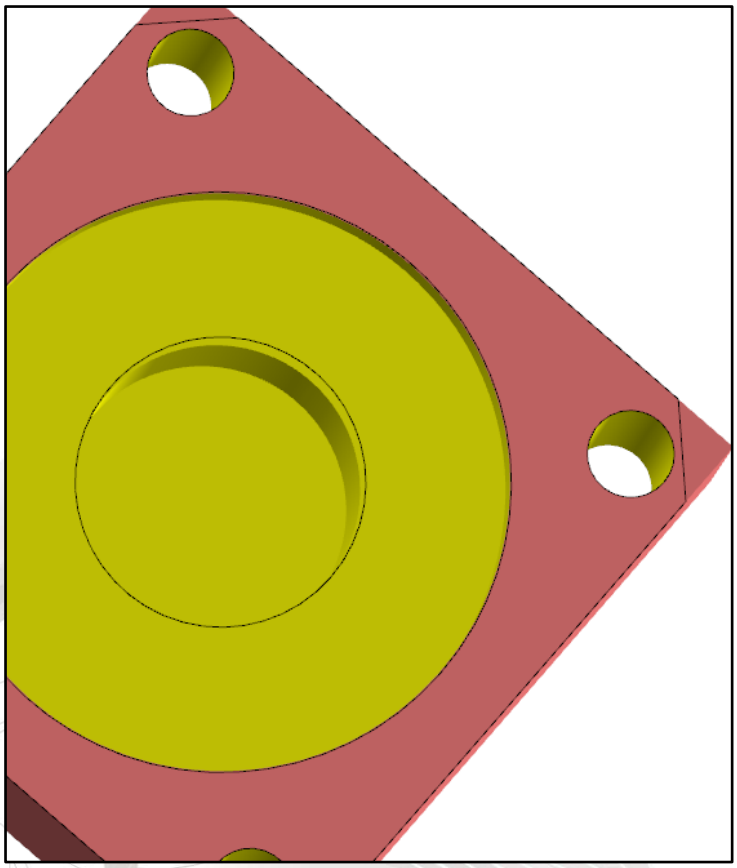
Choose Linking Parameter and enable the clearance. Click the calculator icon.



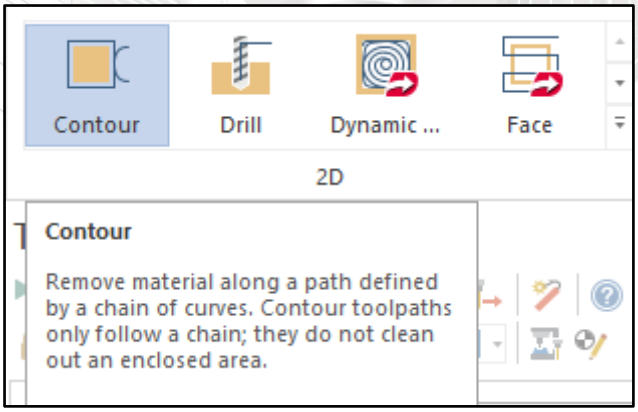
To generate 1.0 chamfer, key in -30 in depth box. Activate the tip comp for breakthrough.



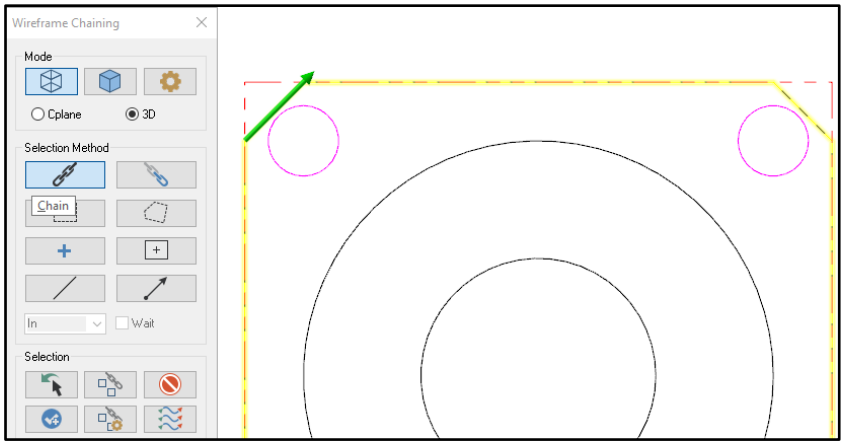
Simulate by verify the operation



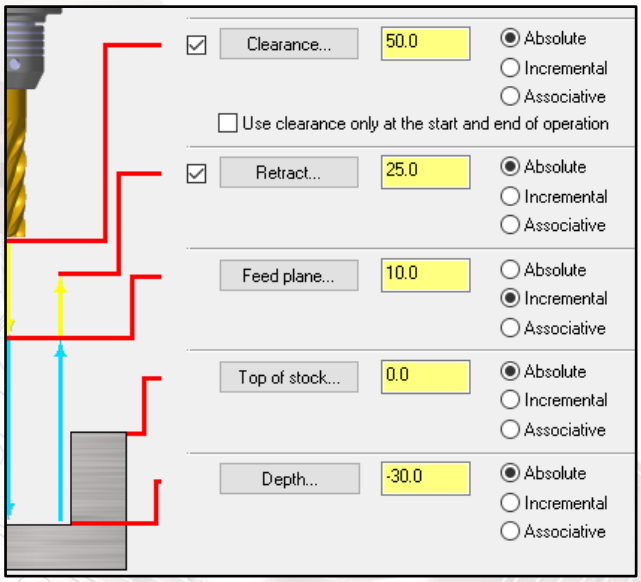
Last operation is to cut 4 corner of stock. We will use contour function to finished it.



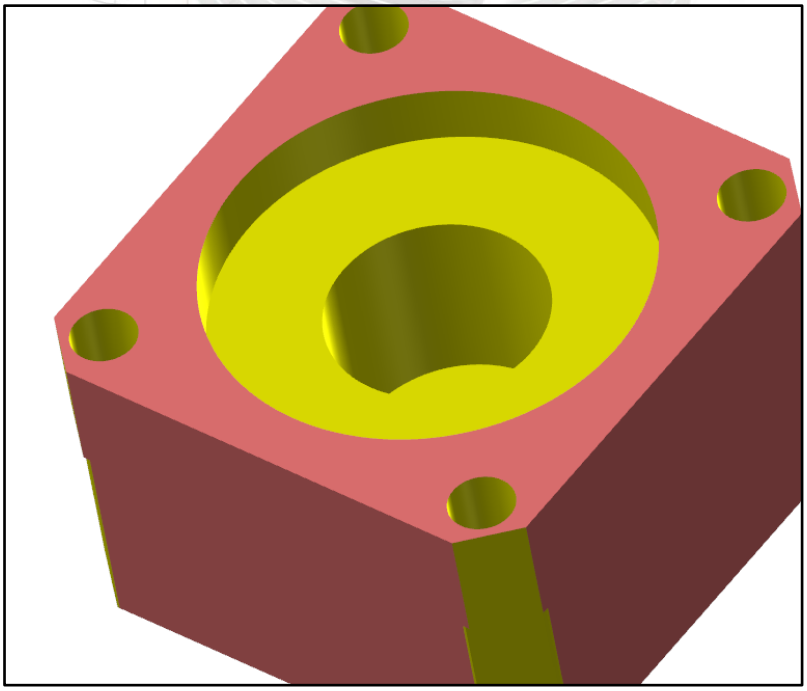
Use chain selected method and click outline of stock. You will see green arrow appear. You can change direction of arrow by click the reverse icon



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



The finished stock will be shown as figure.

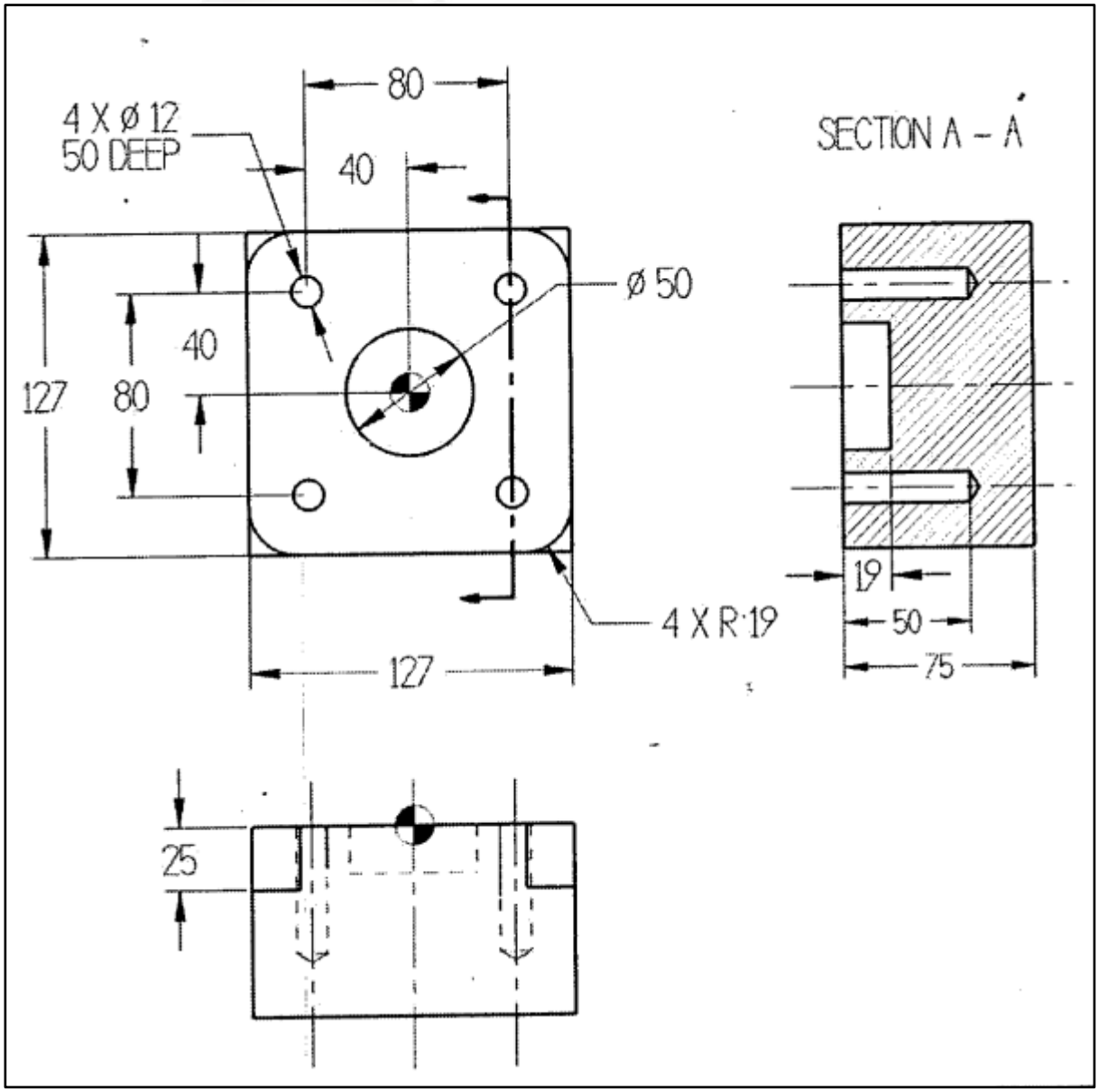


EXCERCISE

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
- ✓ For drill the holes, use a 20mm NC Spot Drill
- ✓ Set the cycle to Drill/counterbore and set Dwell of 1.0 second

SHAPING THE FUTURE
OF MANUFACTURING®
Mastercam 2022

ISSUED:

POLITEKNIK MALAYSIA
SULTAN SALAHUDDIN ABDUL AZIZ SHAH

Politeknik Sultan Salahuddin Abdul Aziz Shah
Persiaran Usahawan, Seksyen U1,
40150 Shah Alam,
Selangor Darul Ehsan
Tel : 03-51634000
https://psa.mypolycc.edu.my/

e ISBN 978-967-0032-24-5
9 7 8 9 6 7 0 0 3 2 2 4 5