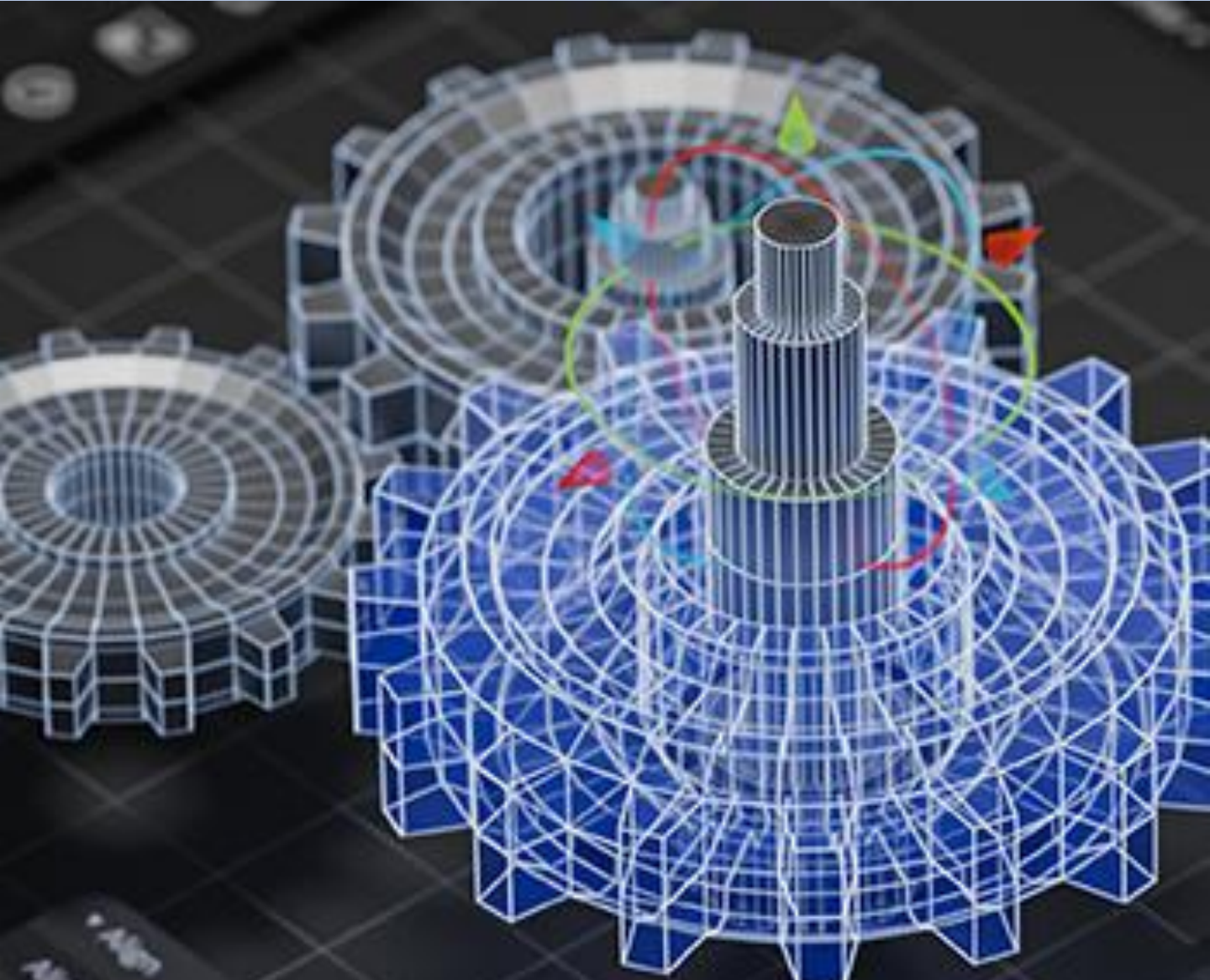




KEMENTERIAN PENGAJIAN TINGGI



CADCAM

BY MASTERCAM

2ND Edition

CAD CAM
by
Mastercam
2nd Edition

ACKNOWLEDGEMENT

PATRON

Mej (K) Dr. Ishak bin Mohamad
Director, Politeknik Port Dickson

ADVISORS

Abdul Rahim bin Ibrahim
Deputy Director (Academic), Politeknik Port Dickson
Khairun Syatirin bin Md Salleh
Head of Mechanical Engineering Department, Politeknik Port Dickson

EDITOR

(Noor Hapizah Binti Abdullah)
Programme, Politeknik Port Dickson

FACILITATORS

Che Azlina binti Che Norohoseni
Zuliana binti Zainal Abidin
Fasi Jab

WRITERS

Hasny Binti Abdul Jalil
Mohd Sallehuddin Bin Yazid
Mohd Hamdan Bin Abdul Razak

We would like to convey our utmost gratitude to the Department of Polytechnic and Community College Education particularly the E-learning and Instructional Division (BIPD) for funding our e-book project.

We hereby declare that this module is our original work. To the best of our knowledge it contains no materials previously written or published by another person. However, if there is any, due acknowledgement and credit are mentioned accordingly in the e-book.



Cataloguing-in-Publication Data

Perpustakaan Negara Malaysia

A catalogue record for this book is available
from the National Library of Malaysia

eISBN 978-967-2897-92-7

PUBLISHED BY:

Politeknik Port Dickson
KM14, Jalan Pantai, 71050 Si Rusa
Port Dickson, Negeri Sembilan

SEPTEMBER 2023

Copyright Each part of this publication may not be reproduced or distributed in any forms by any means or retrieval system without prior written permission.

ACKNOWLEDGEMENT

First of all, we are grateful to Almighty Allah S.W.T for establishing us to complete this book.

This book is written based on the latest syllabus contents of MasterCam for the topic Milling Process. This topic is relevant to students of Mechanical Engineering (Manufacturing) Program. Each topic is carefully written with a combination of notes, examples and tutorials that are suitable for teaching and learning sessions. Constructive criticism and suggestions for improvement of the book will be gratefully acknowledged. Finally, the authors would like to express their deep appreciation to everyone who is involved directly in the writing of this book.

TEAM OF WRITERS

1. **HASNY BINTI ABDUL JALIL**
(Politeknik Port Dickson)
2. **MOHD SALLEHUDDIN BIN YAZID**
(Politeknik Port Dickson)
3. **MOHD HAMDAN BIN ABDUL RAZAK**
(Politeknik Port Dickson)



ABSTRACT



In teaching and learning sessions, practical work also serves as an application to the theory presented in class. This study aims to identify the effectiveness by using the eBook Training Tutorial Milling and Turning Process while conducting practicals for Cad Cam at Department of Mechanical Engineering, Port Dickson Polytechnic. It simplified of procedures for the use of software. This eBook is a teaching aid that has been innovated from previous learning methods. It is used to carry out practical tasks to learn how to use CAD/CAM parameters in generating toolpaths and generating G and M codes from software programs. The concept is used to understand and provide proper planning to produce CNC machining work using MasterCam.



TABLE OF CONTENT

ACKNOWLEDGEMENT

TEAM OF WRITERS

ABSTRACT

1

**INTRODUCTION CAD
CAM**

COMPUTER AIDED
DESIGN (CAD)

01

COMPUTER AIDED
MANUFACTURING (CAM)

02

NUMERICAL CONTROL
PROGRAMMING

04

DIFFERENT BETWEEN NC,
CNC AND DNC

05

NC SYSTEMS
COMPONENTS

08

GEOMETRIC CONTROL

09

TABLE OF CONTENT

2

MILLING PROCESS

PRACTICAL MILLING TASK 1

16

PRACTICAL MILLING TASK 2

47

PRACTICAL MILLING TASK 3

77

PRACTICAL MILLING TASK 4

79

3

TURNING PROCESS

PRACTICAL TURNING TASK 1

80

PRACTICAL TURNING TASK 2

109

PRACTICAL TURNING TASK 3

111

PRACTICAL TURNING TASK 4

146

TABLE OF CONTENT

4

**QUESTIONS AND
ANSWER****147**



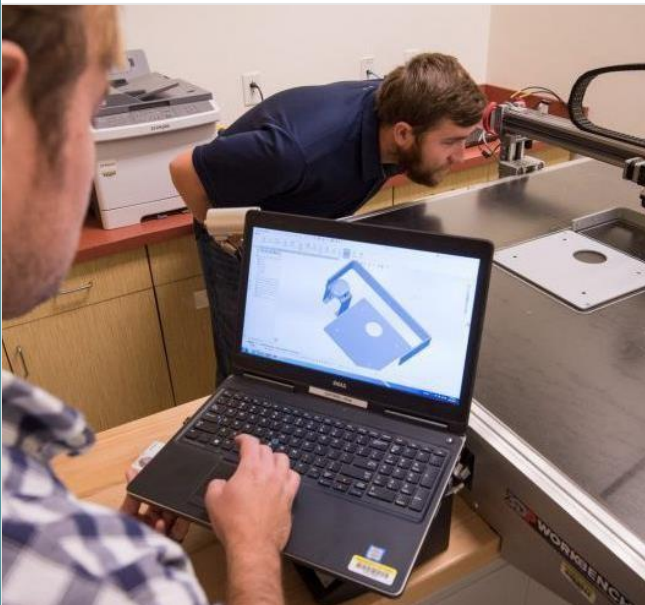
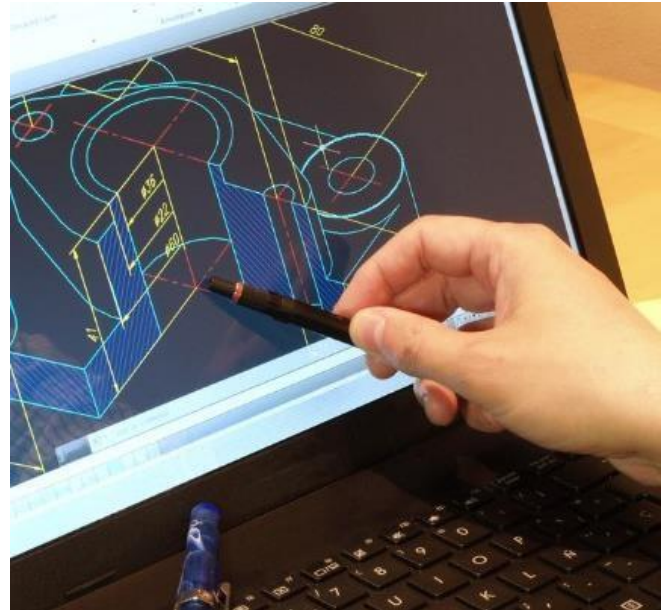
CHAPTER 1
INTRODUCTION
CAD CAM



COMPUTER AIDED DESIGN - CAD

CAD is the process of utilizing computers to create and edit design models and drawings. CAD Technology can provide :

- Faster than conventional methods.
- Easy to develop the model and associated drafting.
- Possible to manipulate various dimension, attributes and distance of drawing.



- Accurately calculate the geometric properties.
- Easy to modify a model.
- Use of standard components (part libraries).
- Provide 3D (three dimensional) visualization



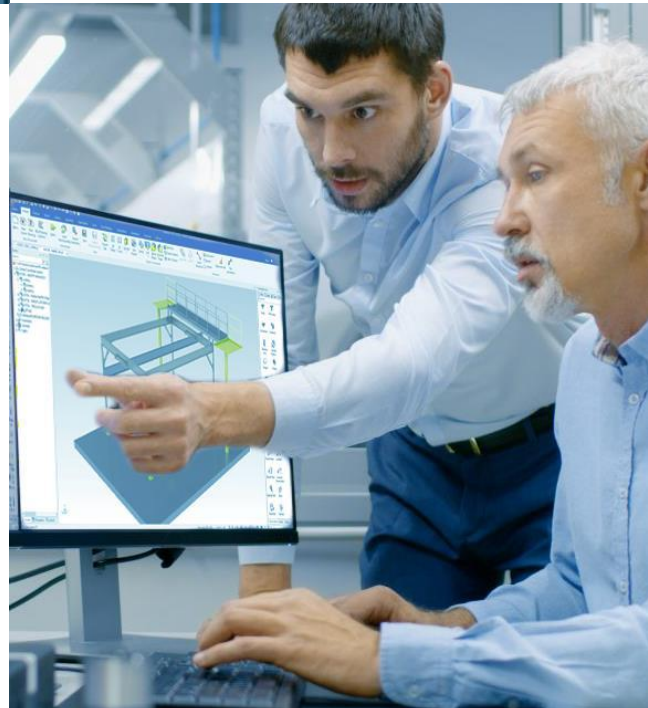
COMPUTER AIDED MANUFACTURING - CAM



CAM can be defined as the effective utilization of computers in direct manufacturing process control & monitoring or indirect manufacturing operation support.

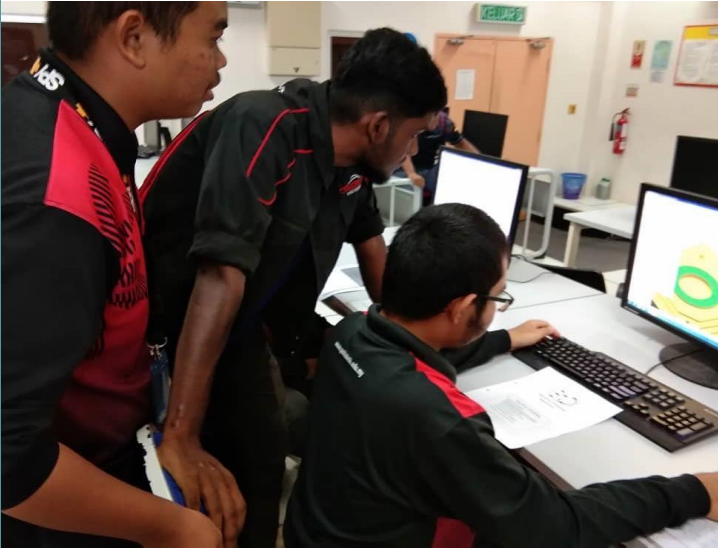
An integrated CAD/CAM system is a devoted system that will let the user to make or create a product geometry and generate CNC programs all in one package.

An integrated CAD/CAM system is a devoted system that will let the user to make or create a product geometry and generate CNC programs all in one package.



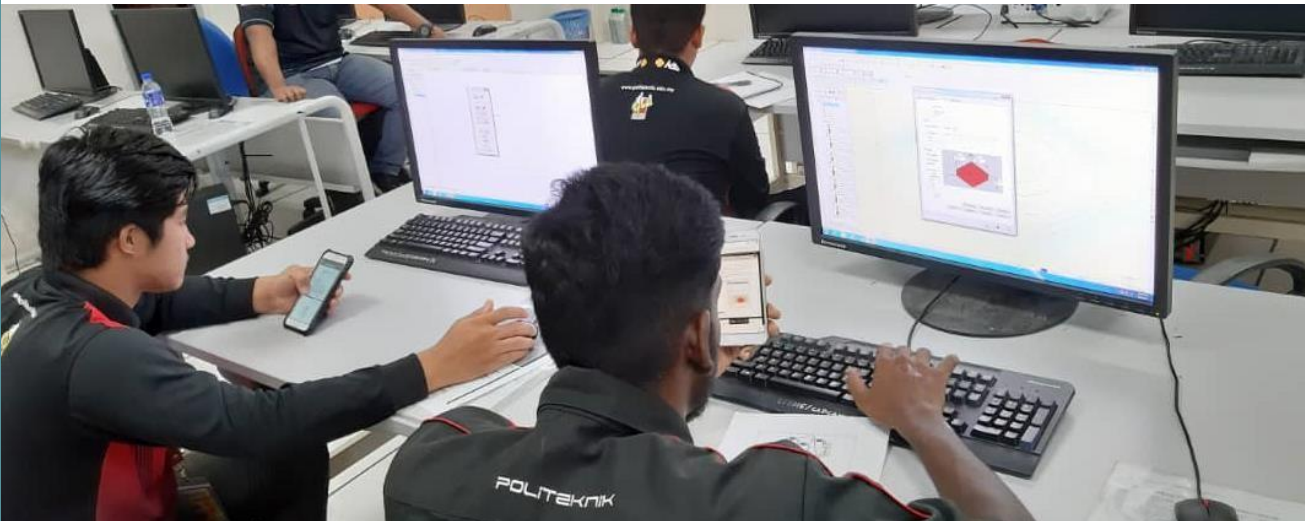


COMPUTER AIDED MANUFACTURING - CAM



CAM is capable to describe tool path in operations such as NC turning, milling and drilling. The programs is automatically determine and optimize the toolpath. It allows coding and classifying parts into groups that have similar shapes.

The emergence of CAD/CAM has a major impact on manufacturing by standardizing product development and reducing design effort, tryout and prototype work. This is resulting in significantly reduced costs and improved productivity.



Benefits of CAM

- Direct applications : device monitoring & control, NC, PLC, manufacturing cell.
- Indirect applications : manufacturing support-planning, MRP, process planning, scheduling, inventory, shop floor control.



NUMERICAL CONTROL PROGRAMMING

The NC is a Numerical Control, DNC stands for Distributed Numerical Control and CNC is the Computerized Numerical Control.

COMPUTERIZED NUMERICAL CONTROL

CNC machines is the function and motion of the machine tools. It is used to prepare program containing coded alphanumeric data.

The program will included a position of coordinates X,Y and Z axis and motion by cutting tools or work piece. The programmer or machine operator can change the program on the controller.

The CNC programs and the logical functions can write by manually or generate the by using the CAM and Cad system.

CNC is useful for control the motions of the work piece or tools and the input parameters (feed rate, depth of cut, etc).

DISTRIBUTED NUMERICAL CONTROL

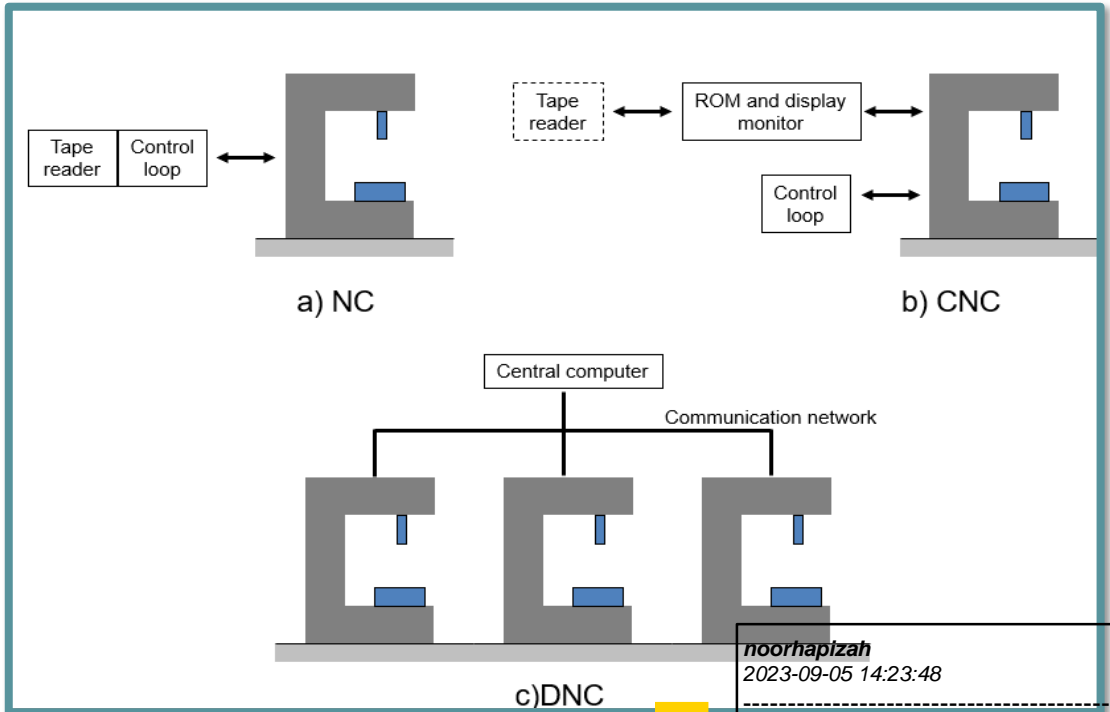
The control unit does not contain distributed numerical control. When CAM programmed are performed in the CNC machine control unit, DNC networking is necessary. The equipment required are RS232 cable and software. The input/output is used to send and receive data such like a port RS232 on CNC machine.

NUMERICAL CONTROL

Numerical Control (NC) are built-in for the control unit. It is permanently wire in the system and used a fixed logical functions. This system not allow any modification in the program. But it is still can interpret a part of program. For input information, punched tapes or punch card are requires for compulsory



DIFFERENT BETWEEN NC, CNC AND DNC



noorhapizah
2023-09-05 14:23:48

Space utk tajuk c) DNC



CNC machines



Numerical Control (NC) is a defined form of programmable automation. This is the mechanical actions of a machine tool or other equipment which are handled by a program (through punched tape) containing coded alphanumeric data.

Part programming contains geometric data concerning the part and motion information to move the cutting tool with respect to the work piece remove one.

Basically, the machine delivery an instructions and order as a sequence of blocks containing commands to set spindle speed, feed rate, machine parameters and other relevant information

The program address G identifies a preparatory command., often called the G code. It is pre set function associated with the movement of machine axes and geometry.

Motion group

G00 – Rapid positioning

G01 – Line interpolation

G02 – Circular interpolation - clockwise

G03 – Circular interpolation - anti-clockwise

Dwell

G04 – Dwell

Active plane selection group

G17 – XY plane selection

G18 – XZ plane selection

G19 – YZ plane selection



Cutter compensation group

G40 – Cutter compensation, cancel

G41 – Cutter radius compensation left

G42 – Cutter radius compensation right

Units Group

G70 – Inch units

G71 – Metric units

Hole making canned cycle group

G80 – Canned cycle cancel

G81-89 – Canned cycle on

Co-ordinate system group

G90 – Absolute co-ordinate system

G91 – Incremental co-ordinate system

Preset

G92 – Absolute pre set, change the datum position





NC SYSTEM COMPONENTS

MACHINE CONTROL UNIT - MCU

MCU is a microcomputer that stores the program and executes it by converting each command into actions by the machines.

MCU consists of both hardware and software. Hardware includes the microcomputer, components to interface with the processing equipment and feedback control elements.

The software in MCU includes control system software, calculation algorithms, and translation software to convert the NC parts program into a usable format for the MCU. MCU also permits the part program to be edited in case the program contains errors or changes in cutting conditions.

PART PROGRAMMING

Part programming is a detail set of commands to be followed by the processing equipment. It specifies a position or motion in x, y and z coordinates by work piece or cutting tool.

Part program also includes spindle speed, spindle direction, feed rate, tool change etc. The part program is written manually or by using computer assisted language such as APT (Automated Programming Tool).

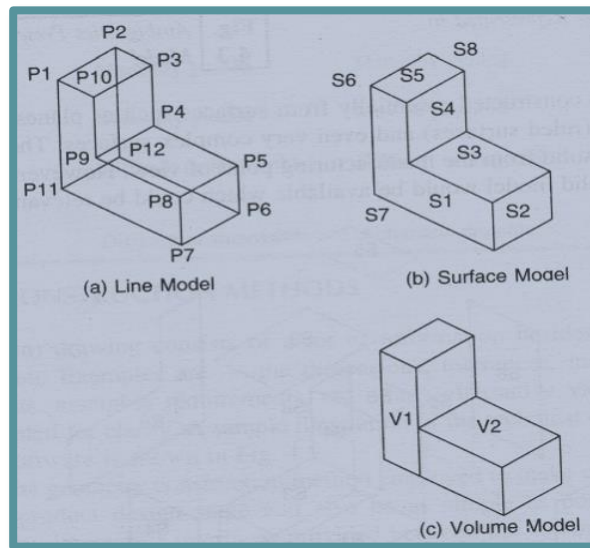
PROCESSING EQUIPMENT

The processing equipment is a machine tool could be one of the following: milling machine, turning machine, wire cut, laser, plasma, coordinate measuring machine etc. Machine tools accomplishes the sequence of processing steps to transform the starting workpart into a complete part. MCU gives an instructions from part program in order for machine to operate



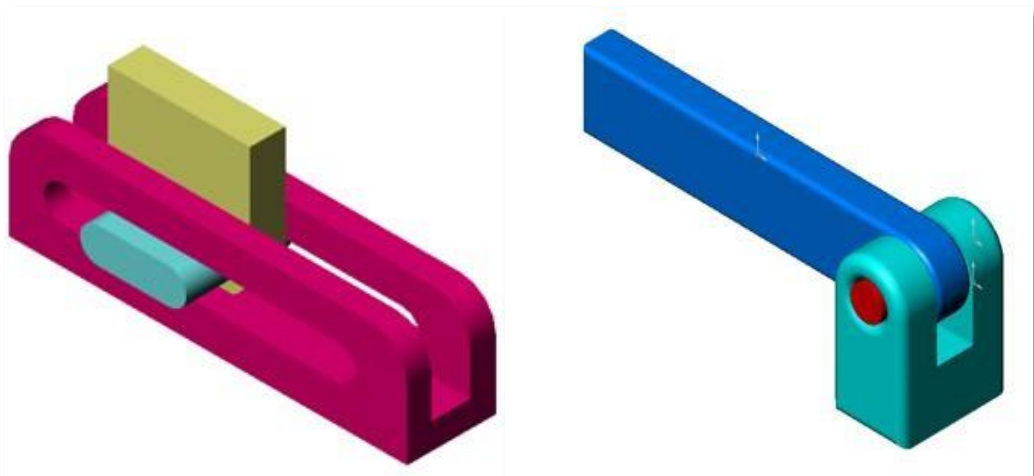
GEOMETRIC MODELLING-GM

A geometric modeling is a technical drawing that describes the shapes of the object. It can be built by using drafting software such like AutoCAD, Catia, SolidWorks or Mastercam. The Cad designer must be expert to create a 2D or 3D modeling by using features such like wireframe, solid, surface and curve.



Types of geometric modelling

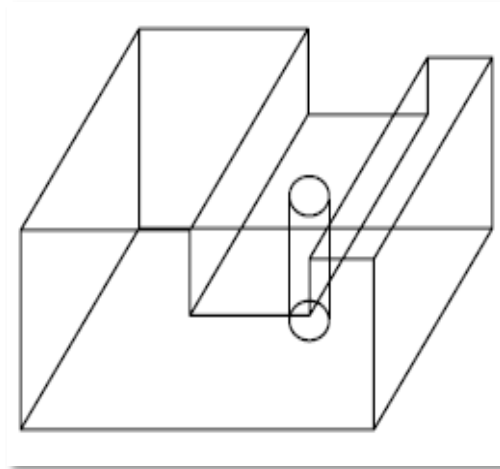
A geometric model should accurately depict the related object, be distinct from other models, and be comprehensive for all engineering tasks, including documentation, engineering analysis, and manufacturing. The purpose of a geometric model is to build solid models by starting with points, lines, and curves, extending the curves to build surfaces, and building solids from surfaces.



Solid models



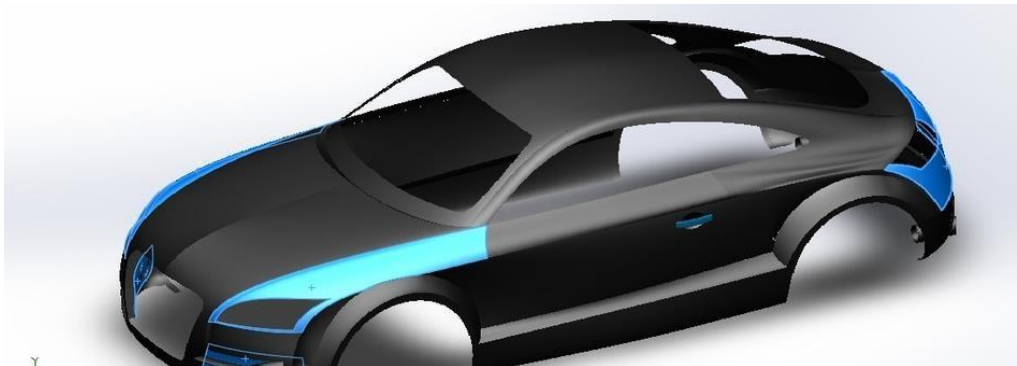
GM – WIREFRAME AND SURFACE



Wire frame

The **wire frame** represents a 3 dimension of the techniques used for draughting, in view of the simpler manipulation methods.

Projection is represented by a coordinate system, consisted of world coordinate system (WCS) and user coordinate system (UCS). Inadequate for representing more complex solids.



Surface creation

Surface creation usually starts from curves, might require two boundary curves and displayed as a mesh. CAD/CAM systems provide surface entities, which can be divided into:

- Analytic entities: ruled surface, plane surface, surface of revolution and tabulated cylinder.
- Synthetic entities: rectangular, bicubic Hermite spline surface, B-Spline surface and triangular Bezier patches, and triangular Coons patches and NUBS (nonuniform B-splines).



GM – SOLID MODELLING

Solid modelling is complete, valid and unambiguous representations of objects. It consists of both topological (combination structure) and geometrical data, complete description of the solid in a certain form for manufacturing.

Solid Modelling is a natural extension from the use of essentially 1D entities (curves) or 2D entities (surface) to the modeling of shape using 3D solids.. There are two approaches to create solid models:

1. Primitives:
2. Features:

1.0 PRIMITIVES

Primitives are point and straight line segment which are simple, basic shapes which can be combined by a mathematical set of Boolean operations to create the solid design. Allows designer to use predefined shapes (primitives) as building block to create complex solids.

Boolean methods is used to combine the primitives and limited by the restricted shapes of the primitives. Common primitives shape available: block, cylinder, cone, sphere, wedge and torus.

2.0 FEATURES

Features is defined as a shape and an operation to build parts. More flexible and let the construction of more complex and elaborate solids. Three steps which are involved to create a parts:

1. Create sketches
2. Create features
3. Use features to build parts

Major common features available in CAD system. Example:

Extruded : use to create solid models of 2.5D objects with uniform thickness.



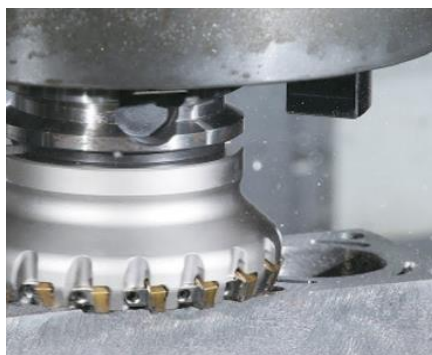
MILLING PROCESS



Rotation of cutting

A milling machine is the machine tool that typically handles this process. The milling process involves intermittent cutting; with each rotation, the milling cutter's teeth enter and depart the work.

A work piece is cut during a milling process by a revolving cylindrical tool with several cutting edges. The cutting tool's axis of rotation is parallel to the feed direction.



Cutting tools in milling process

Mastercam is an engineering software that is used widely in the manufacturing industry. This software provides CAD and CAM functions in one package. It is used to drive CNC machines for optimized productivity and efficiency.

Milling is a machining process which is typically used to produce parts by providing a rotating spindle for the cutter and has many features, such as holes for drilling or grooving, slots, pockets, and even three-dimensional surface contours. Parts that are fabricated completely through milling often include components that are used in limited quantities, perhaps for prototypes, such as custom-designed fasteners, mold or brackets.

There are many types of milling machines such as slab, slotting, straddle side and CNC milling. Another application of milling is the fabrication of tooling for other processes. Different machine tool designs are frequently utilized as an additional step to add or enhance characteristics to components that were made using a different technique. An item whose basic shape has already been produced is perfect for adding precise features since milling offers excellent tolerances and surface finishes.

Mastercam started as a 2D CAM system and then improved to 3D and solid modeling systems. The CAD tools let the designer operate making a design and part programmed for a CNC machine. This is used widely for the manufacturing industry in the world.



OVERVIEW of steps for making the final part of milling :

Drafting the Cad Model:

- The student will check the drawing or drafting to understand how the part is created in the tutorial.
- From the design of the drafting or drawing, the student can choose how to create the geometry by using Mastercam.

Making 2D CAD Model and Produce a Form of Toolpaths:

- The student will draft the part of geometry in 2D or 3D. They also need to create the toolpath for the geometry they have made.
- The geometry command such as draw arc, line endpoints, circle, fillet, chamfer, rectangle, trim, divide, modify and offset will be used.

Determine the necessary Toolpaths to machine the part:

- Once the geometry is completely created, the student has to set up the tool setting, stock size and display of geometry.
- The contour and pocketing toolpath process will be created to remove the material.
- A making holes process by using a drilling toolpath such as countersink, center drill will be created by machine.

Backplot and Verify the task:

- The Backplot is produced to show a path of the tools taken up to cut the desired part. It will show the cutting tool motion and toolpath display.
- The Verify is a simulator feature that will be used when attempting to spot the error in the program. It is a simulation of the toolpath for your analysis and verification before machining the part.

Generate :

- The G and M code will be processed after the student chooses to post a file that obtains the NC file coding.
- When the student completes all the process toolpath operations, the G-code will be generated and ready for machining use.



CHAPTER 2

MILLING PROCESS



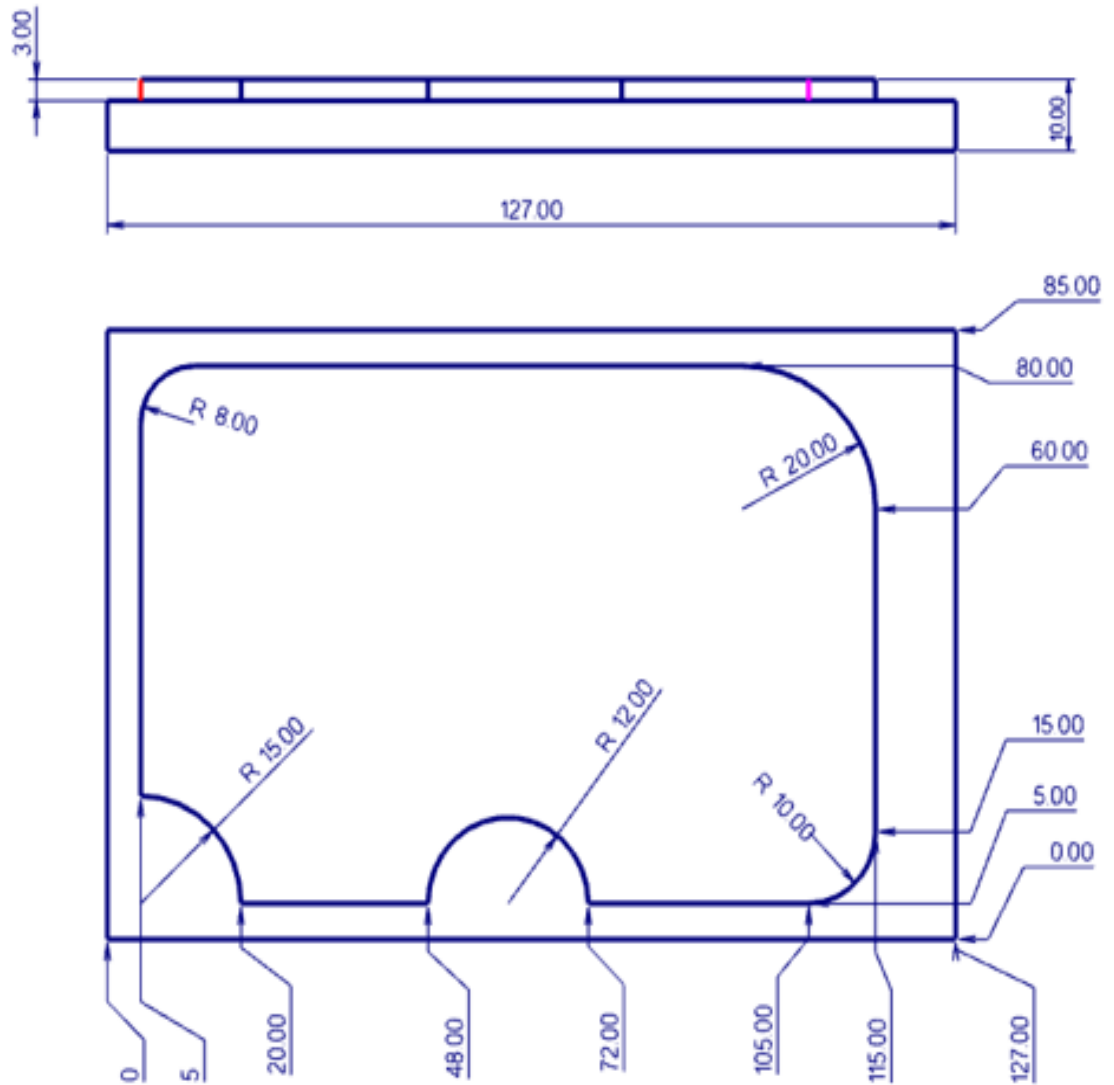
PRACTICAL TASK 1

Overview of making part in milling process :

1. The students will check the drafting and construct the part of geometry.
2. Once the geometry is complete, the student needs to set up the toolpath.
3. It is also included to set up the tool settings, stock size, contour or pocketing toolpath, and the display of geometry will be created by machine.

Instruction :

1. Draw a drafting task by using MasterCam software.
2. Set up the roughing, semi-finishing and finishing processes, including parameters and tool selection.
3. Generate the NC codes from the task.



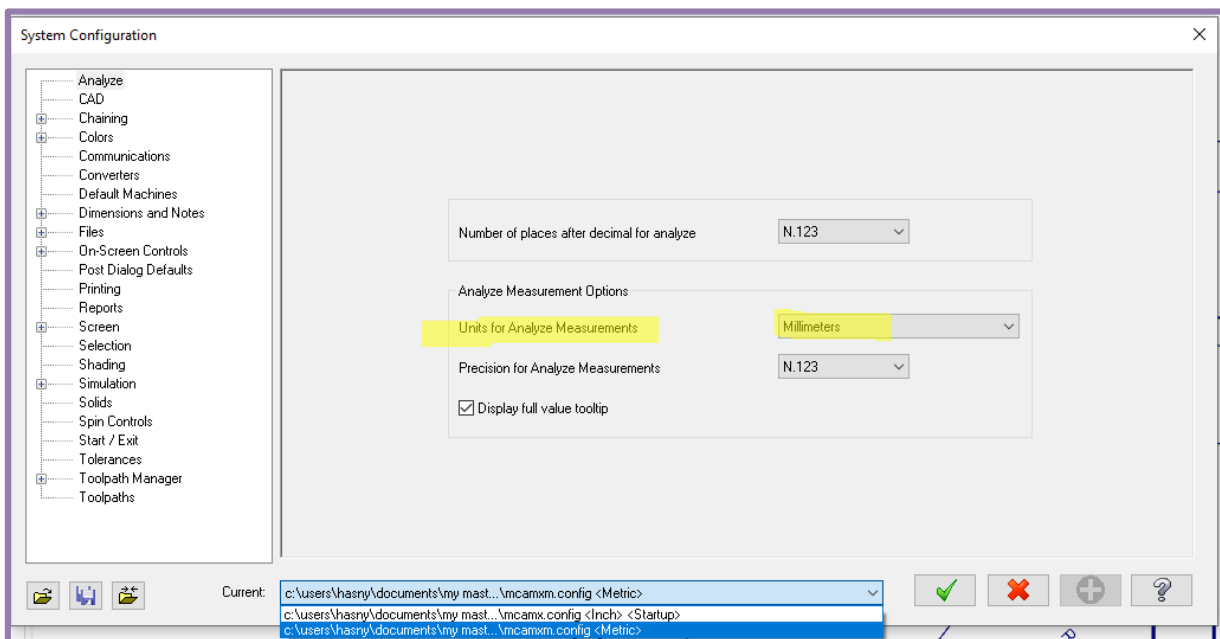
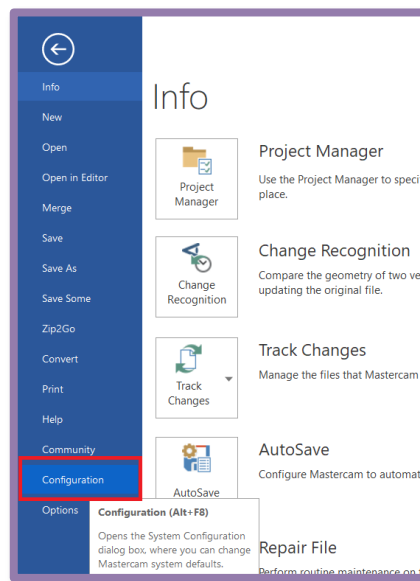
TITLE	MILLING: PRACTICAL TASK 1	DEPT	MECHANICAL ENGINEERING
POLITEKNIK PORT DICKSON		COURSE	DJF41042 (CAD CAM)
		MATERIAL	ALUMINIUM 6061
		UNIT	METRIC (MM)



STEP 1: CREATE A GEOMETRY

1

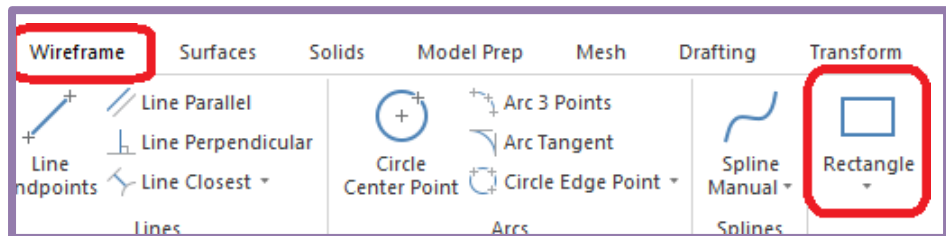
- Open Mastercam software
- From toolbar, click “**SETTING**”
 - Click “**Configuration**”
 - Select **Current** configuration in Metric unit
 - Make sure the **Units** for Analyze Measurements is in **Milimeters**.
- Click **Ok**
- Enter shortcut key “**F9**” for show coordinates axes



2

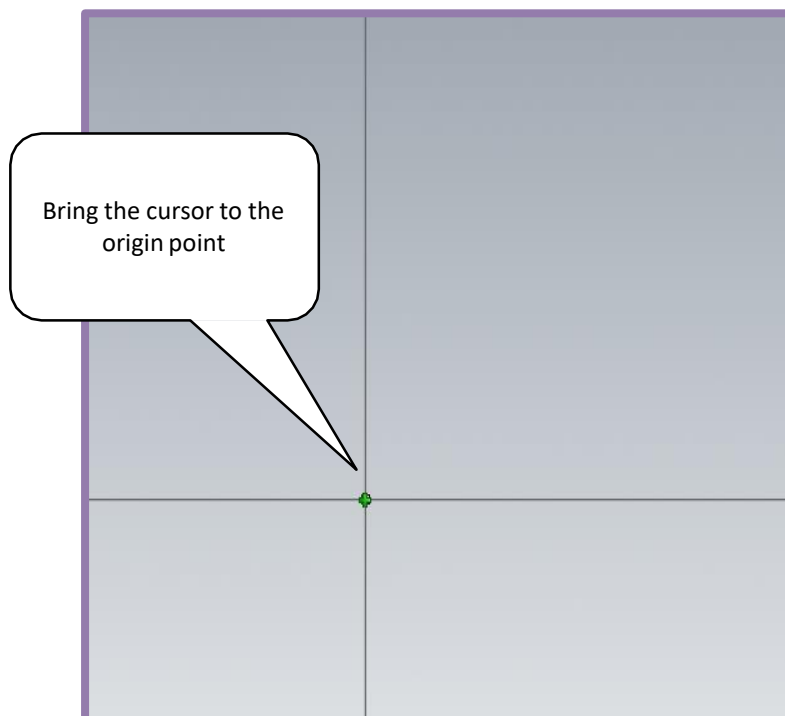
Draw a picture that has been given.

- From toolbar, select **Wireframe**
 - **Rectangle**



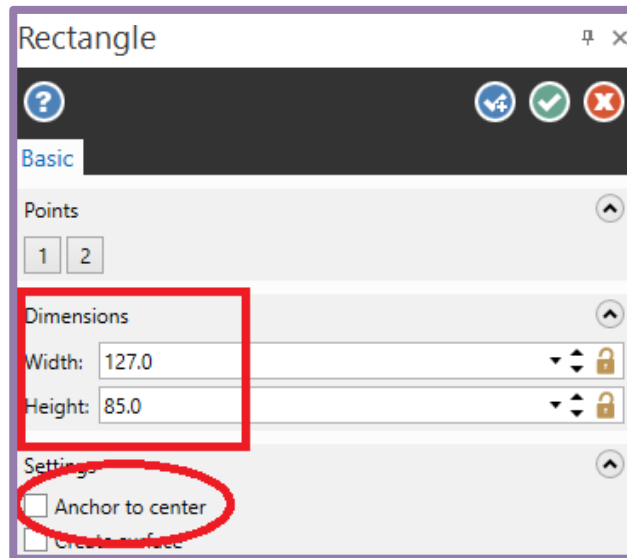
3

- Select position of corner
 - AutoCursor (0,0,0) Or
 - Bring the cursor to the origin point and click.



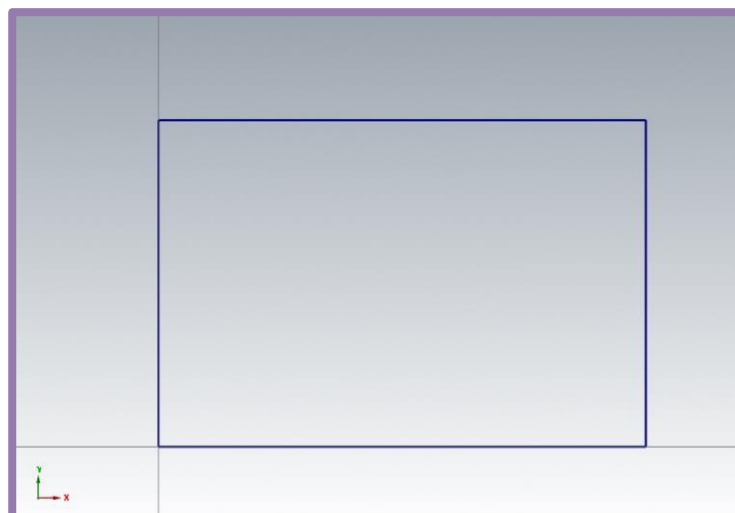
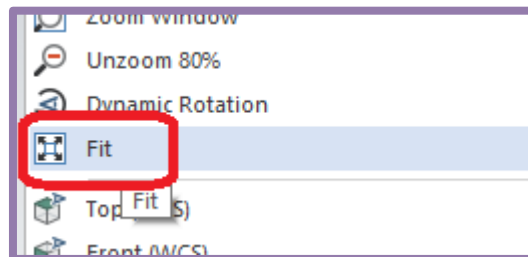
4

- In a Rectangle parameter tab, setting the **width: 127.0** and **height: 85.0**
- Unable **Anchor to center**
- Click **Ok**



5

- Click the **right of mouse** and choose **FIT** command
- The drawing will be zoom all and fit the window.

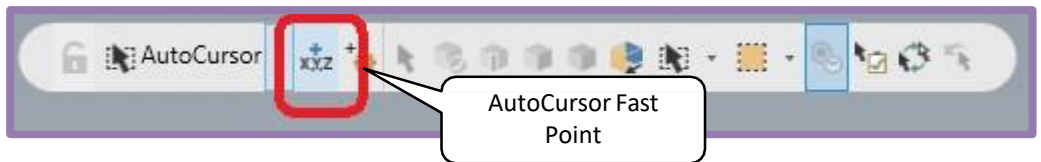


6

- Again, from toolbar, select **Wireframe**
 - **Rectangle**

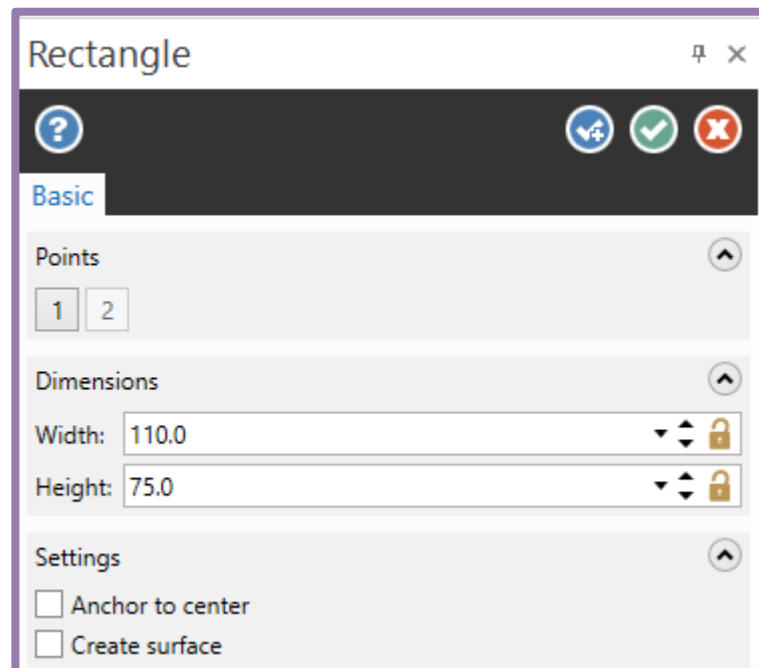
7

- Select position of corner
 - **AutoCursor Fast Point** and type : **5 , 5 , 0**
 - **Enter**



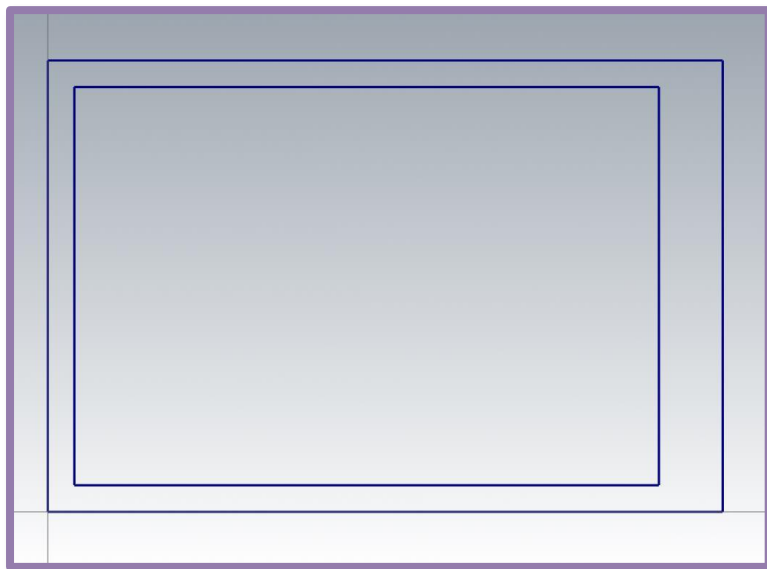
8

- In a Rectangle parameter tab, setting the **width: 110.0** and **height: 75.0**
- Unable **Anchor to center**
- Click **Ok**

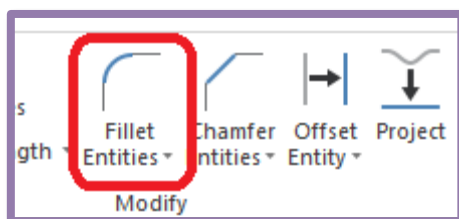


9

- Click the **right of mouse** and choose **FIT** command
- The drawing will be zoom all and fit the window.

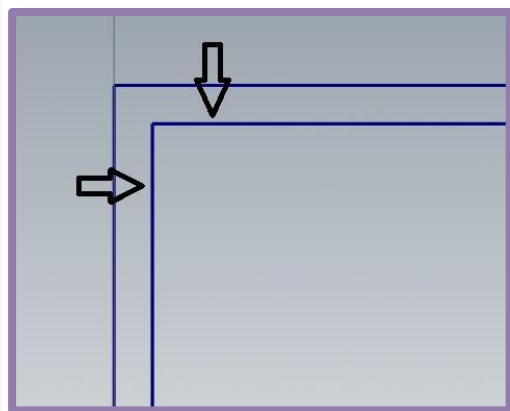
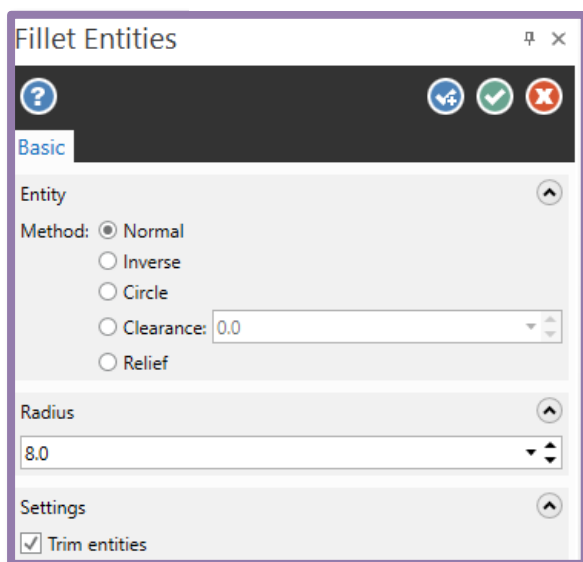


- From toolbar **Wireframe**, select **Fillet Entities**.



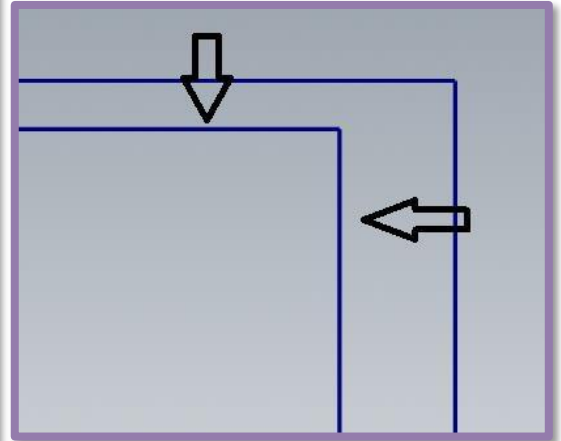
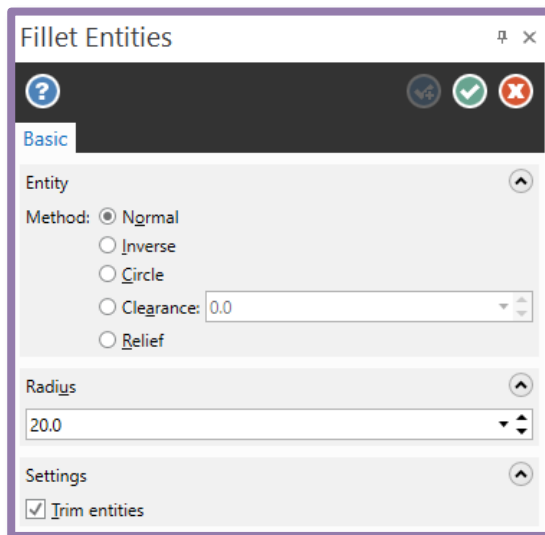
10

- Choose the Fillet Parameter tab and adjust everything as needed, as indicated in the picture.
- Select both entities as shown in the figure.
- Click Ok



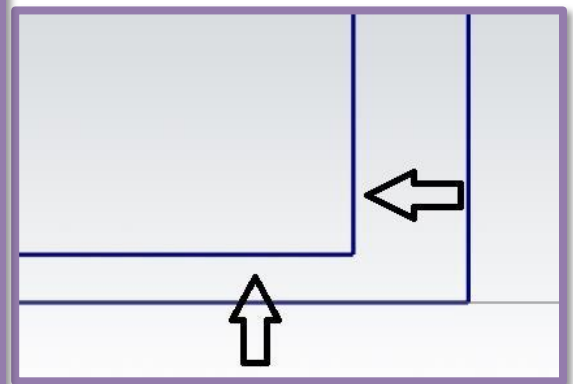
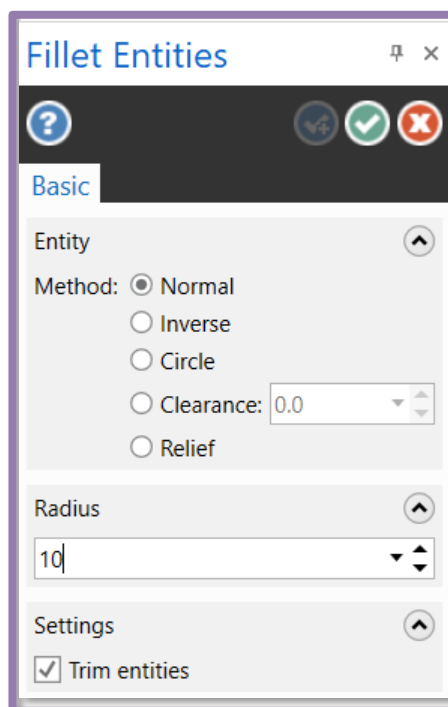
11

- Select the **Fillet Parameter** tab and make all of the necessary changes as shown in the figure.
- Select both entities as shown in the figure.
- Click Ok



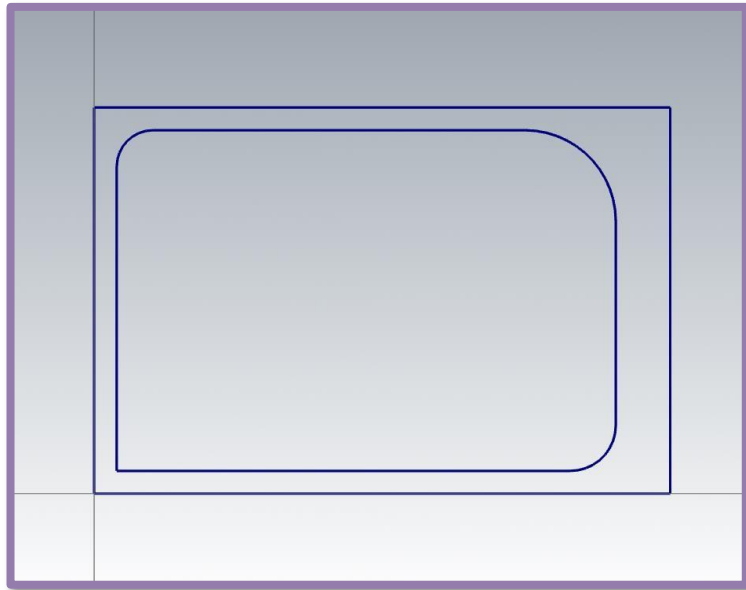
12

- Select the **Fillet Parameter** tab and make all of the necessary changes as shown in the figure.
- Select both entities as shown in the figure.
- Click Ok



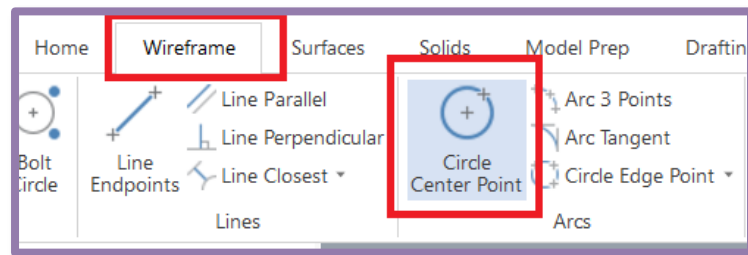
13

- Click right mouse and choose “FIT”



14

- **Select WIREFRAME** feature
 - Circle Center Point



15

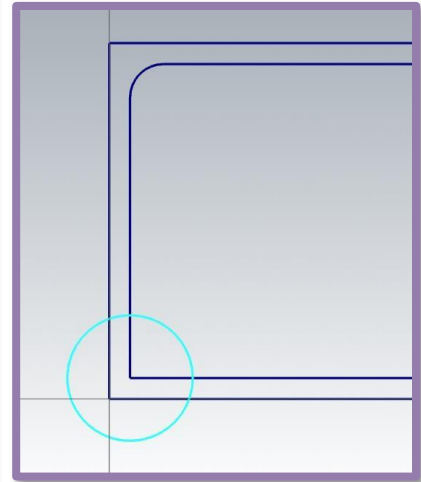
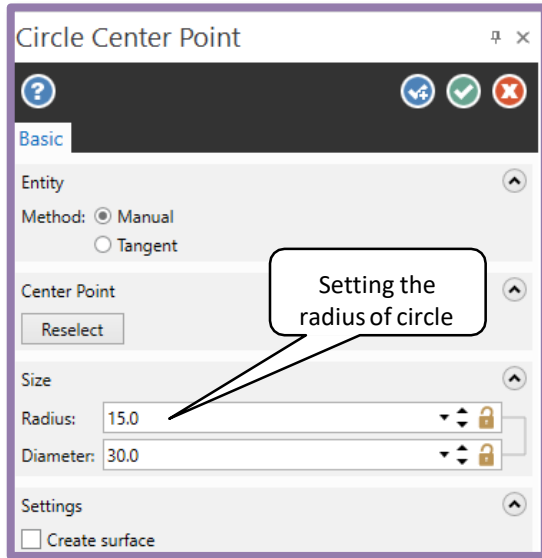
- Setting a coordinate of center point of circle
 - **AutoCursor Fast Point** and type : **5 , 5 , 0**
 - **Enter**



Autocursor
Fast Point

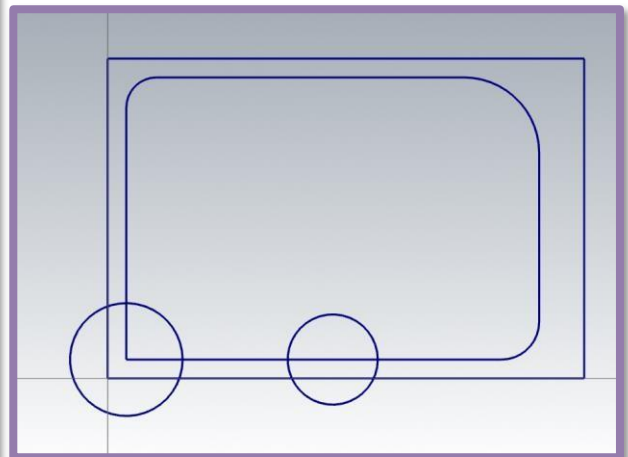
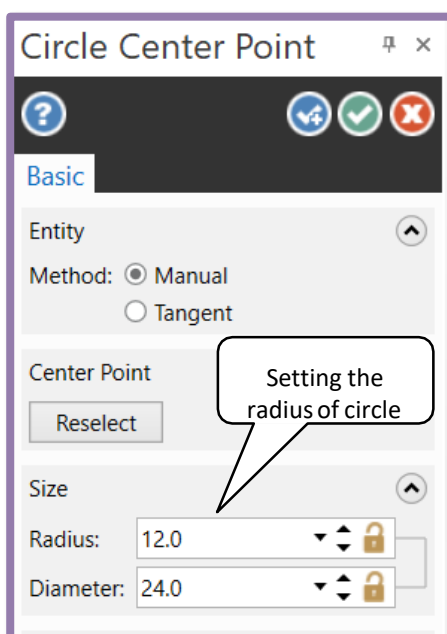
16

- Setting a **radius of circle**
- Click **Ok**



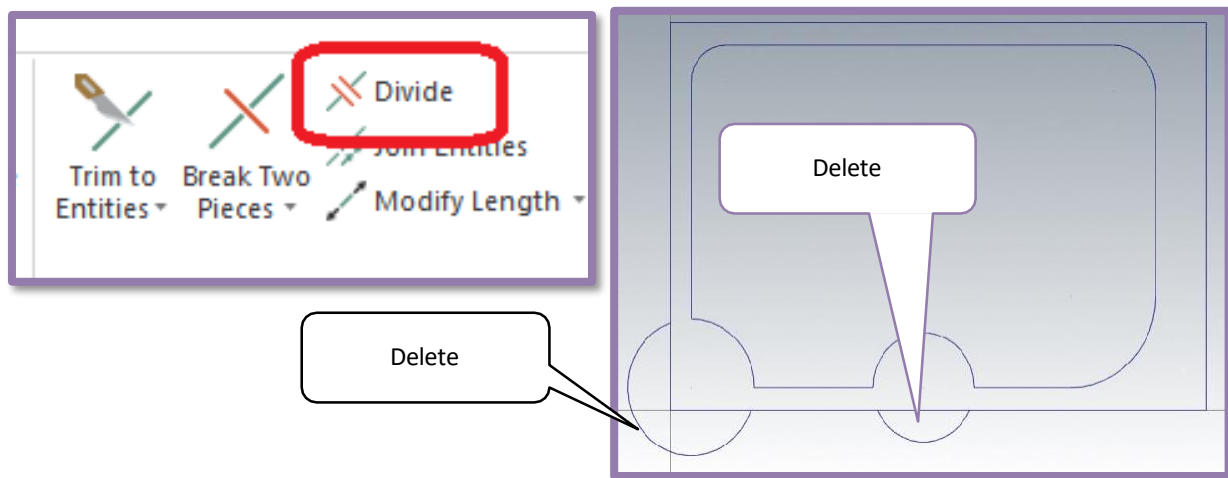
17

- Setting a coordinate of center point of circle
 - **AutoCursor Fast Point (60,5,0)**
- Setting a **radius of circle**
- Click **Ok**



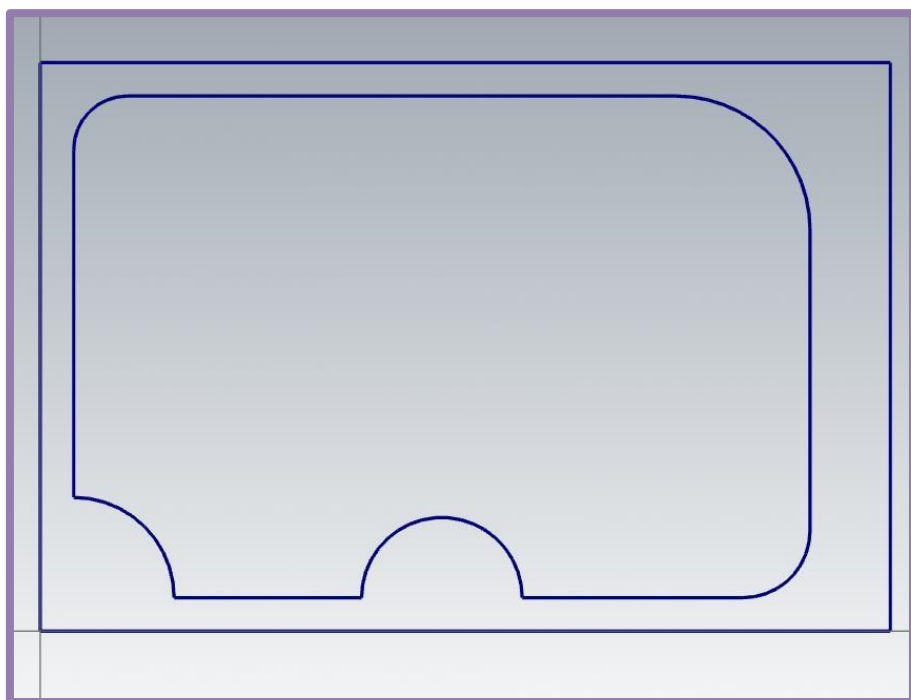
18

- Select **WIREFRAME**
 - **Divide**
 - Select the entities you want to delete.



19

- Save the file : **"Milling Task_1"**



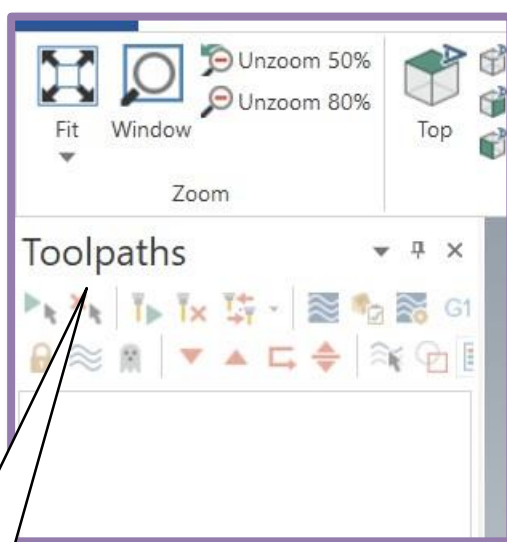
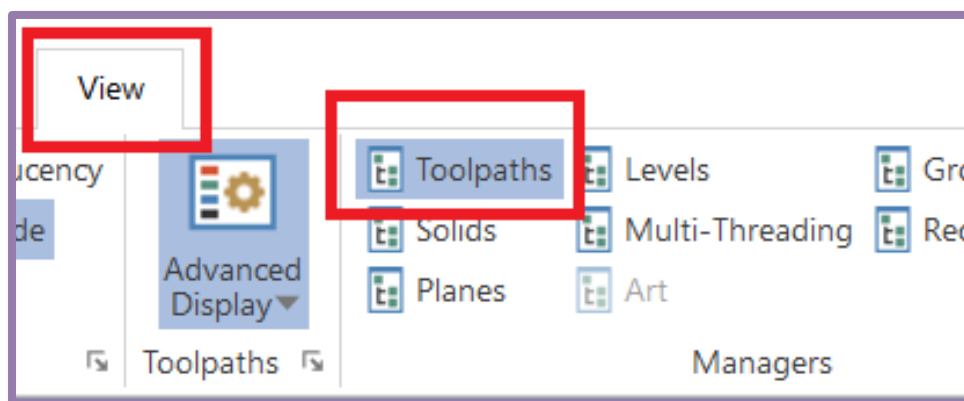


STEP 2: SETUP THE MACHINE DEFINATION AND STOCK

Before we proceed to make any toolpath, we have to select a Machine Definition. The Machine Definition are included mill, lathe, wire router, or mill-turning. It is a template which you can set up the command, features, tool setting, toolpath, spindle speed, feed rate, and plunge rate. It can converted to the G-code when you post the processing after finish the task.

1

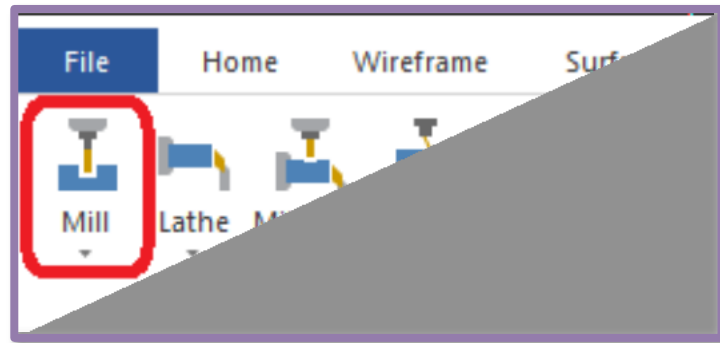
- To display the **Operations Manager**, press **ALT+O** or use **View features**
- Use the **Fit** icon to fit the drawing to the screen



Display of Operating Manager

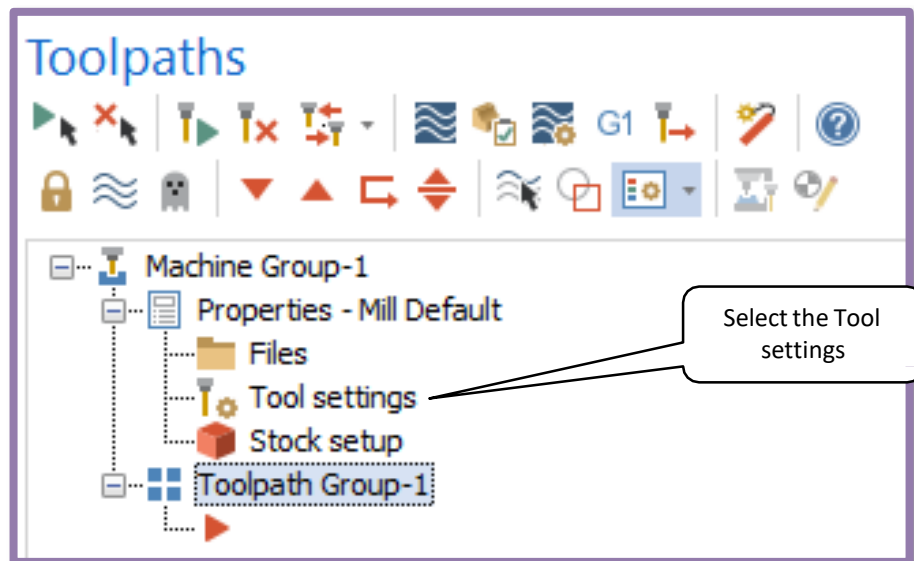
2

- From toolbar, click “Machine Type”.
 - Mill
 - Default



3

- Select **Tool Settings** to set the tool parameters to match figure



4

- Set the **Tool Setting** parameters to match like in the figure

Machine Group Properties

Files Tool Settings Stock Setup

Default program number

Feed Calculation

From tool
 From material
 From defaults
 User defined

Spindle speed
 Feed rate
 Retract rate
 Plunge rate

Adjust feed on arc move
 Minimum arc feed

Toolpath Configuration

Assign tool numbers sequentially
 Warn of duplicate tool numbers
 Use tool's step, peck, coolant
 Search tool library when entering a tool number

Advanced options

Override defaults with modal values
 Clearance height
 Retract height
 Feed plane

Sequence number

Start
 Increment

Program # is sequence number in series that used for any CNC machine consists the movements of cutting tool, to command the spindle speed, and feed rate and external M code in the command. The program is an order which the machine coded in a block programmed.

Assign tool numbers sequentially (setting as a default for your machining group which allows student to overwrite the tool number from the library.

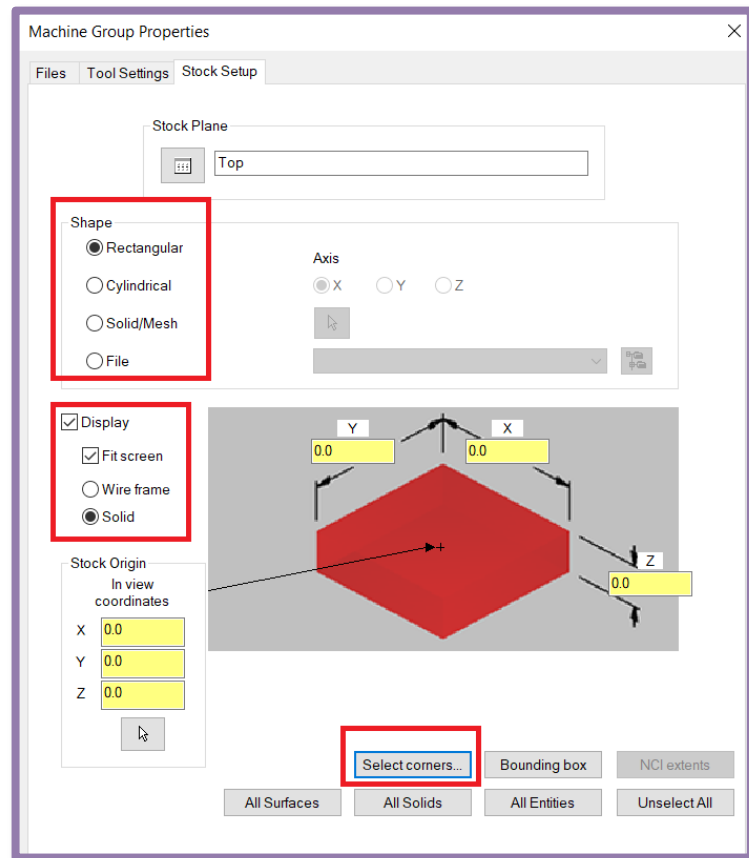
Warn of duplicate tool numbers (The warning will be given when you enter the same number of two tools).

Override Defaults with Modal values enables the system to keep the values that you enter.

Search tool library when entering a tool number (the tool library allows you to select from existing tools in a library as well as define new tools. Tool definitions can be saved in a library or just for the part you work on).

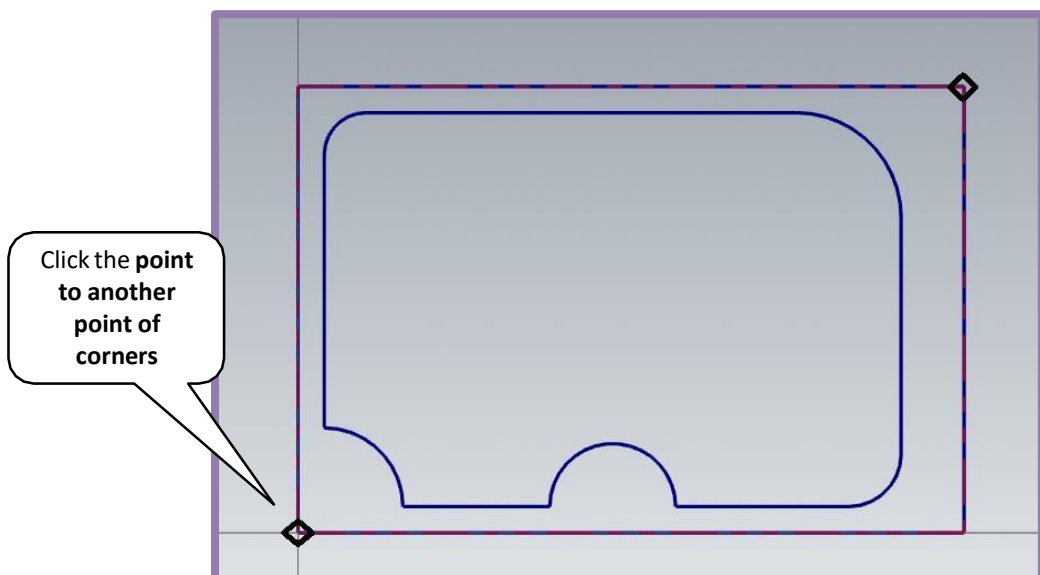
5

- In **Machine Group Properties**, choose the **Stock Setup** tab.
- Set the shape of stock in **Rectangular**.
- Click the **Select corners**



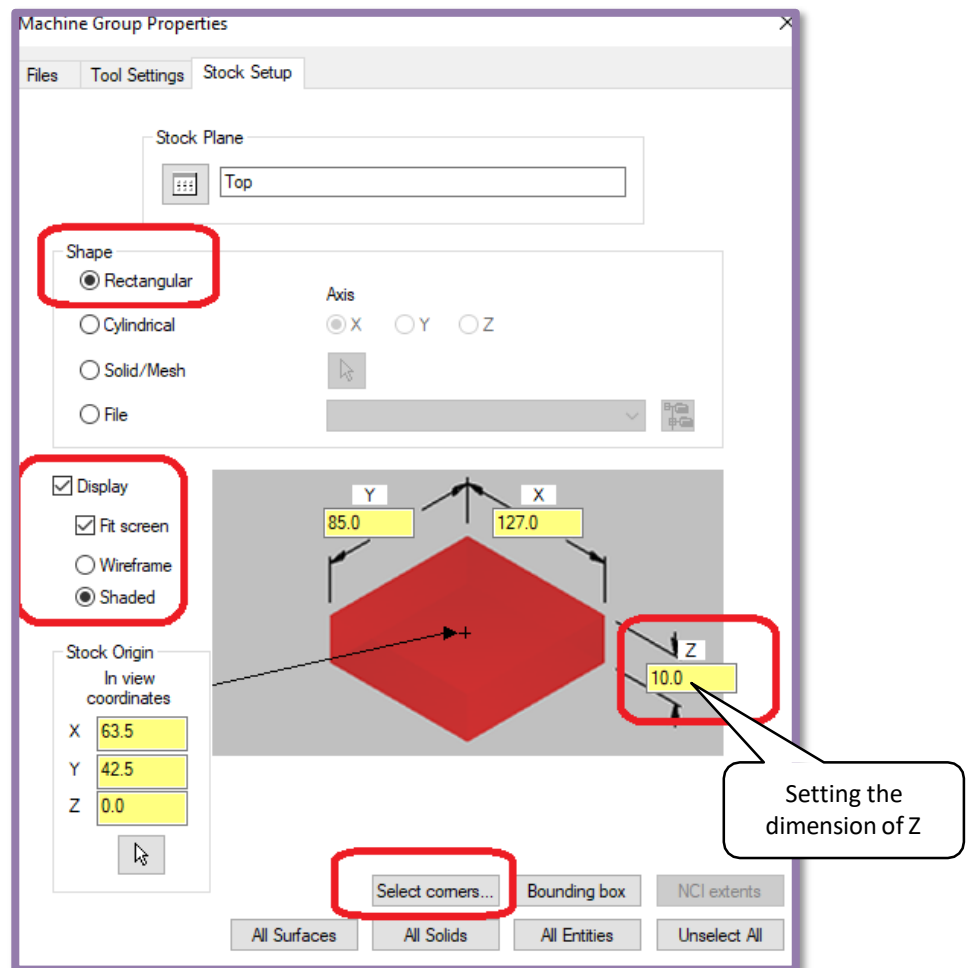
6

- Click the **point to another point of corners**. (These points is the outer boundary of stock).



7

- Enter the dimension of **Z : 10.0** as shown in Figure.



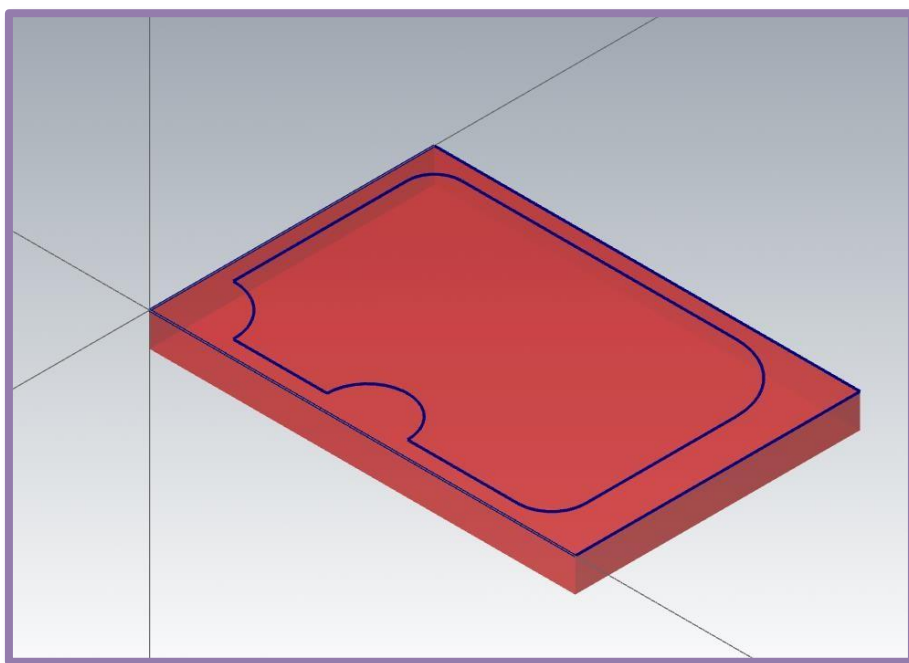
8

- Enter the dimension of **Z:10.0** as shown in Figure.

9

- Click the **OK** button to exit **Machine Group Properties**.
- Select **Isometric** view from the View Toolbar to see the part of the isometric.

The stock model will appear as shown in Figure.



Note: You can display the part of geometry or the toolpath that you have created in the stock model. Used a features backplot, or while verifying toolpaths to showed all the displays.

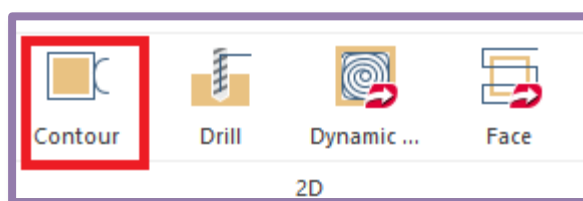


STEP 3: MACHINE THE TWO STEPS USING OPEN CONTOUR TOOLPATH.

Contour toolpath is a 2D high Speed Dynamic milling toolpath. It removes the material along the direction of the tool axis. This path is defined by a loop or chain of curves. Contouring removes an enclosed area and follows a chain only.

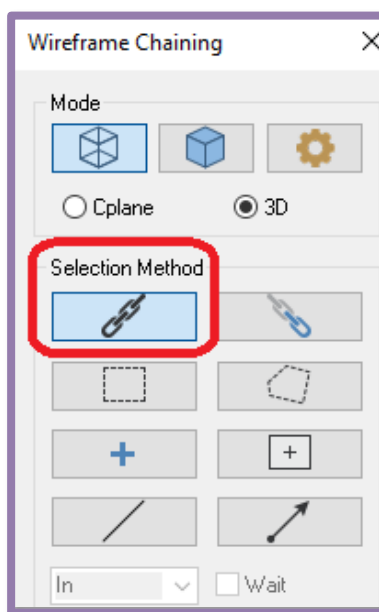
1

- From Toolbar click **“Toolpaths”**.
 - Click **“Contour Toolpath 2D”**.
- Select **OK** button.



2

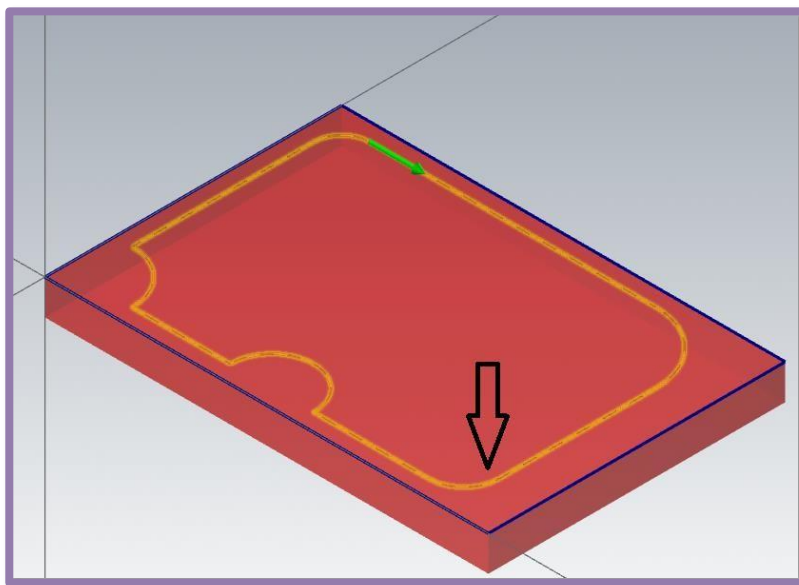
- Enable the Chain button in the chaining dialog box to be able to select just the outside contour.



Note: A chain of entities which are made up of one or more paths or entities. The length has to be same or less than the chaining tolerance 0.002mm (in between the endpoints of two consecutive entities or path). The entities must be together in direction and order

3

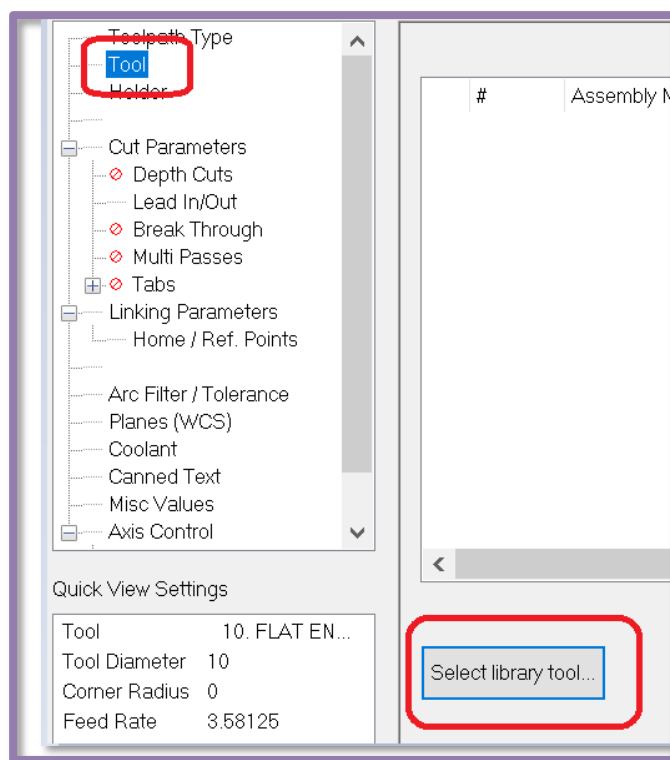
- Select the first chain as shown in the Figure.
- Click “OK” button to exit **Chaining**.
- After that, the toolbar of “Contour (2D)” will come out.



Note: When you are modify the pages, the Mastercam will updates the them (in the Tree View).

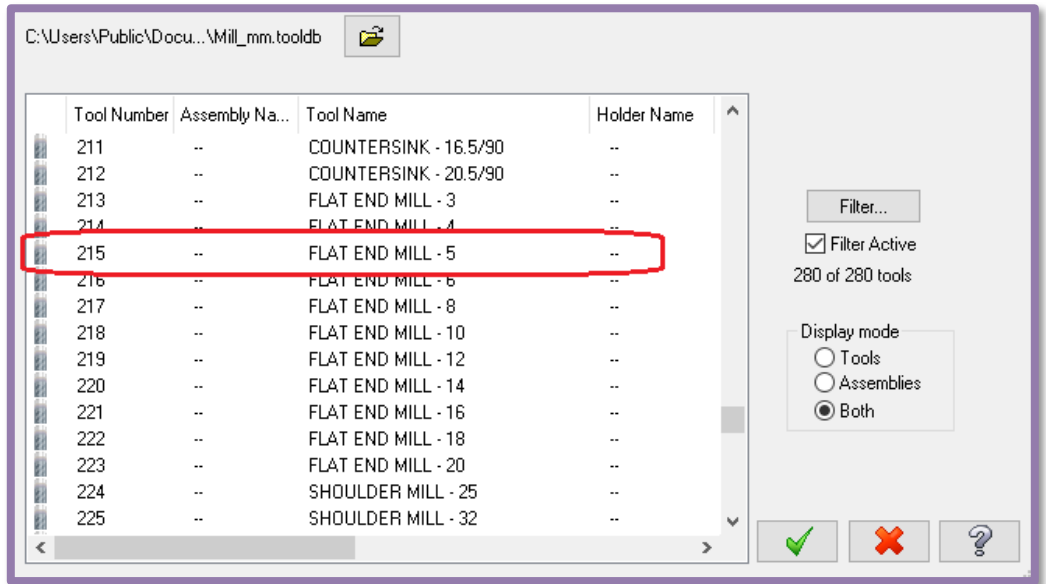
4

- From the Tree View List, choose **Tool**.
- Select **Library Tool** by clicking the button.



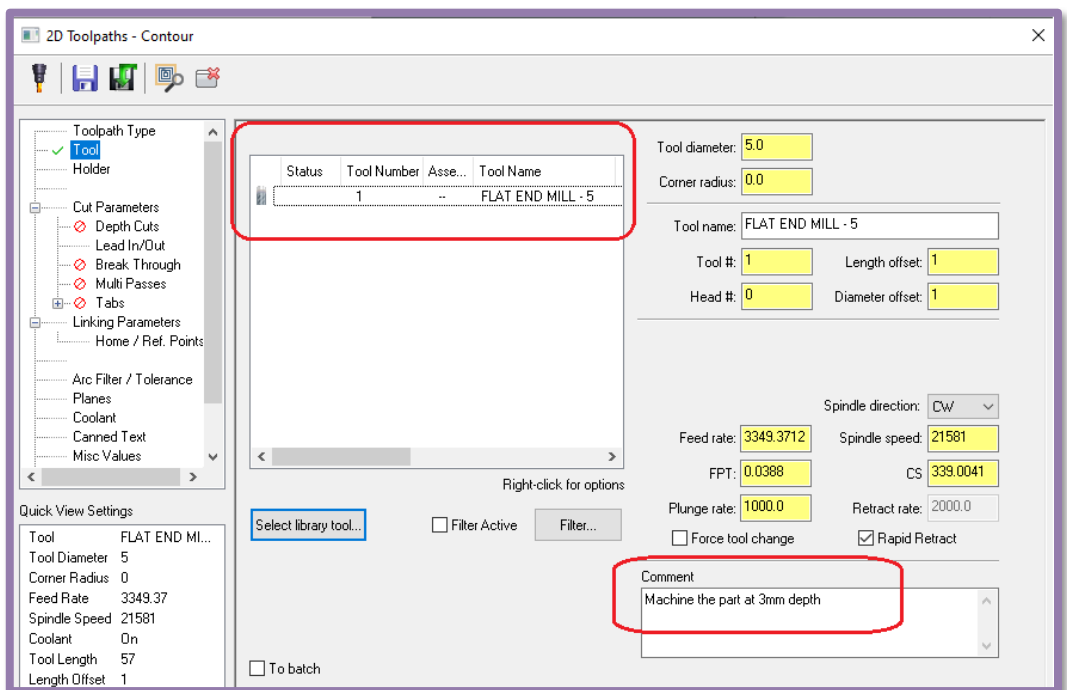
5

- Click **Select library tool** button to choose tool size.
- Choose tool size **Flat Endmill diameter 5.00 mm** as shown in Figure



6

- Select the Tool Selection page and write comment of the process
- Make all the necessary changes as shown in Figure.
- Then select the **OK** button to exit.

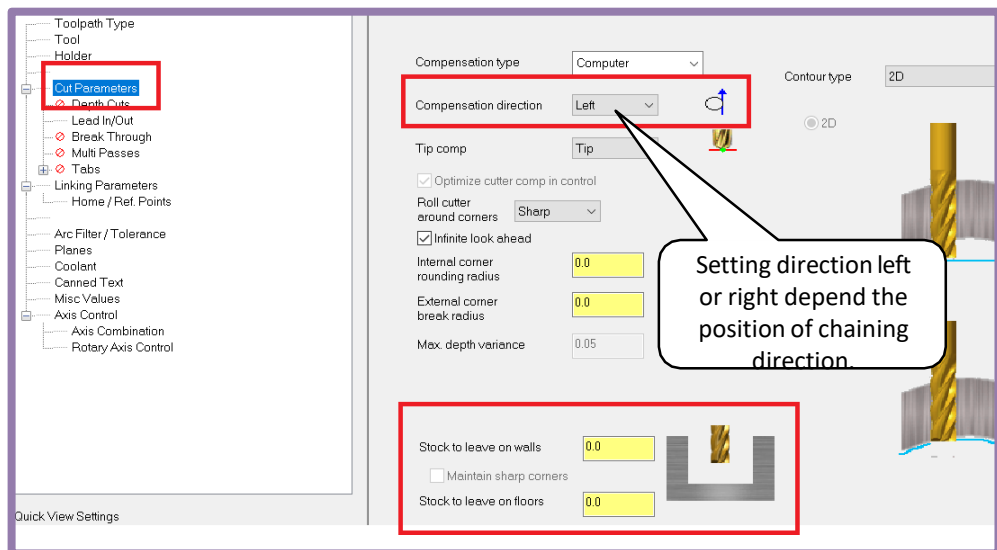


The **Feed rate**, **Plunge rate**, **Retract rate** and **Spindle speed** are roughly based on the part material Aluminium and HSS tooling. The tool parts and material can be chosen to change in the program.

In the **Comment** part is to help you to remark the toolpath as an identity for **The Toolpaths/ Operation Manager**. Please refer the figure below.

7

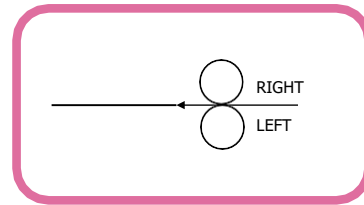
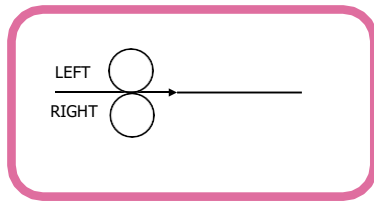
- From the Contour Parameter, select compensation director to set left or right **depend the position of chaining direction**.



8

- From the Contour Parameter, select compensation director to set left or right **depend the position of chaining direction**.

Note: Compensation Direction is to set the chaining direction. You can set the direction to offset either left or right depending on the location of the cutting/thread point entity outside or inside the contour.



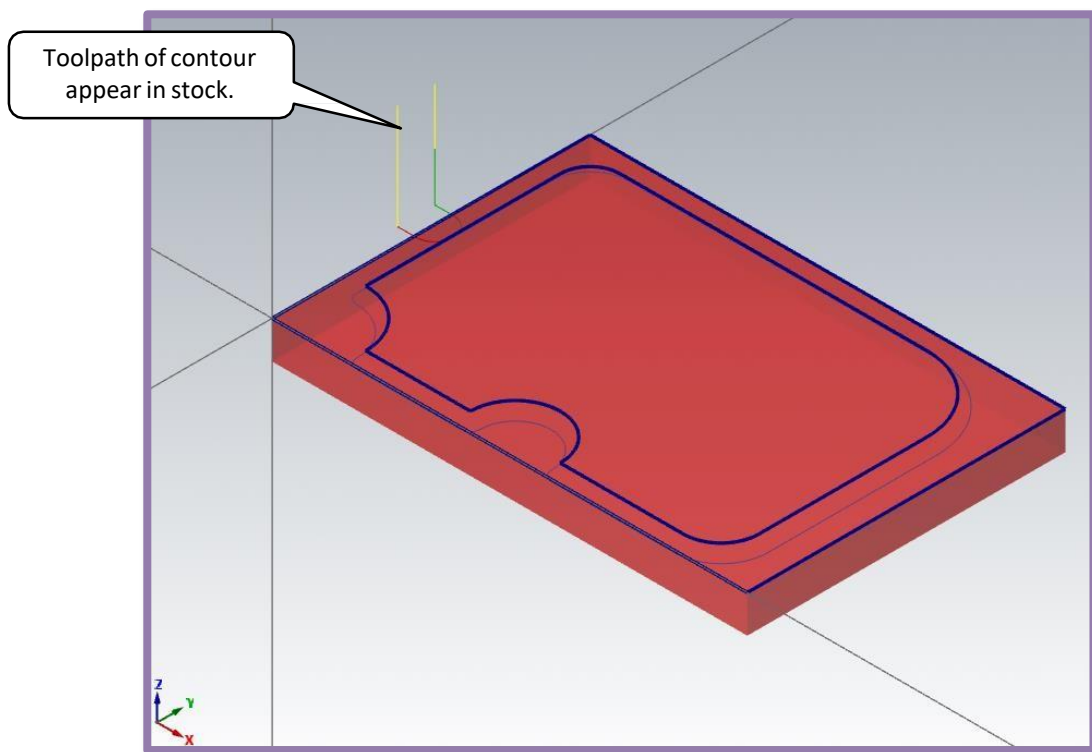
9

- Select the **Linking Parameter**
- Determine the parameters of depth : **3mm**
- Click the **OK** for exit the command purposes.

The screenshot displays the Mastercam software interface for configuring toolpath parameters. On the left, a tree view shows various settings, with 'Linking Parameters' highlighted in a red box. The main panel on the right shows a 3D model of a drill bit and a workpiece. Several parameters are visible, with 'Depth...' set to -3.0 and highlighted in a red box. Other parameters include 'Retract...' at 25.0, 'Feed plane...' at 10.0, and 'Top of stock...' at 0.0. The 'Depth...' field is set to 'Incremental' mode.

10

- Click the **OK** button to exit **2D Toolpath- Contour** parameter.
- Select **Isometric** view from the View Toolbar to see the part of the isometric. The toolpath of contour in will appear as shown in Figure.



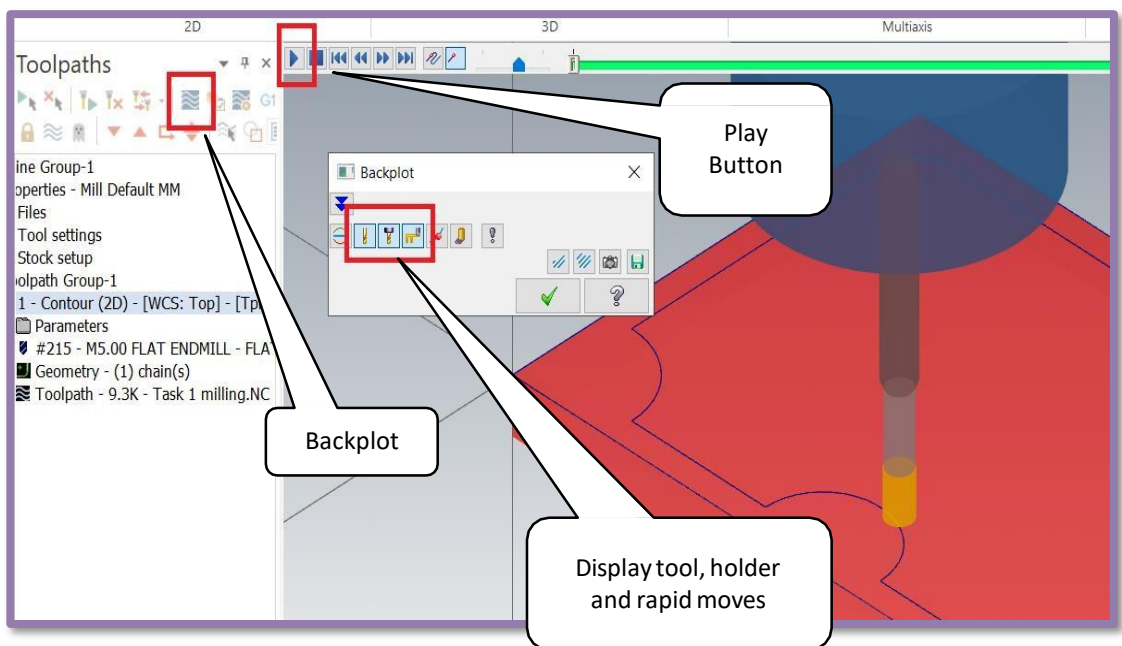


STEP 4: BACKPLOT THE TOOLPATHS

Backplotting shows the toolpath of cutting tool to cut the part. This display will let you spot an error when you run the mode. You can define it in the program before you machine the parts. MasterCam will display the coordinates of X, Y and Z axis in the screen when you backplot the paths.

1

- On the operations button select the **Backplot**.
- Turn on all the buttons (to **display tool, holder and rapid moves**).
- Adjust the speed of the backplot.
- Click a **Play** button.
- Click **Ok** to exit the Backplot



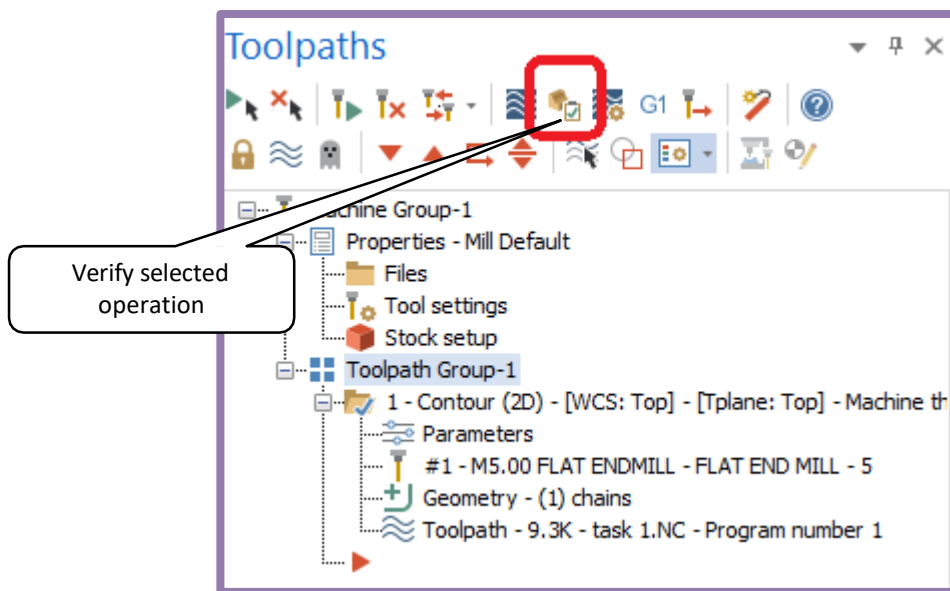


STEP 5 : VERIFY THE TOOLPATHS

Verify allows you to simulate the machining of a part by using a solid model. It is a simulation graphic of the movement of the cutting tool and the material removing pass by pass. It also can give a reminder of collisions between the tool and stock if any happened.

1

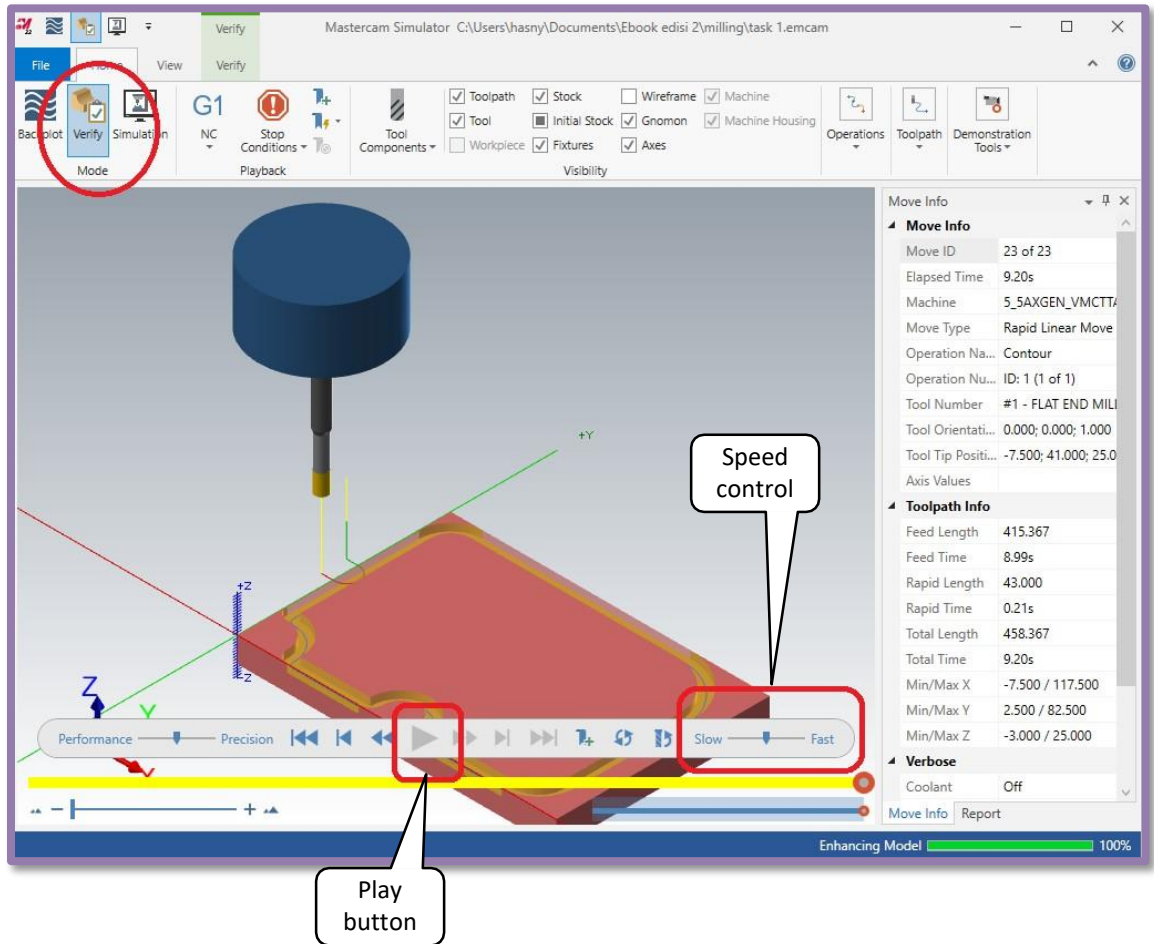
- From **Operation Manager**, choose **Verify selected operations** icon as shown in Figure.



- The MasterCam Simulator window will pop up as shown in the figure.
- To start simulation select the Play button.
- The simulation will be played.

2

- The speed control can be adjusted as shown in the figure.



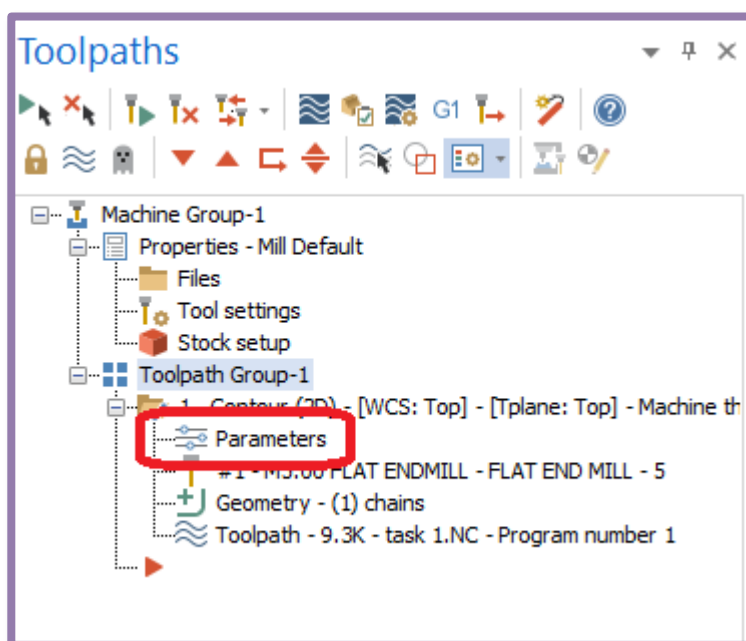


STEP 6: Adding MULTI PASSES of toolpath

Multi Passes lets you make more than one passes of cutting tool in your toolpath. It is for clearance purposes (to remove material as you desired). The tool proposed the multiple cutting passes for the **roughing** and **finishing** passes.

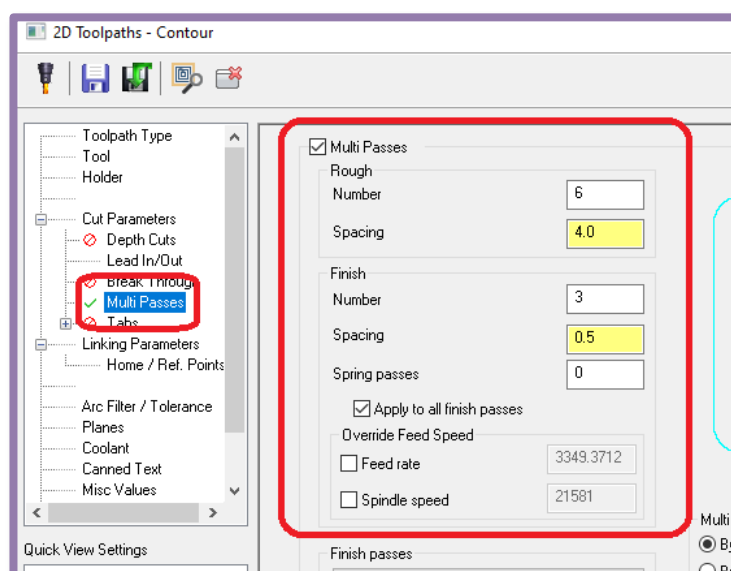
1

- Select **Parameter** to open ToolPath Setting



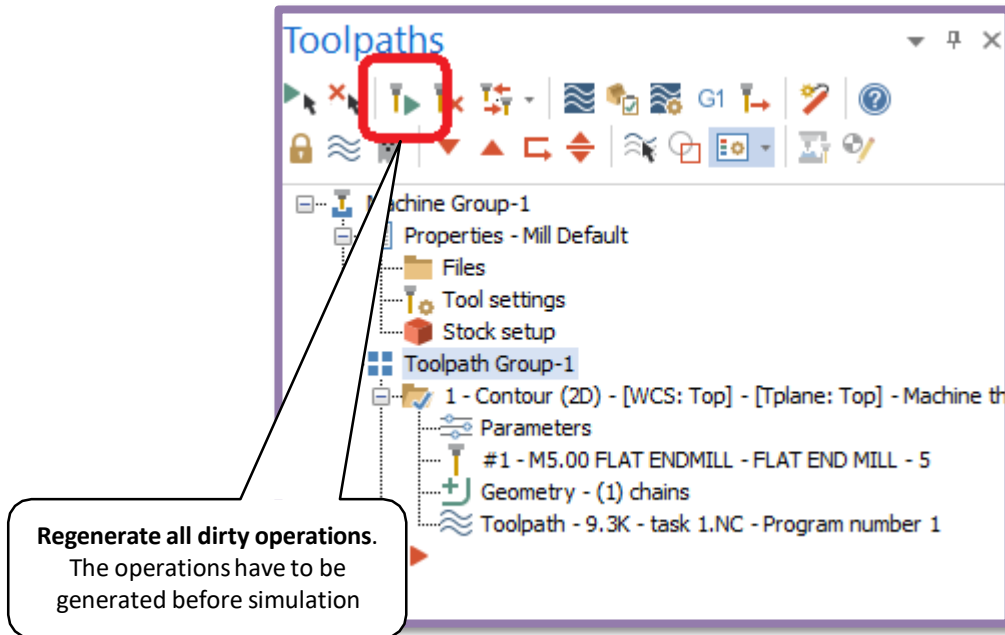
2

- Select **Multi Passes** from the **Tree view list**
- Select the necessary changes.
- Enter a value for the **Number of passes**.
- Click **OK** to exit the **Multi Passes** parameters.



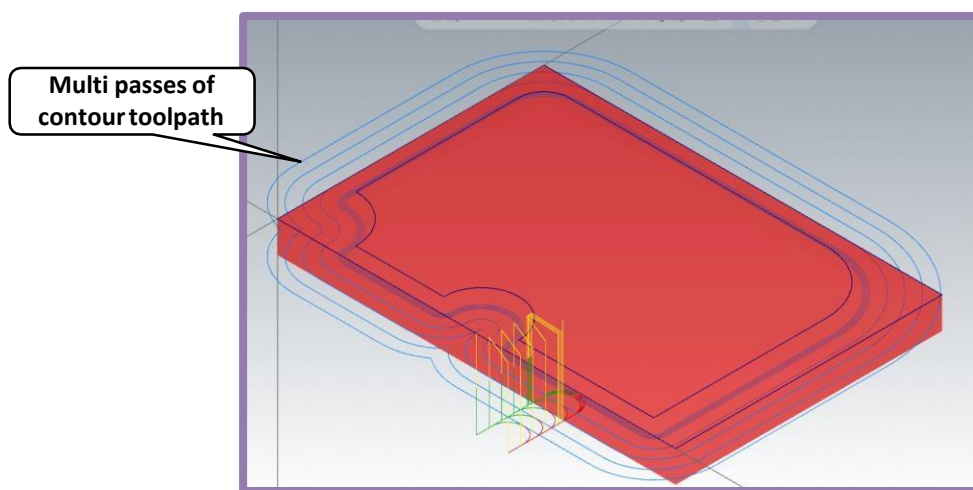
3

- Select the button of “**Regenerate all dirty operations**”.
- This feature will regenerate the modification that you have made to the toolpaths.



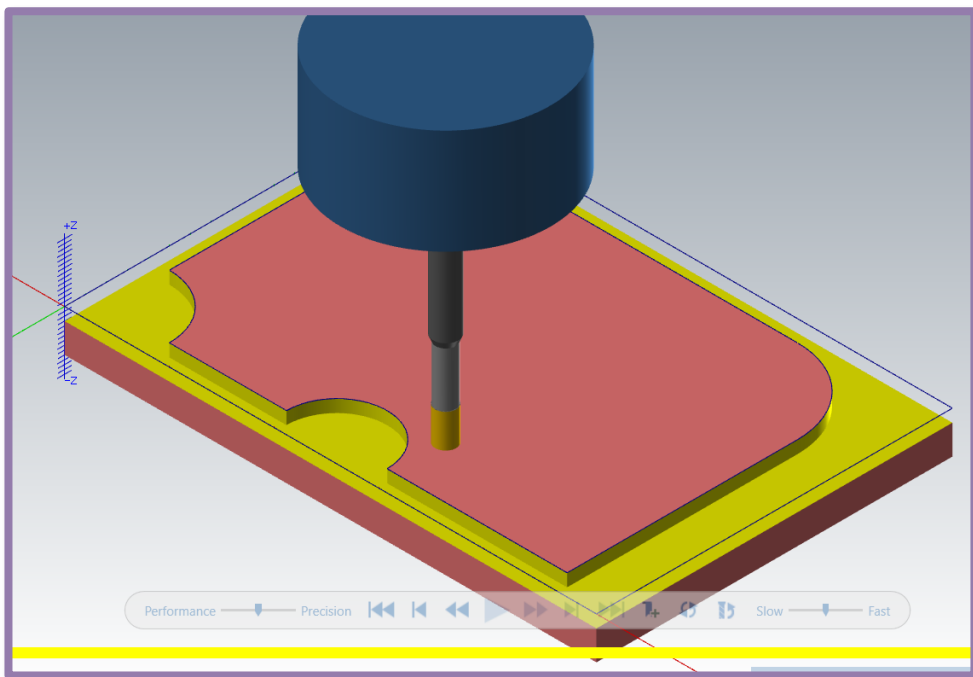
4

- Select **Isometric** view from the View Toolbar.
- The multi passes of contour toolpath will appear as shown in Figure.



5

- To **Backplot** and **Verify** the toolpaths, please refer back the topic to review these procedures.



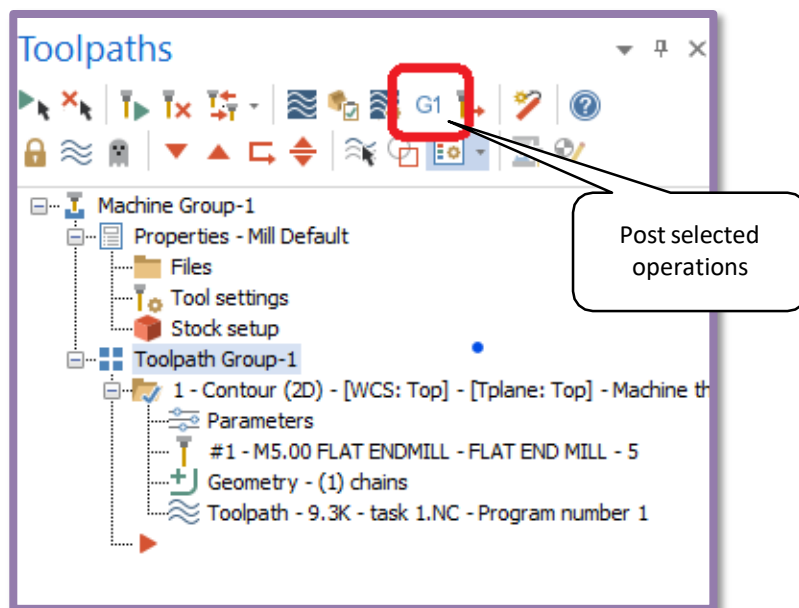


STEP 7: POST THE FILE

Post processing, or **posting a program**, it refers to convert the process of toolpaths in your Mastercam part files to a format that can be understood by your CNC machine tool's control. For example, G-codes and M-codes.

1

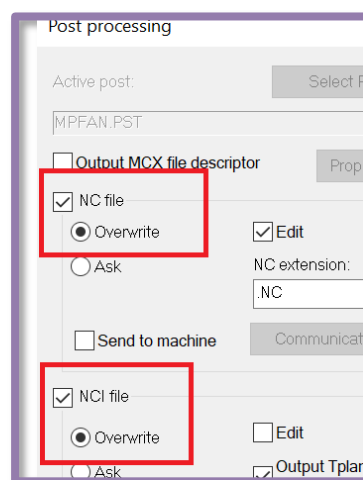
- Click the **Post selected operation** button in the Operation Manager.
- Make the necessary changes as shown in **Post Processing** window in the figure above.



- Note: Make sure all the operations are selected. **Select Overwrite** in the **Operations Manager**.

2

- Set the **Post Processing** like the Figure.
- Select **OK** button to continue.



3

- After generates NC codes, copy all the codes and save in as **Notepad** version.
- You can use this post in the **NC code** at CNC machine.

```
2 O0001(T)
3 (DATE=DD-MM-YY - 13-08-21 TIME=HH:MM -
4 (MCX FILE - C:\USERS\LENOVO\DOCUMENTS\P
5 (NC FILE - C:\USERS\LENOVO\DOCUMENTS\MY
6 (MATERIAL - ALUMINUM MM - 2024)
7 ( T1 | 5. FLAT ENDMILL | H1 )
8 N100 G21
9 N110 G0 G17 G40 G49 G80 G90
10 ( CONTOUR THE PART DEPTH 3.0MM )
11 N120 T1 M6
12 N130 G0 G90 G54 X-34. Y41. A0. S2000 M3
13 N140 G43 H1 Z25.
14 N150 Z10.
15 N160 G1 Z-3. F10.
16 N170 X-29.
17 N180 G3 X-24. Y46. I0. J5.
18 N190 G1 Y72.
19 N200 G2 X13. Y109. I37. J0.
20 N210 G1 X105.
21 N220 G2 X144. Y70. I0. J-39.
22 N230 G1 Y25.
23 N240 G2 X95. Y-24. I-49. J0.
24 N250 G1 X72.
25 N260 G2 X60. Y-21.401 I0. J29.
26 N270 X48. Y-24. I-12. J26.401
27 N280 G1 X20.
28 N290 G2 X-6.585 Y-6.585 I0. J29.
29 N300 X-24. Y20. I11.585 J26.585
30 N310 G1 Y46.
```



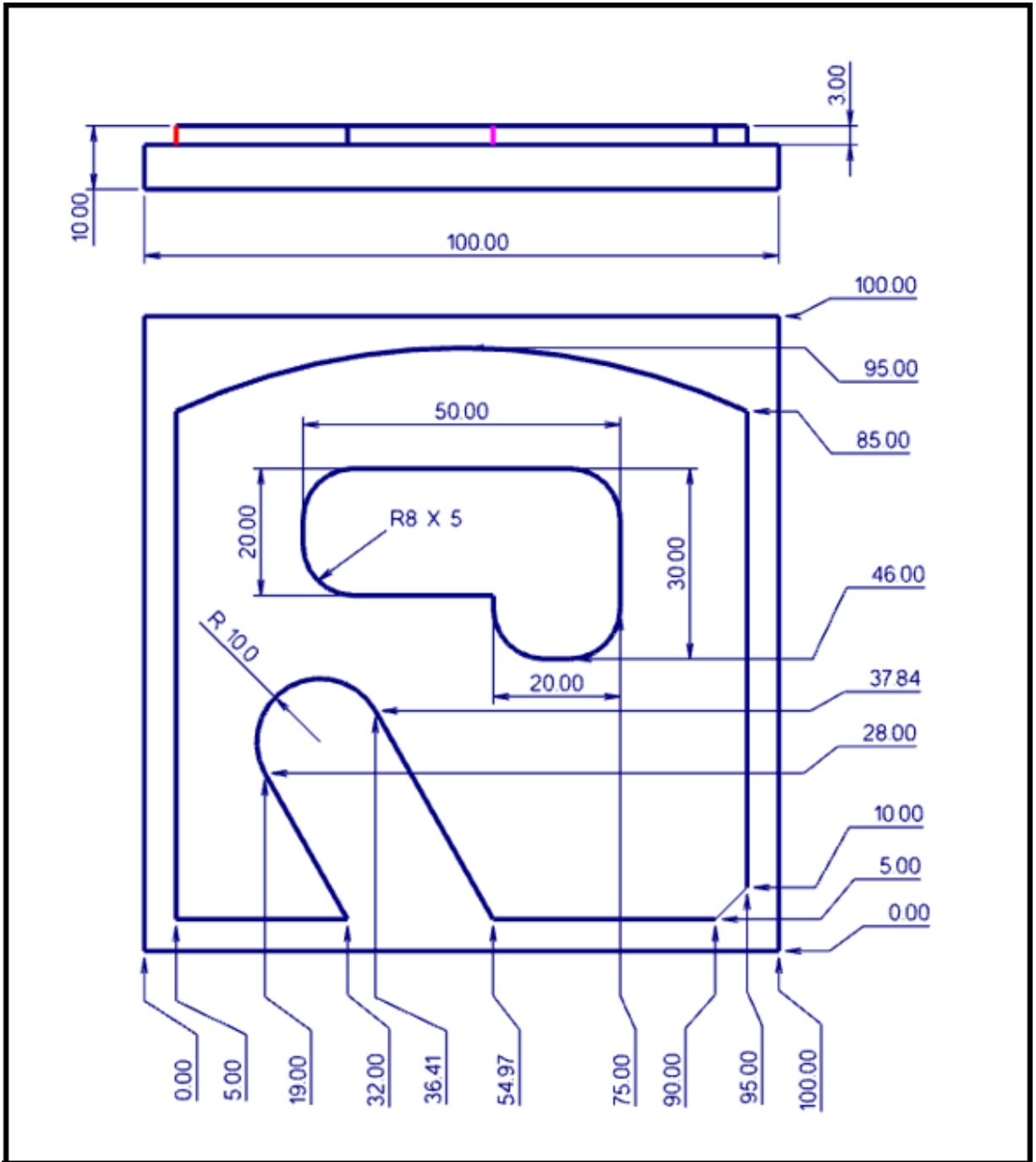
PRACTICAL TASK 2

Overview of making part in milling process :

1. The students will checking the drafting and construct the part of geometry.
2. Once the geometry complete create, the student need to setup the toolpath.
3. It is also included setup the tool setting, stock size, contour or pocketing toolpath and display of geometry will be created by machine.

Instruction :

1. Draw a drafting tasks by using MasterCam software.
2. Setup the roughing, semi finishing and finishing process include parameter and tool selection.
3. Generate the NC codes from the task.



TITLE	MILLING: PRACTICAL TASK 2	DEPT	MECHANICAL ENGINEERING
POLITEKNIK PORT DICKSON		COURSE	DJF41042 (CAD CAM)
		MATERIAL	ALUMINIUM 6061
		UNIT	METRIC (MM)



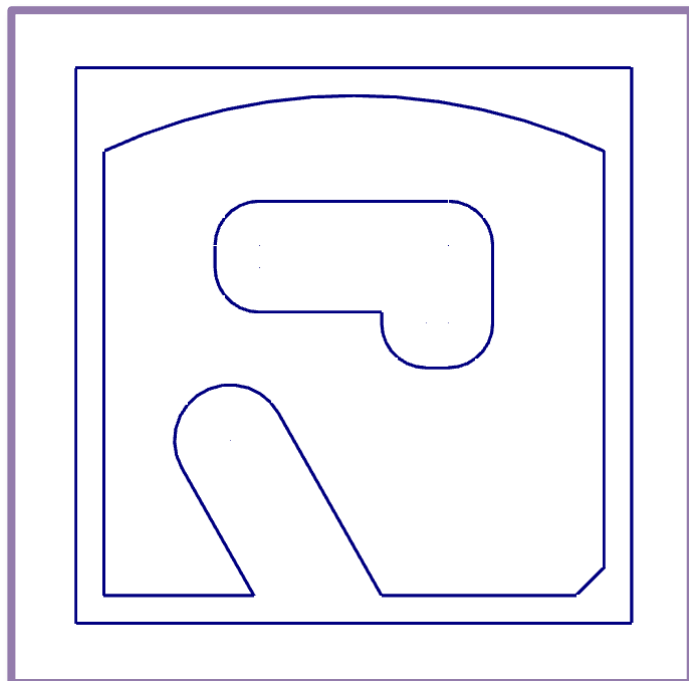
STEP 1: CREATE A GEOMETRY

1

- Scan the QR code.



- OR click the link :
<https://drive.google.com/file/d/1Xx8j32icFEhQgPNkveUEhcAxJniz6UBI/view?usp=sharing>
- Watch the tutorial video how to drafting the task in MasterCam .
- Draw a picture that has been given.
- Save the file : “Task_2”



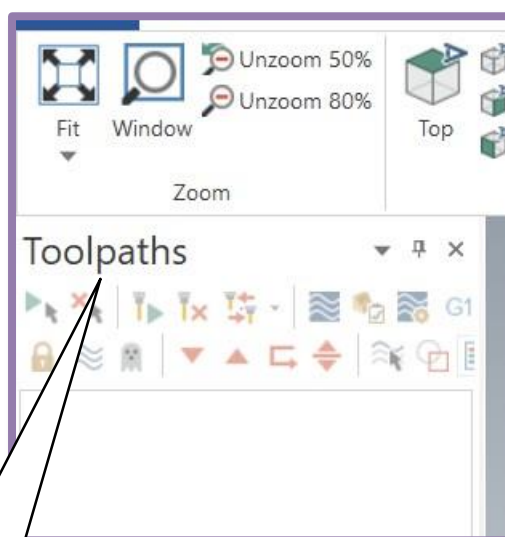
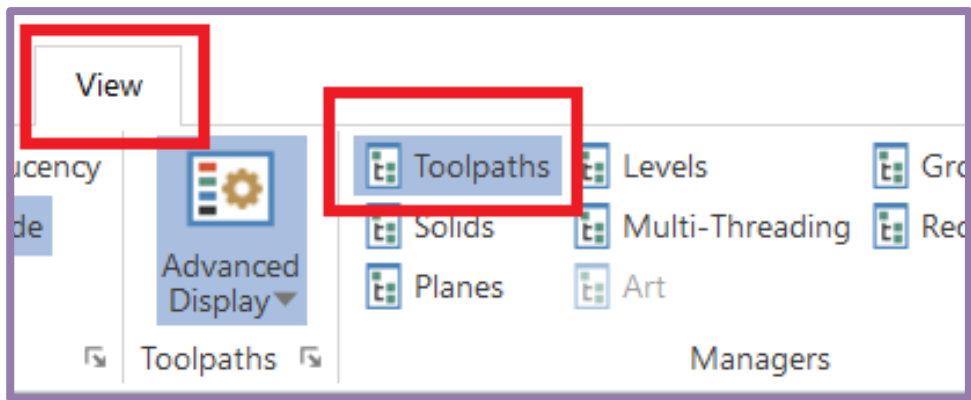


STEP 2: SETUP THE MACHINE DEFINITION AND STOCK

Before we proceed to make any toolpath, we have to select a **Machine Definition**. The Machine Definition are included mill, lathe, wire router, or mill-turning. It is a template which you can set up the command, features, tool setting, toolpath, spindle speed, feed rate, and plunge rate. It can convert to the G-code when you post the processing after finish the task.

1

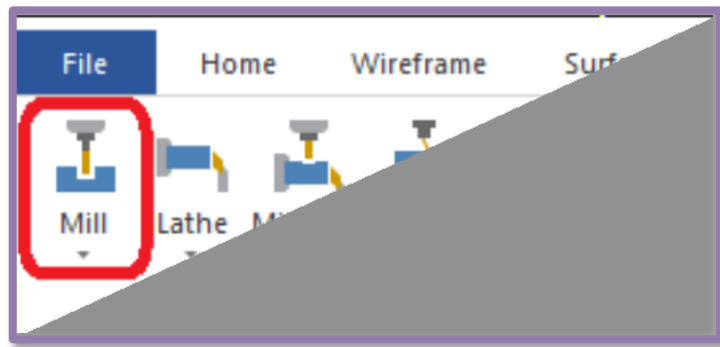
- To display the **Operations Manager**, press **ALT+O** or use **View features**
- Use the **Fit** icon to fit the drawing to the screen



Display of Operating Manager

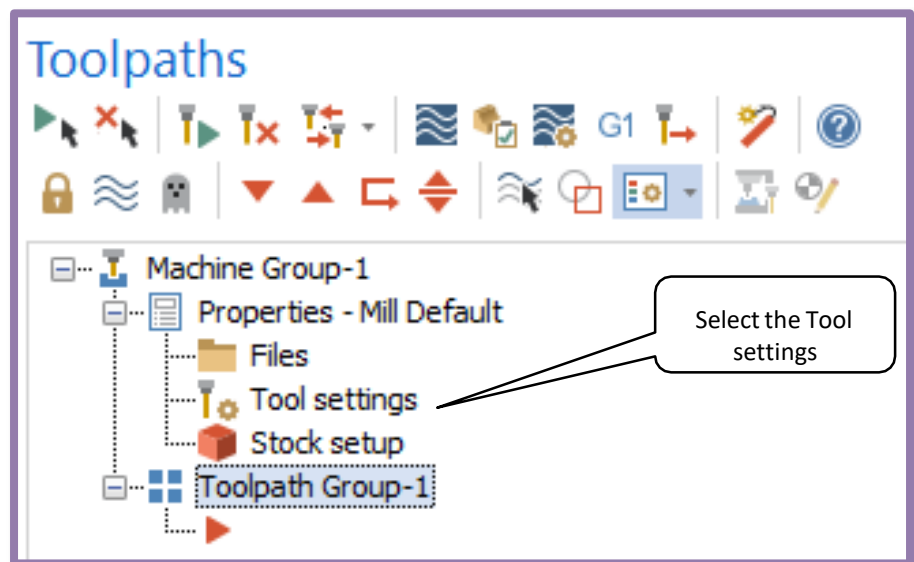
2

- From toolbar, click “Machine Type”.
- Mill
- Default



3

- Select **Tool Settings** to set the tool parameters to match figure



4

- Set the **Tool Setting** parameters to match like in the figure

Machine Group Properties

Files | Tool Settings | Stock Setup

Default program number: 2

Feed Calculation

- From tool
- From material
- From defaults
- User defined

Spindle speed: 5000.0

Feed rate: 100.0

Retract rate: 150.0

Plunge rate: 25.0

Adjust feed on arc move

Minimum arc feed: 125.0

Toolpath Configuration

- Assign tool numbers sequentially
- Warn of duplicate tool numbers
- Use tool's step, peck, coolant
- Search tool library when entering a tool number

Advanced options

- Override defaults with modal values
 - Clearance height
 - Retract height
 - Feed plane

Sequence number

Start: 100.0

Increment: 10.0

Material: ALUMINUM mm - 2024

Edit... Select...

Program # is sequence number in series that used for any CNC machine consists the movements of cutting tool, to command the spindle speed, and feed rate and external M code in the command. The program is an order which the machine coded in a block programmed.

Assign tool numbers sequentially (setting as a default for your machining group which allows student to overwrite the tool number from the library).

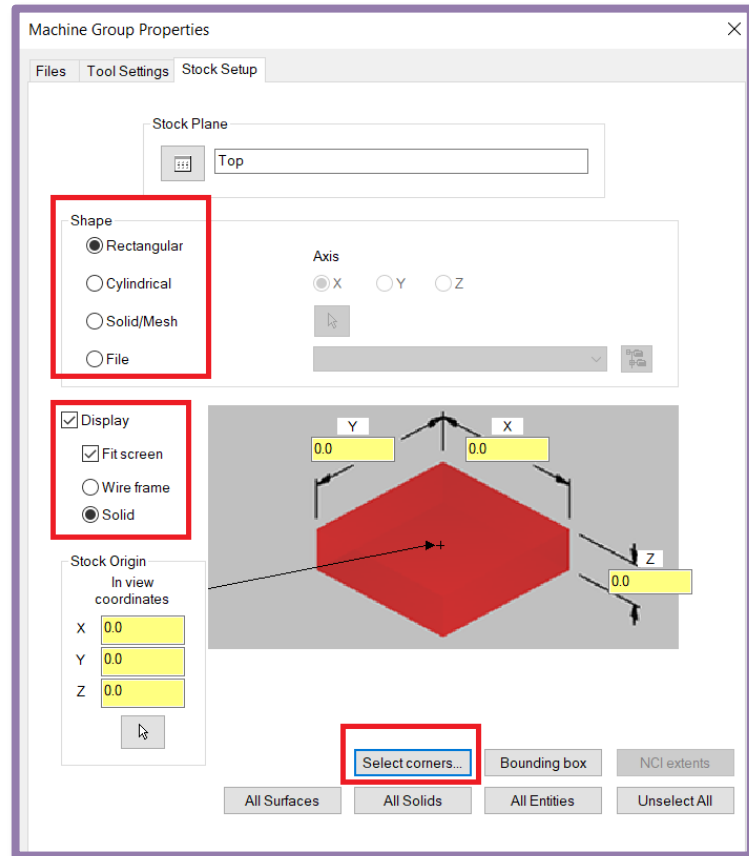
Warn of duplicate tool numbers (The warning will be given when you enter the same number of two tools).

Override Defaults with Modal values enables the system to keep the values that you enter.

Search tool library when entering a tool number (the tool library allows you to select from existing tools in a library as well as define new tools. Tool definitions can be saved in a library or just for the part you work on).

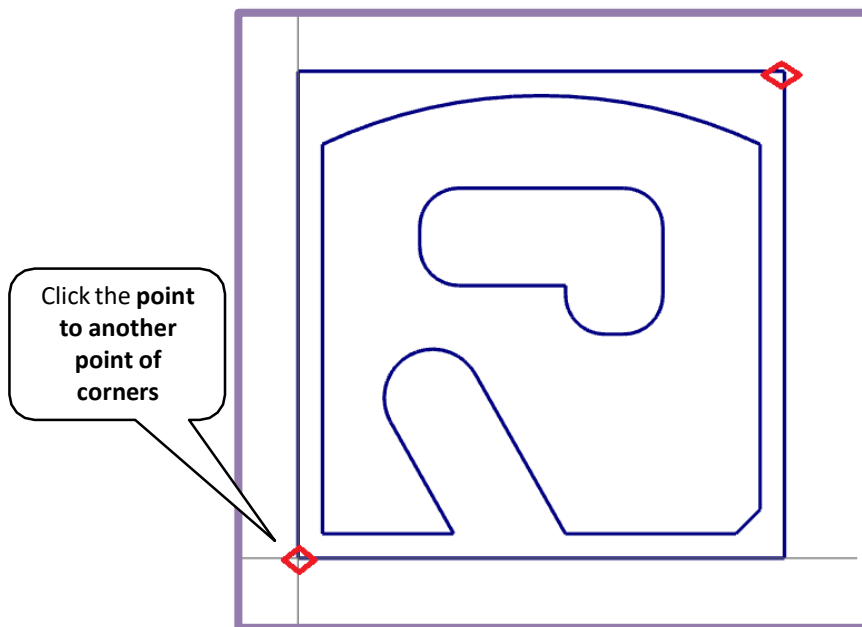
5

- In **Machine Group Properties**, choose the **Stock Setup** tab.
- Set the shape of stock in **Rectangular**.
- Click the **Select corners**



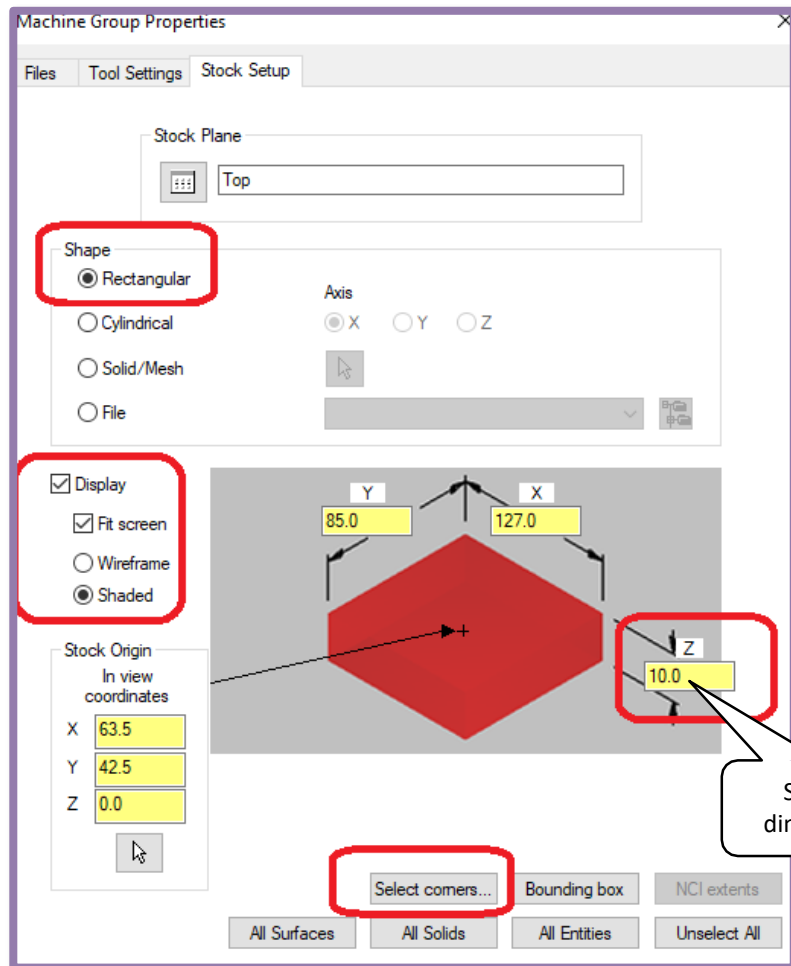
6

- Click the **point to another point of corners**. (These points is the outer boundary of stock).



7

- Enter the dimension of **Z : 10.0** as shown in Figure.



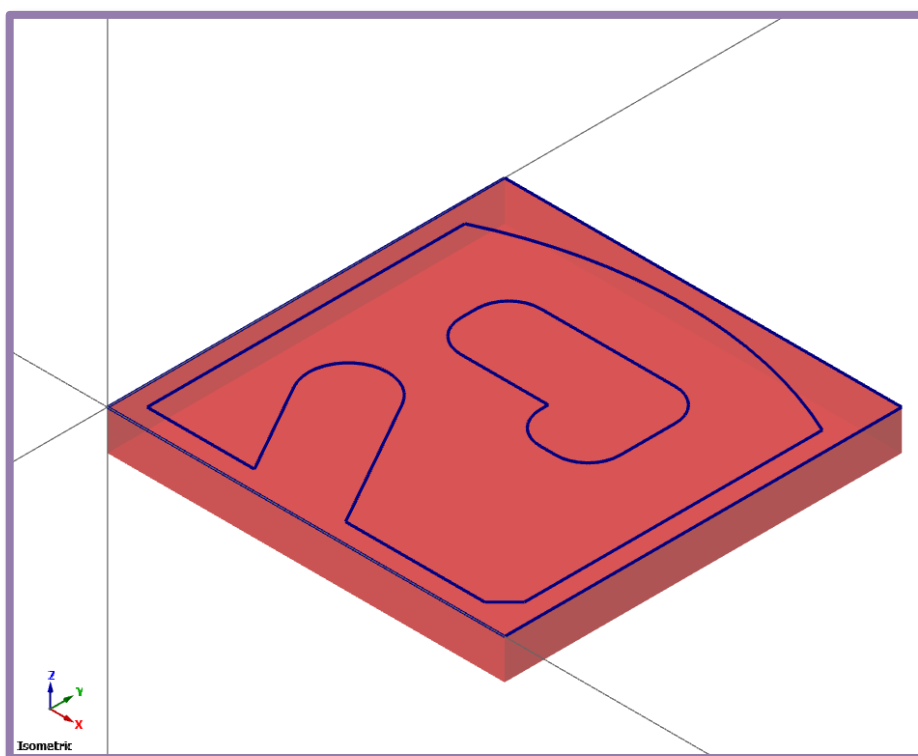
8

- Enter the dimension of **Z:10.0** as shown in Figure.

9

- Click the **OK** button to exit **Machine Group Properties**.
- Select **Isometric** view from the View Toolbar to see the part of the isometric.

The stock model will appear as shown in Figure.



Note: You can display the part of geometry or the toolpath that you have created in the stock model. Used a features backplot, or while verifying toolpaths to showed all the displays.

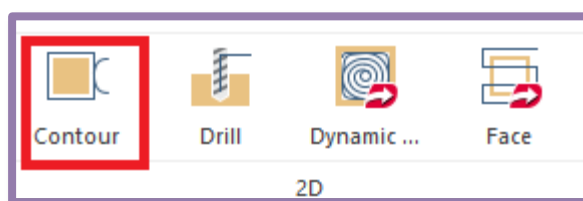


STEP 3: MACHINE THE TWO STEPS USING OPEN CONTOUR TOOLPATH.

Contour toolpath is a 2D high Speed Dynamic milling toolpath. It is removing the material along direction of the tool axis. This path is defined by a loop or chain of the curves. Contouring is removed an enclosed area and follows a chain only.

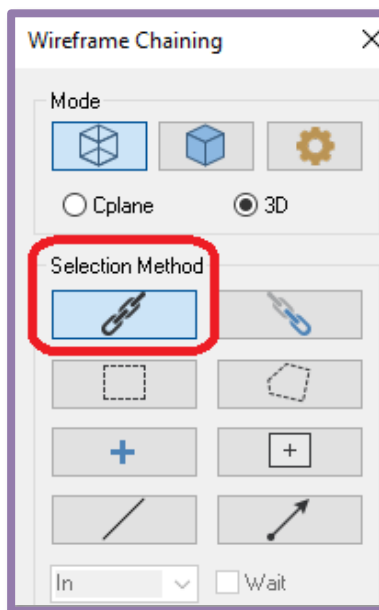
1

- From Toolbar click “**Toolpaths**”.
- Click “**Contour Toolpath 2D**”.
- **Select OK** button.



2

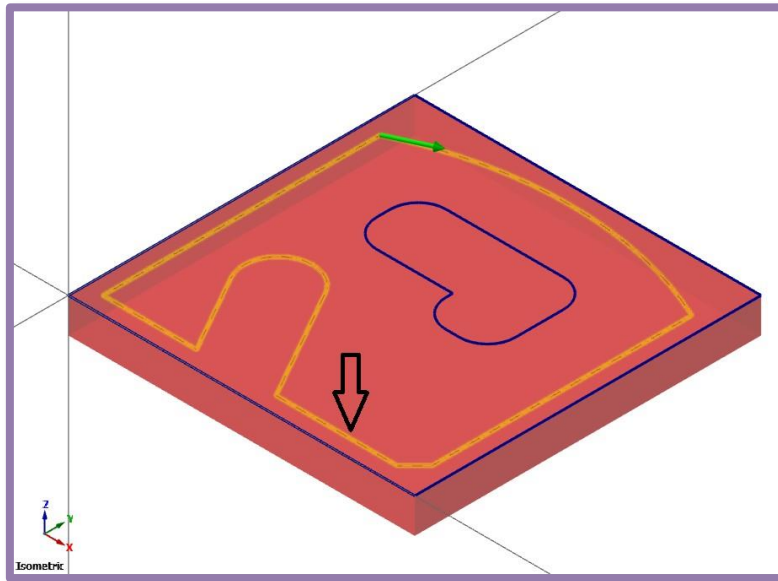
- Enable the Chain button in the chaining dialog box to be able to select just the outside contour.



Note: A chain of entities which is are made up of one or more paths or entities. The length has to be same or less than the chaining tolerance 0.002mm (in between the endpoints of two consecutive entities or path). The entities must be together in direction and order

3

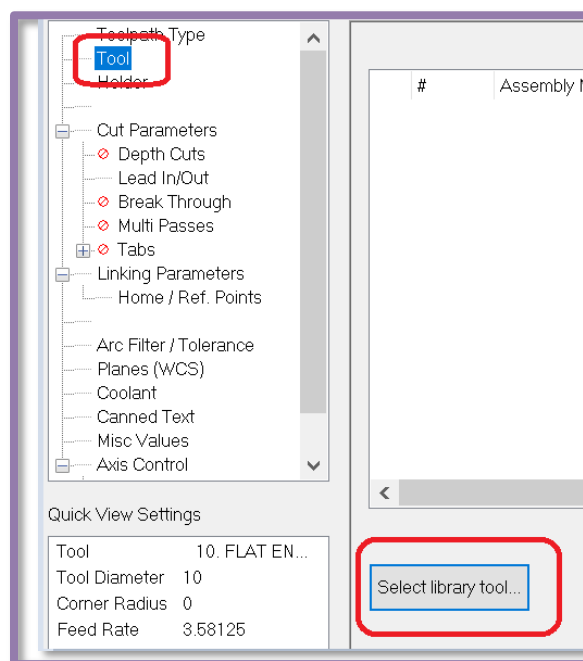
- Select the first chain as shown in the Figure.
- Click “OK” button to exit **Chaining**.
- After that, the toolbar of “Contour (2D)” will come out.



Note: When you are modify the pages, the Mastercam will updates the them (in the Tree View).

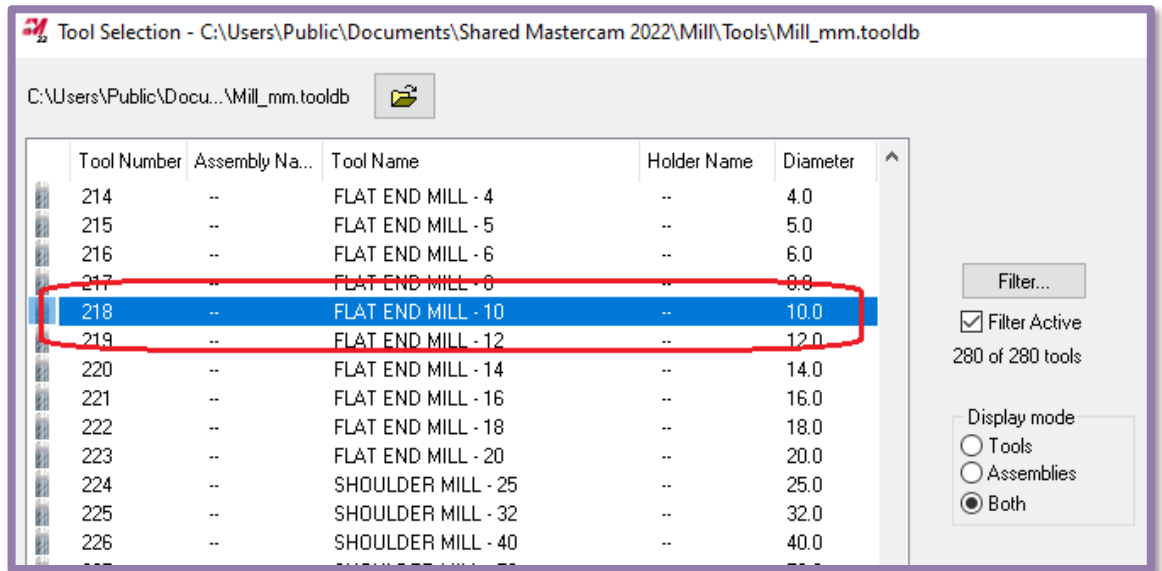
4

- Select Tool from the Tree View List.
- Click on the **Select Library Tool** button.



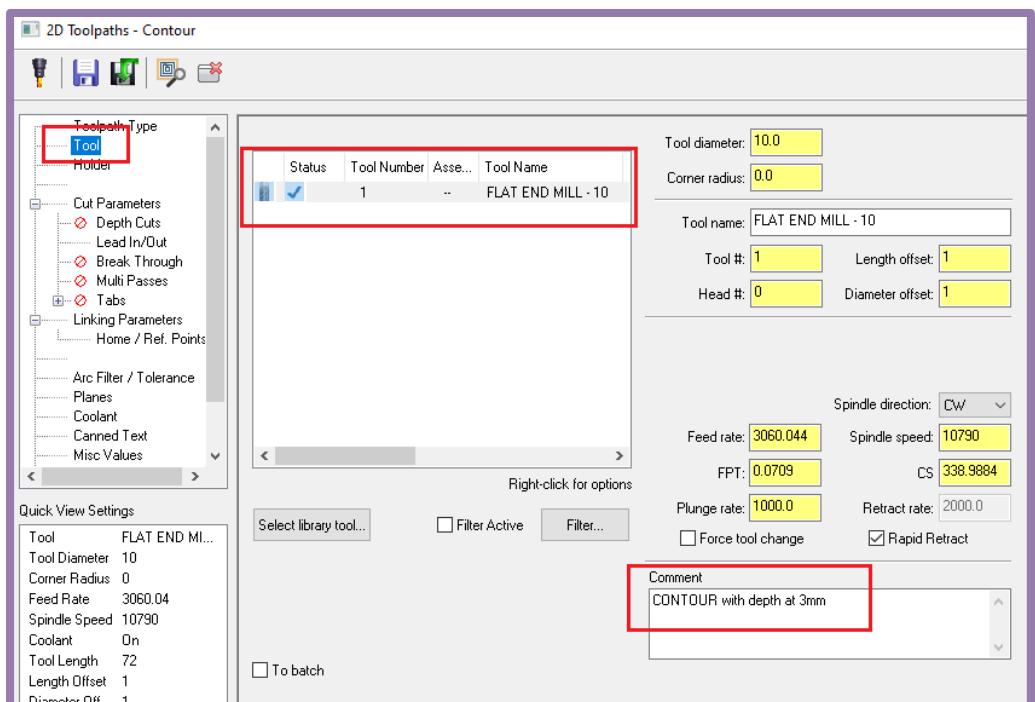
5

- Click **Select library tool** button to choose tool size.
- Choose tool size **Flat Endmill diameter 10.00 mm** as shown in Figure



6

- Select the Tool Selection page and write comment of the process
- Make all the necessary changes as shown in Figure.
- Then select the **OK** button to exit.

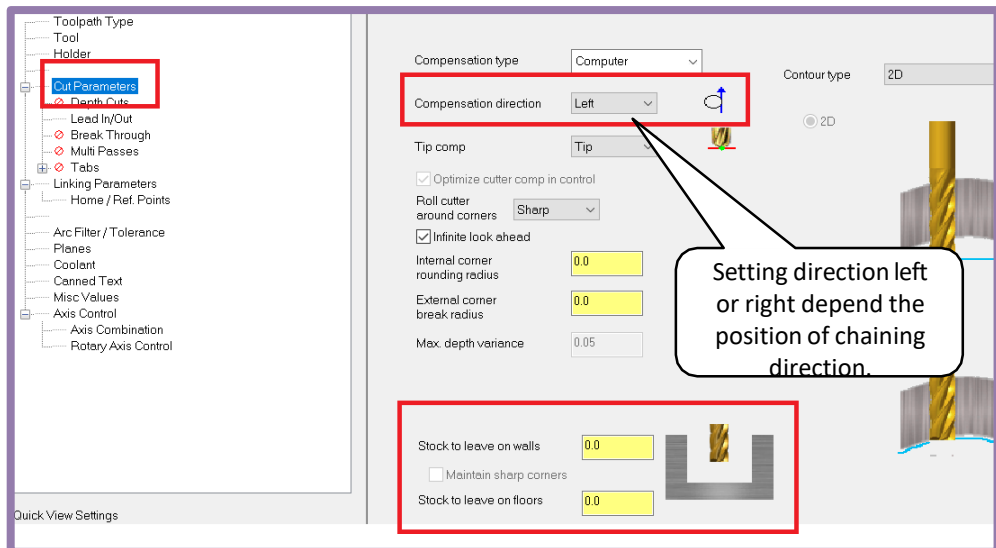


The **Feed rate, Plunge rate, Retract rate** and **Spindle speed** are roughly based on the part material Aluminium and HSS tooling. The tool parts and material can be choose to change in the program.

In the **Comment part** is to help you to remark the toolpath as an identity for **The Toolpaths/ Operation Manager**. Please refer the figure below.

7

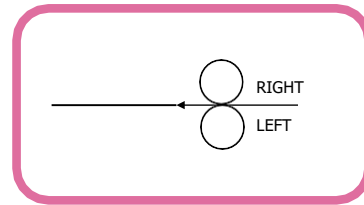
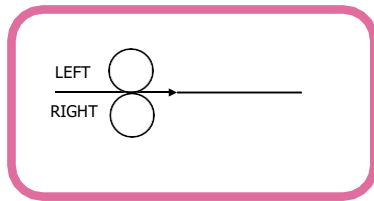
- From the Contour Parameter, select compensation director to set left or right **depend the position of chaining direction**.



8

- From the Contour Parameter, select compensation director to set left or right **depend the position of chaining direction**.

Note: Compensation Direction is to set the chaining direction. You can set the direction to offset either left or right depending on the location of the cutting/thread point entity outside or inside the contour.



9

- Select the Linking Parameter
- Determine the parameters of depth : **3mm**
- Click the **OK** for exit the command purposes.

Toolpath Type
Tool
Holder

Cut Parameters
 Depth Cuts
 Lead In/Out
 Break Through
 Multi Passes

LINKING
 Linking Parameters
 Home / Ref. Points

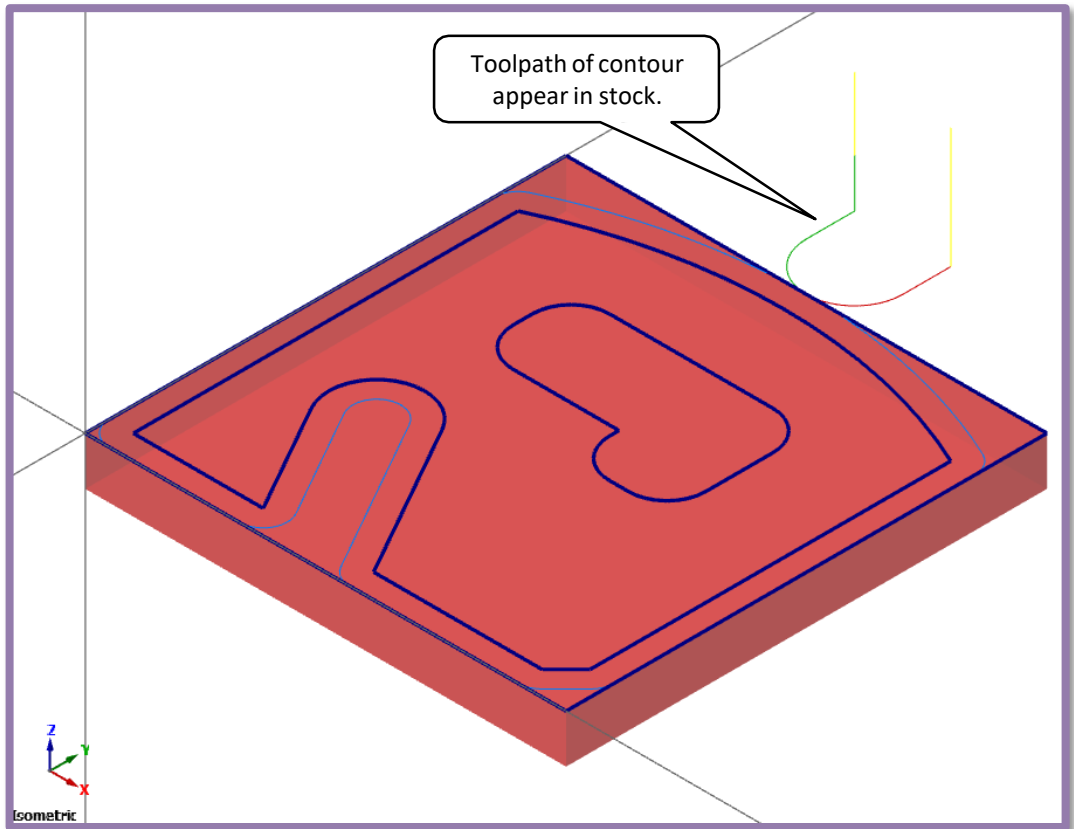
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Misc Values
Axis Control
 Axis Combination
 Rotary Axis Control

Quick View Settings
 Tool FLAT END MILL - 5
 Tool Diameter 5

Arc fit maximum radius 12.0
 Output feed move 13000.0
 Clearance... 50.0
 Use clearance only at the start and end of operation
 Retract... 25.0
 Feed plane... 10.0
 Top of stock... 0.0
 Depth... -3.0
 Absolute
 Incremental
 Associative

10

- Click the **OK** button to exit **2D Toolpath- Contour** parameter.
- Select **Isometric** view from the View Toolbar to see the part of the isometric. The toolpath of contour in will appear as shown in Figure.



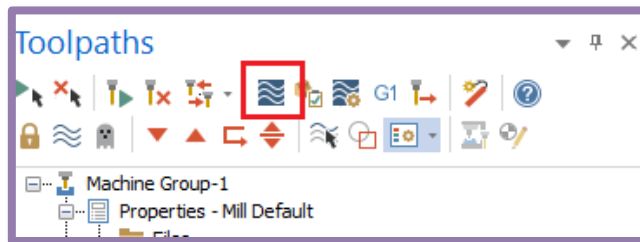


STEP 4 : BACKPLOT THE TOOLPATHS

Backplotting shows the toolpath of cutting tool to cut the part. This display will let you to spot an errors when you run the mode.. You can define it in the program before you machining the parts. MasterCam will displays the coordinates of X, Y and Z axis in the screen when you backplot the paths.

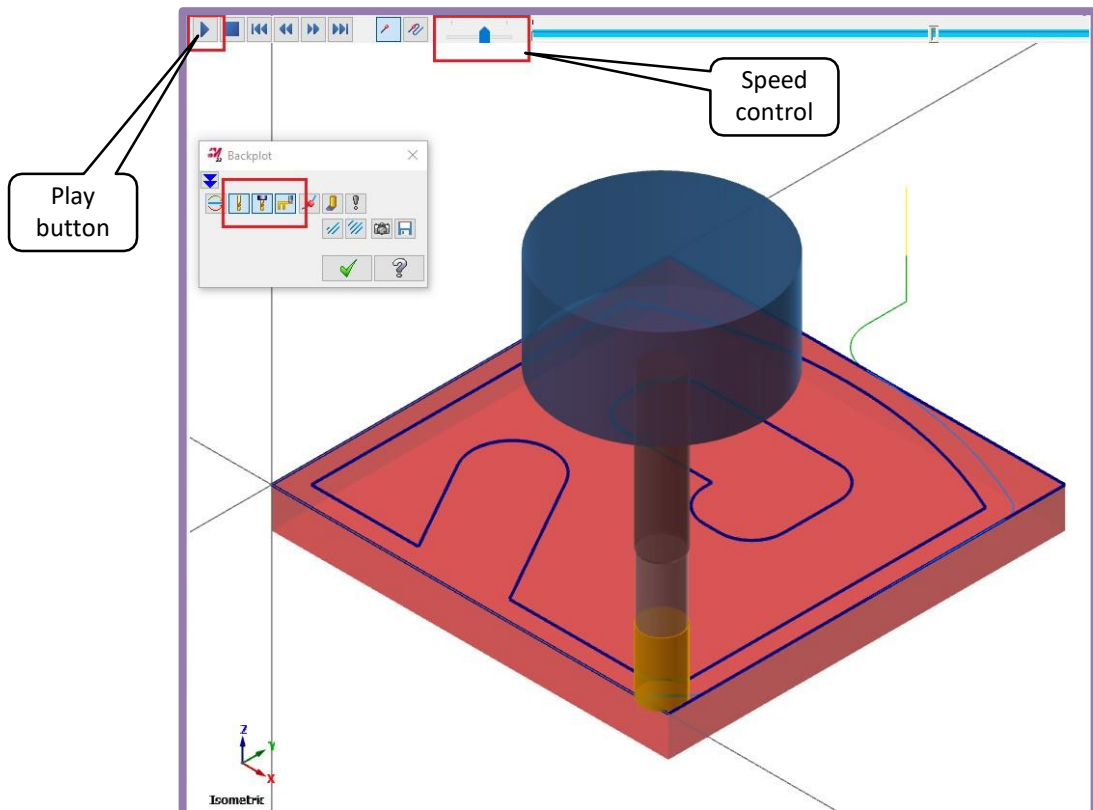
1

- On the operations button select the **Backplot**.



2

- Turn on all the buttons (to **display tool, holder and rapid moves**).
- Adjust the speed of the backplot.
- Click a **Play** button.
- Click **Ok** to exit the Backplot



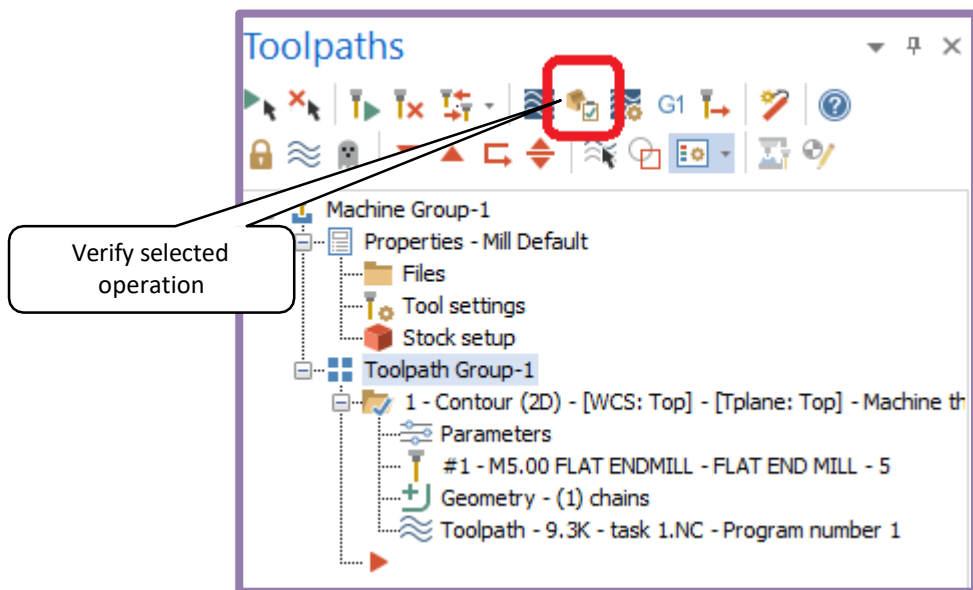


STEP 5 : VERIFY THE TOOLPATHS

Verify allows you to simulate the machining of a part by using as solid model. It is simulation graphic of movement the cutting tool and the material removing pass by pass. It also can give a reminder of collisions between tool and stock if any happened.

1

- From **Operation Manager**, choose **Verify selected operations** icon as shown in Figure.

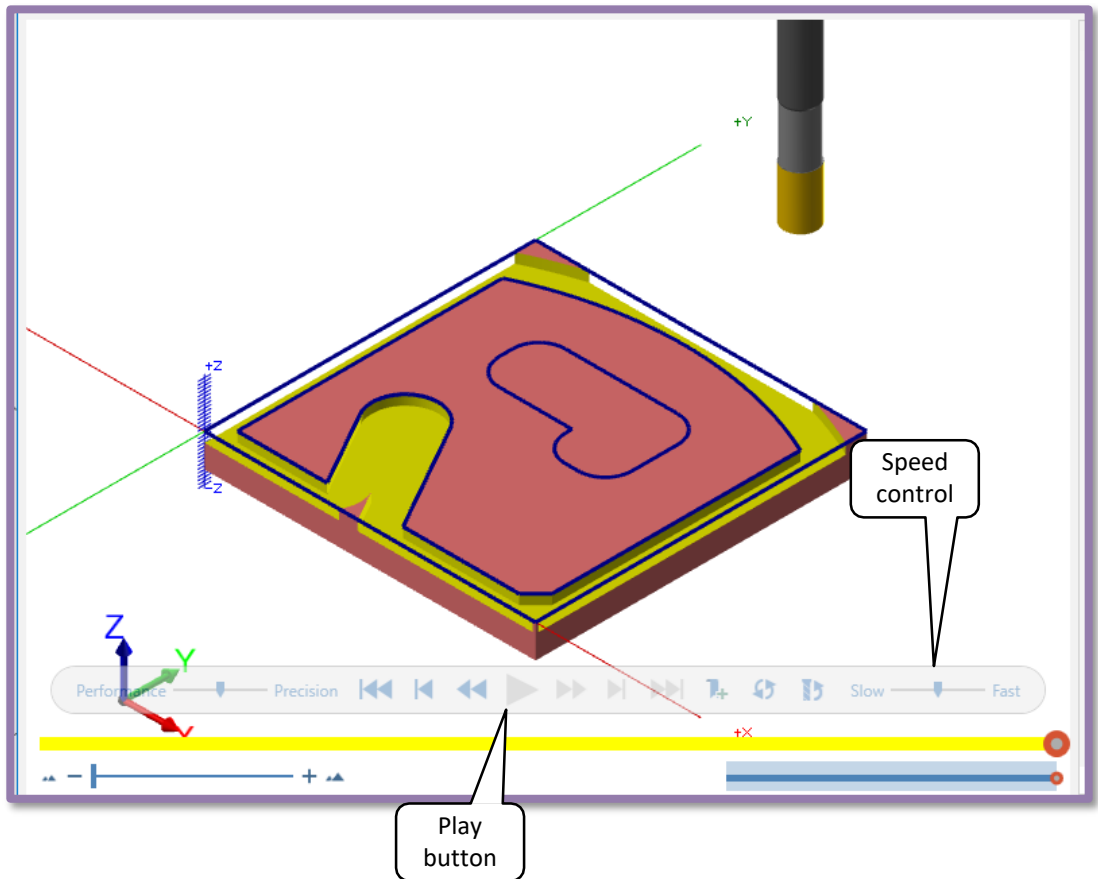


2

- The MasterCam Simulator window will pop up as shown in the figure.
- To start simulation select the Play button.
- The simulation will be played.

3

- The speed control can be adjusted as shown in the figure.



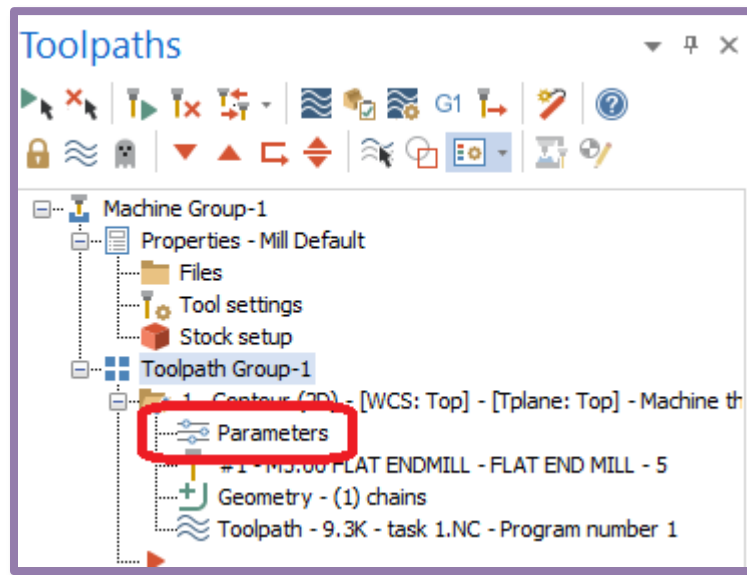


STEP 6: ADDING MULTI PASSES OF TOOLPATH

Multi Passes lets you make more than one passes of cutting tool in your toolpath. It is for clearance purposes (to remove material as you desired). The tool proposed the multiple cutting passes for the **roughing** and **finishing** passes.

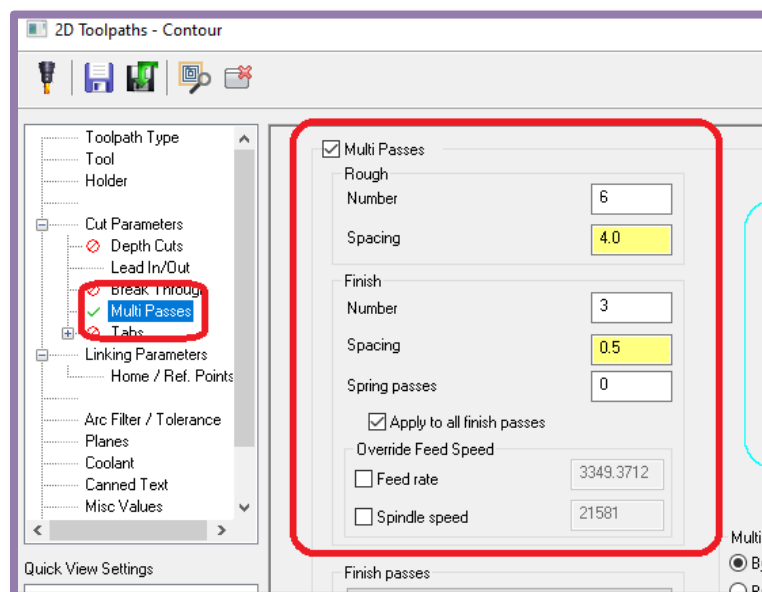
1

- Select **Parameter** to open ToolPath Setting



2

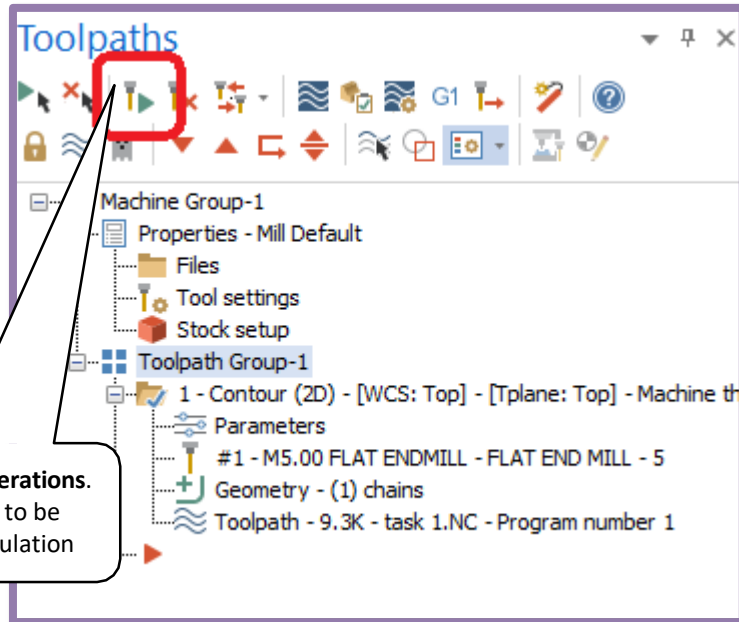
- Select **Multi Passes** from the **Tree view list**
- Select the necessary changes.
- Enter a value for the **Number of passes**.
- Click **OK** to exit the **Multi Passes** parameters.



3

- Select the button of “**Regenerate all dirty operations**”.
- This feature will regenerate the modification that you have made to the toolpaths.

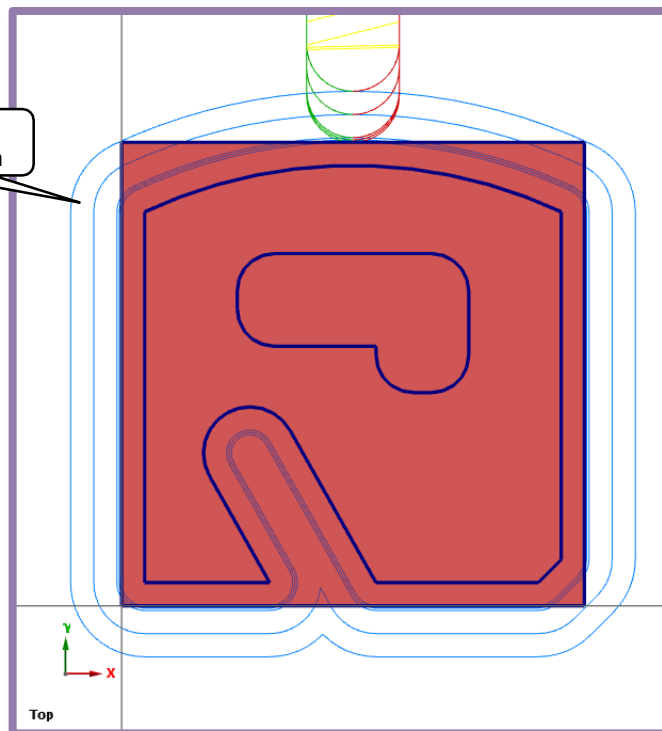
Regenerate all dirty operations.
The operations have to be generated before simulation



4

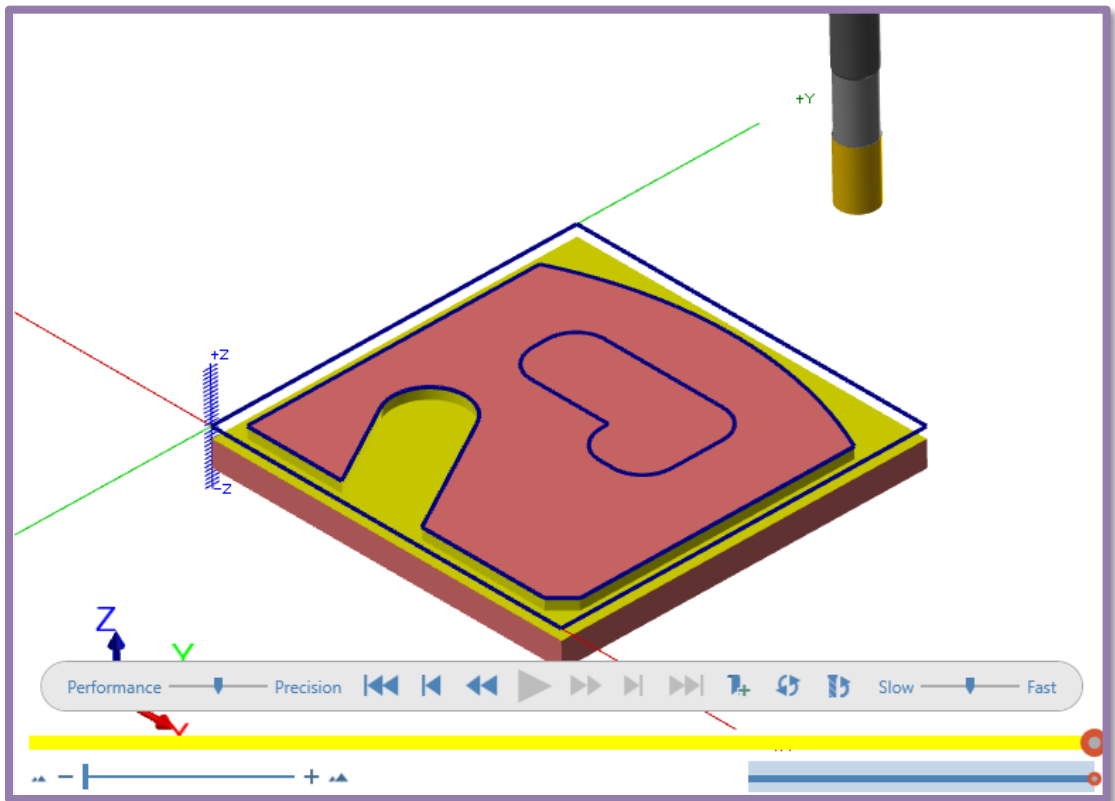
- Select **Isometric** view from the View Toolbar.
- The multi passes of contour toolpath will appear as shown in Figure.

Multi passes of contour toolpath



5

- To **Backplot** and **Verify** the toolpaths, please refer back the topic to review these procedures.



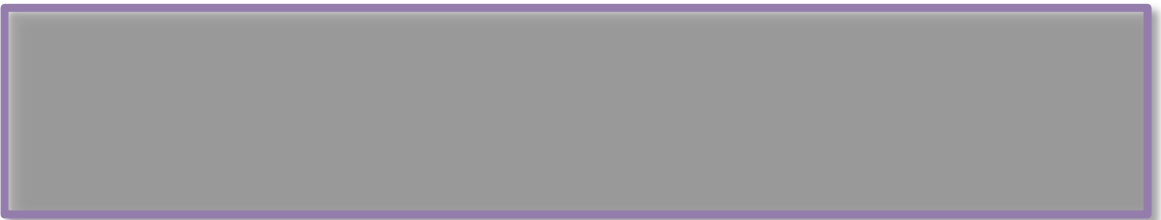


STEP 7 : MACHINING THE STOCK USING 2D POCKETING

Note: Pocket toolpaths is removed the part of material from an enclosed boundary.

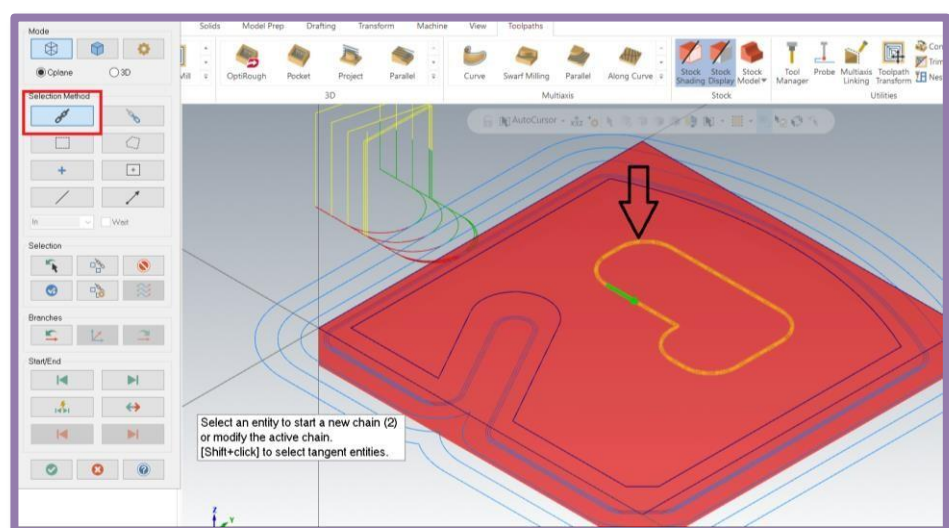
1

- From “**Toolbar**” Click “**Toolpaths**”.
- Click “**Pocket Toolpath**”.



2

- Enable the **Chain** button in the chaining dialog box
- Select the path of the pocket.
- Select the chain as shown in Figure.



3

- Setting **Toolpath** parameters:
- Click **Select library tool** button to choose tool size
- Disable **Filter Active** to able to see all the tools from the library.
- Select tool size **Flat Endmill diameter 8.00 mm** as shown **Figure**.

C:\Users\Public\Docume...\Mill_mm.tooldb

#	Tool Name	Dia.	Cor. r...	Length	# Flut...	Type	Rad...
222	FLAT END MILL - 18	18.0	0.0	29.0	4	Flat...	None
223	FLAT END MILL - 20	20.0	0.0	32.0	4	Flat...	None
213	FLAT END MILL - 3	3.0	0.0	5.0	4	Flat...	None
214	FLAT END MILL - 4	4.0	0.0	7.0	4	Flat...	None
215	FLAT END MILL - 5	5.0	0.0	8.0	4	Flat...	None
216	FLAT END MILL - 6	6.0	0.0	10.0	4	Flat...	None
217	FLAT END MILL - 8	8.0	0.0	13.0	4	Flat...	None
206	FORMING THREAD TAP - GTX 2.31	33.249-2.31	0.0	26.0	1	High...	None

4

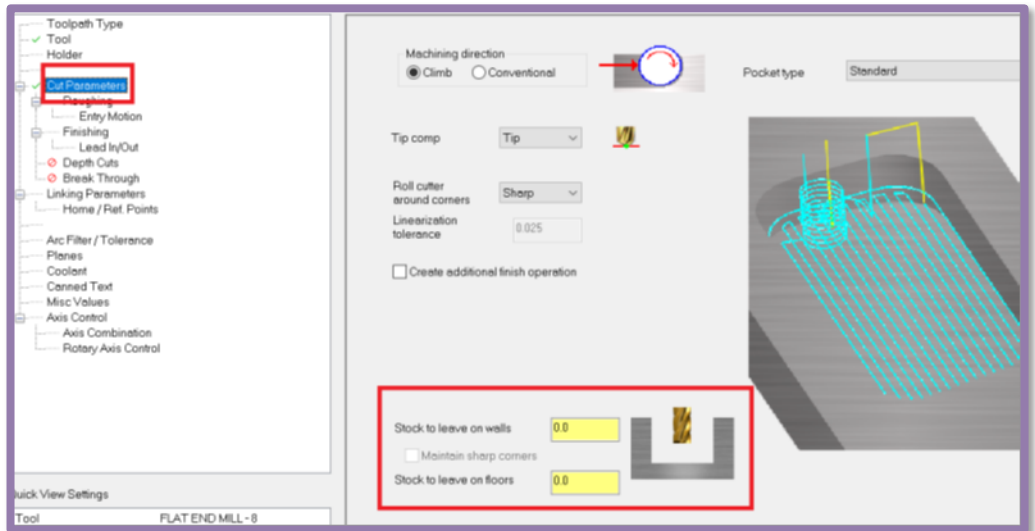
- Select the **Tool Selection** page
- Write the comment of the process.
- Choose all the necessary thing as shown in **Figure**.

Tool Selection page parameters:

- Tool diameter: 8.0
- Corner radius: 0.0
- Tool name: FLAT END MILL - 8
- Tool #: 2
- Length offset: 2
- Head #: 0
- Diameter offset: 2
- RCTF
- Spindle direction: CW
- Feed rate: 2980.0768
- Spindle speed: 13448
- FPM: 0.0554
- CS: 337.9956
- Plunge rate: 1000.0
- Retract rate: 2000.0
- Force tool change
- Rapid Retract
- Comment: Cut the pocket in the center of the part

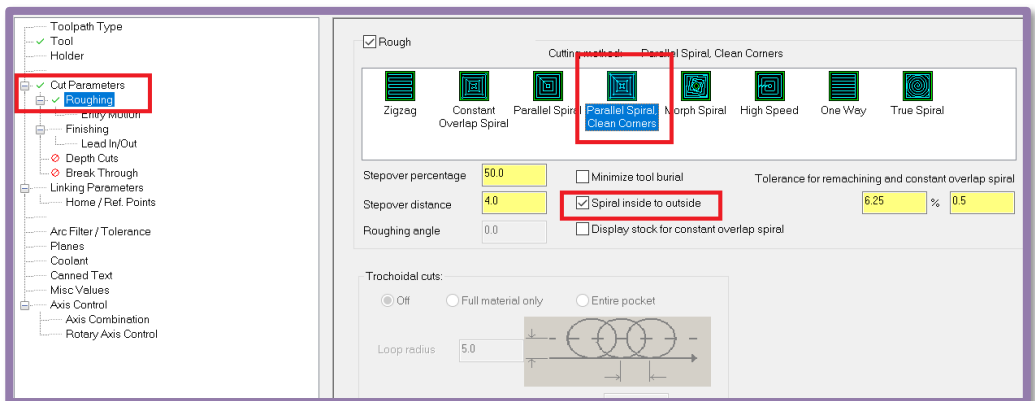
5

- From the **Tree view list**, select **Cut Parameters**. Change the necessary settings as shown in Figure.



6

- From the **Tree view list**, select **Roughing Parameters**. Choose the option **Parallel Spiral Clean Corners** and change the necessary settings as shown in Figure.

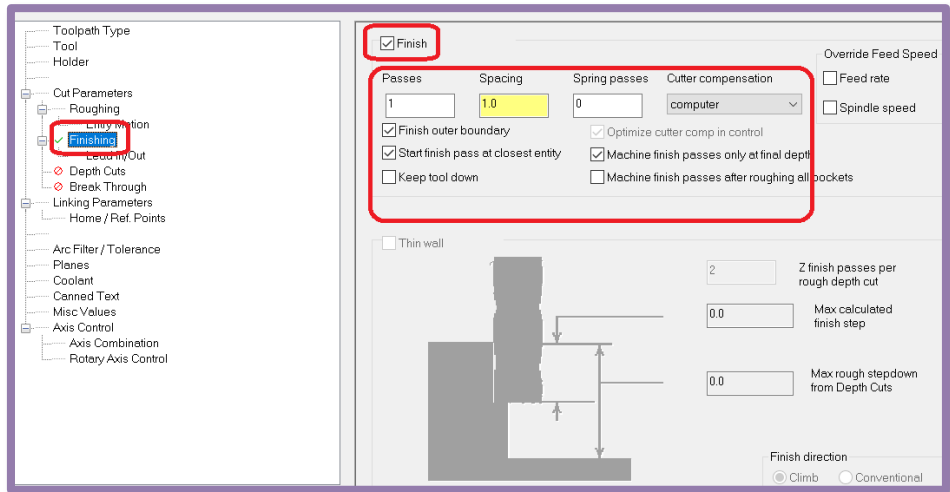


Parallel spiral roughs out the pocket in a similar method to parallel spiral but adds small clean out moves in the corners of the pocket to remove more stock.

Spiral Inside to Outside applies to all spiral pocket toolpaths. The toolpaths spiral from the center to the wall of the pocket.

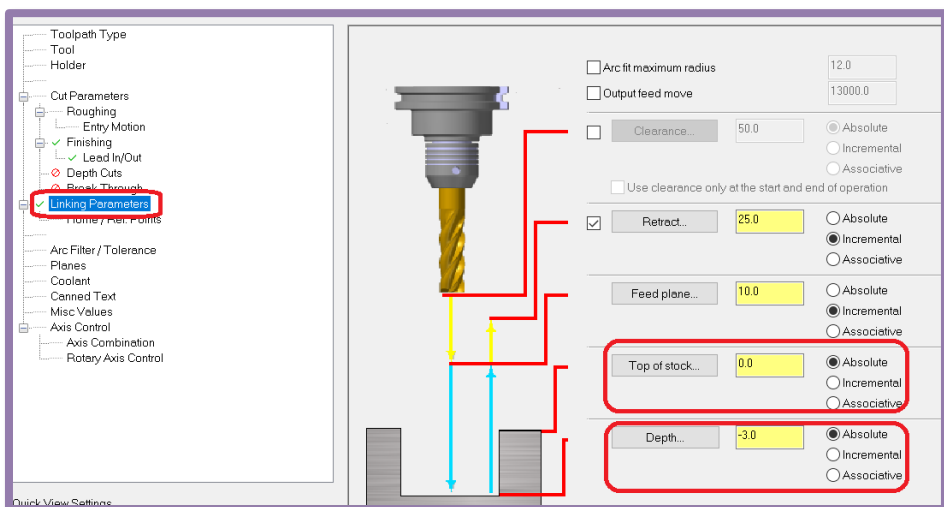
7

- From the **Tree view list**, select **Finishing (enabled)**.
- This will finish the part after roughing process.
- Choose enabled the necessary settings as shown in Figure.



8

- Select **Linking Parameters** from the **Tree View list**.
- Change the **Top of stock** to zero and set the depth to **-3.0**.
- Ensure all the values are set to **Absolute**.
- Select the **OK** button.

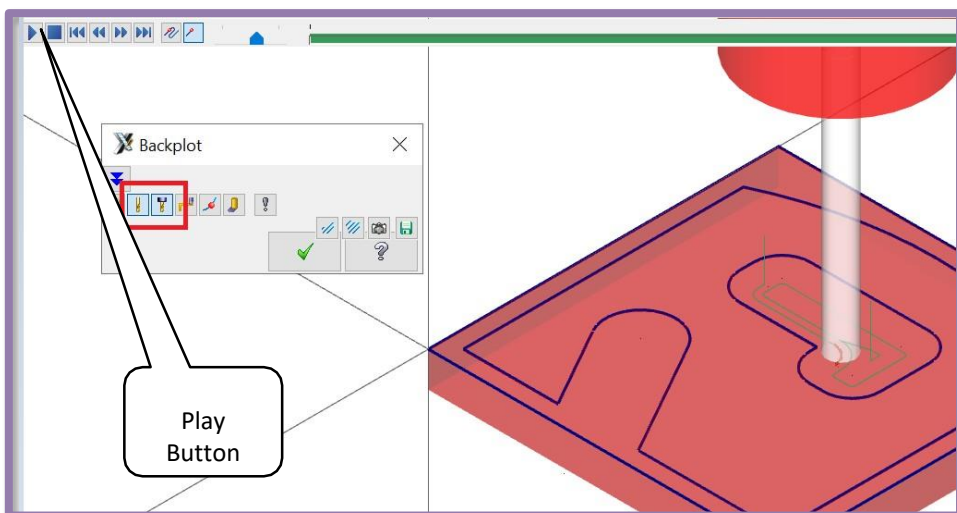
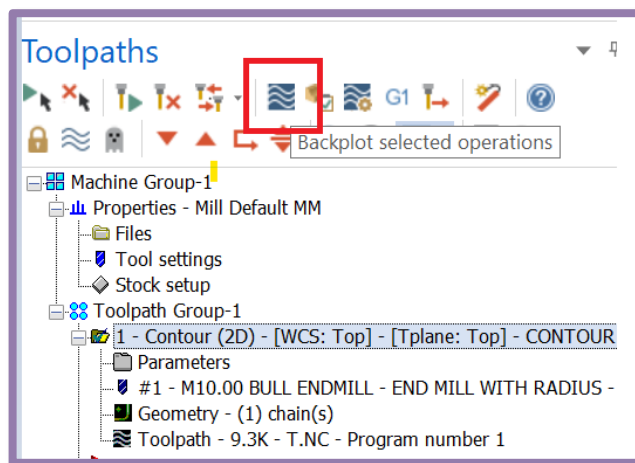




STEP 8 : BACKPLOT THE TOOLPATHS

Backplotting shows the toolpath of cutting tool to cut the part. This display will let you spot an error. You can define it in the program before you machining the parts. The current X, Y and Z coordinate will be display in the screen when you backplot the paths.

- 1
 - On the operations button select the **Backplot**.
 - Turn on all the buttons .(the cutting tool will appear pushed down). You can see the tool is running and follow the chain of toolpath.
 - Adjust the speed of the backplot. Click a **Play** button.



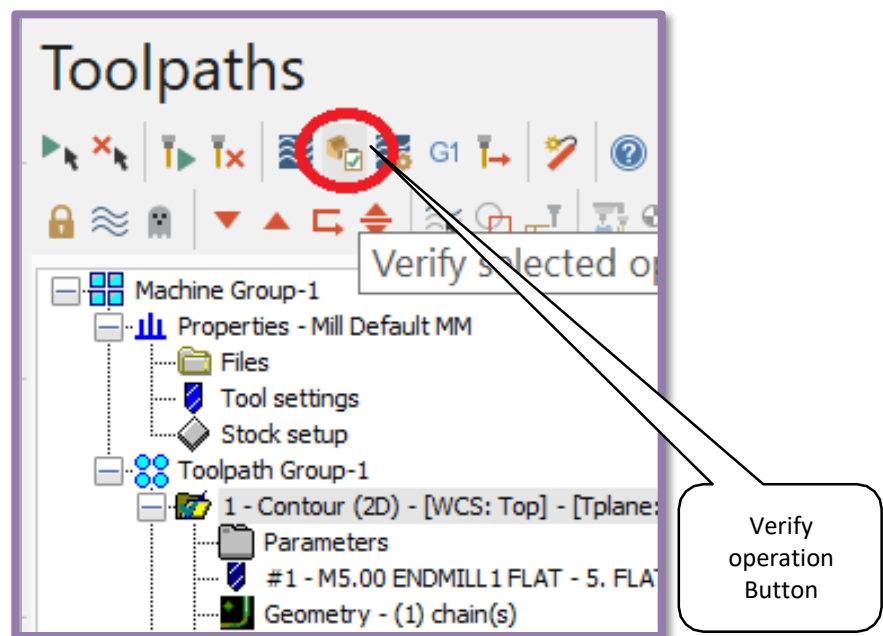


STEP 9 : VERIFY THE TOOLPATH

Verify allows you to simulate the machining of a part by using as solid model. The surface polishing and roughing are represented by the solid model produced during verification. Additionally, it displays collisions, if any.

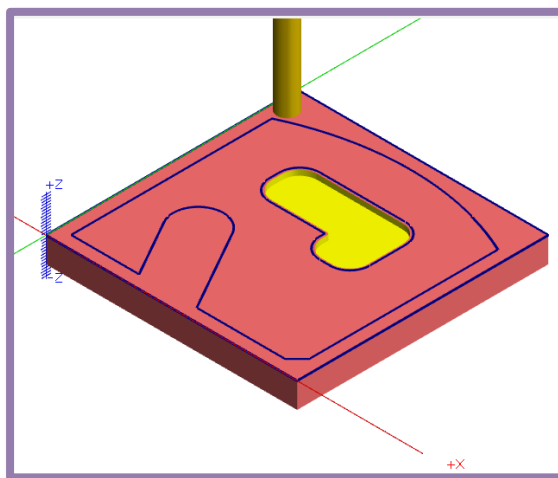
1

- From **Operation Manager**, choose **Verify selected operations** icon as shown in Figure.



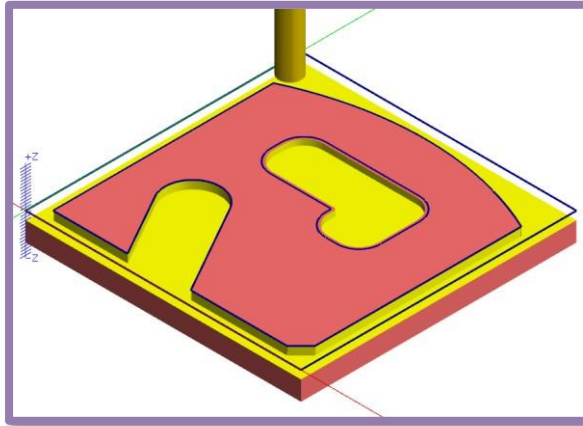
2

- To start simulation, select the **Play** button.
- The simulation will be shown in window.



3

- After play the Verify, the simulation of contouring and pocketing will be shown in the figure.



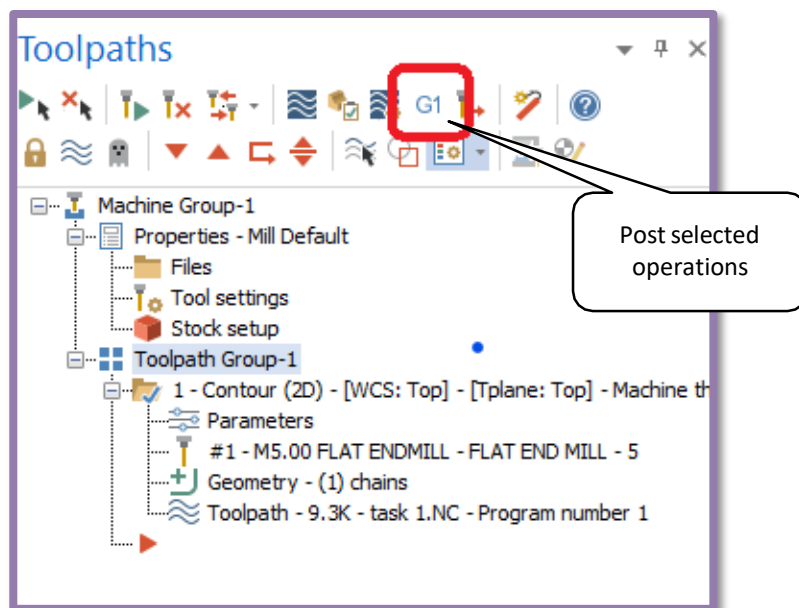


STEP 10 : POST THE FILE

Post processing, or **posting a program**, it is refers to convert the process of toolpaths in your Mastercam part files to a format that can be understood by your CNC machine tool's control. For example, G-codes and M-codes.

1

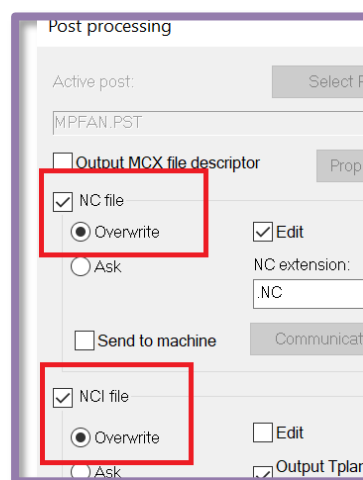
- Click the **Post selected operation** button in the Operation Manager.
- Make the necessary changes as shown in **Post Processing** window in the figure above.



- Note: Make sure all the operations are selected. **Select Overwrite** in the **Operations Manager**.

2

- Set the **Post Processing** like the Figure.
- Select **OK** button to continue.



3

- After generates NC codes, copy all the codes and save in as **Notepad** version.
- You can use this post in the **NC code** at CNC machine.

```
2 O0001(T)
3 (DATE=DD-MM-YY - 13-08-21 TIME=HH:MM -
4 (MCX FILE - C:\USERS\LENOVO\DOCUMENTS\P
5 (NC FILE - C:\USERS\LENOVO\DOCUMENTS\MY
6 (MATERIAL - ALUMINUM MM - 2024)
7 ( T1 | 5. FLAT ENDMILL | H1 )
8 N100 G21
9 N110 G0 G17 G40 G49 G80 G90
10 ( CONTOUR THE PART DEPTH 3.0MM )
11 N120 T1 M6
12 N130 G0 G90 G54 X-34. Y41. A0. S2000 M3
13 N140 G43 H1 Z25.
14 N150 Z10.
15 N160 G1 Z-3. F10.
16 N170 X-29.
17 N180 G3 X-24. Y46. I0. J5.
18 N190 G1 Y72.
19 N200 G2 X13. Y109. I37. J0.
20 N210 G1 X105.
21 N220 G2 X144. Y70. I0. J-39.
22 N230 G1 Y25.
23 N240 G2 X95. Y-24. I-49. J0.
24 N250 G1 X72.
25 N260 G2 X60. Y-21.401 I0. J29.
26 N270 X48. Y-24. I-12. J26.401
27 N280 G1 X20.
28 N290 G2 X-6.585 Y-6.585 I0. J29.
29 N300 X-24. Y20. I11.585 J26.585
30 N310 G1 Y46.
```



PRACTICAL TASK 3

Overview of making part in milling process :

1. The students will check the drafting and construct the part of geometry.
2. Once the geometry is complete, the student needs to set up the toolpath.
3. It is also included to set up the tool settings, stock size, contour or pocketing toolpath, and the display of geometry will be created by machine.

Instruction :

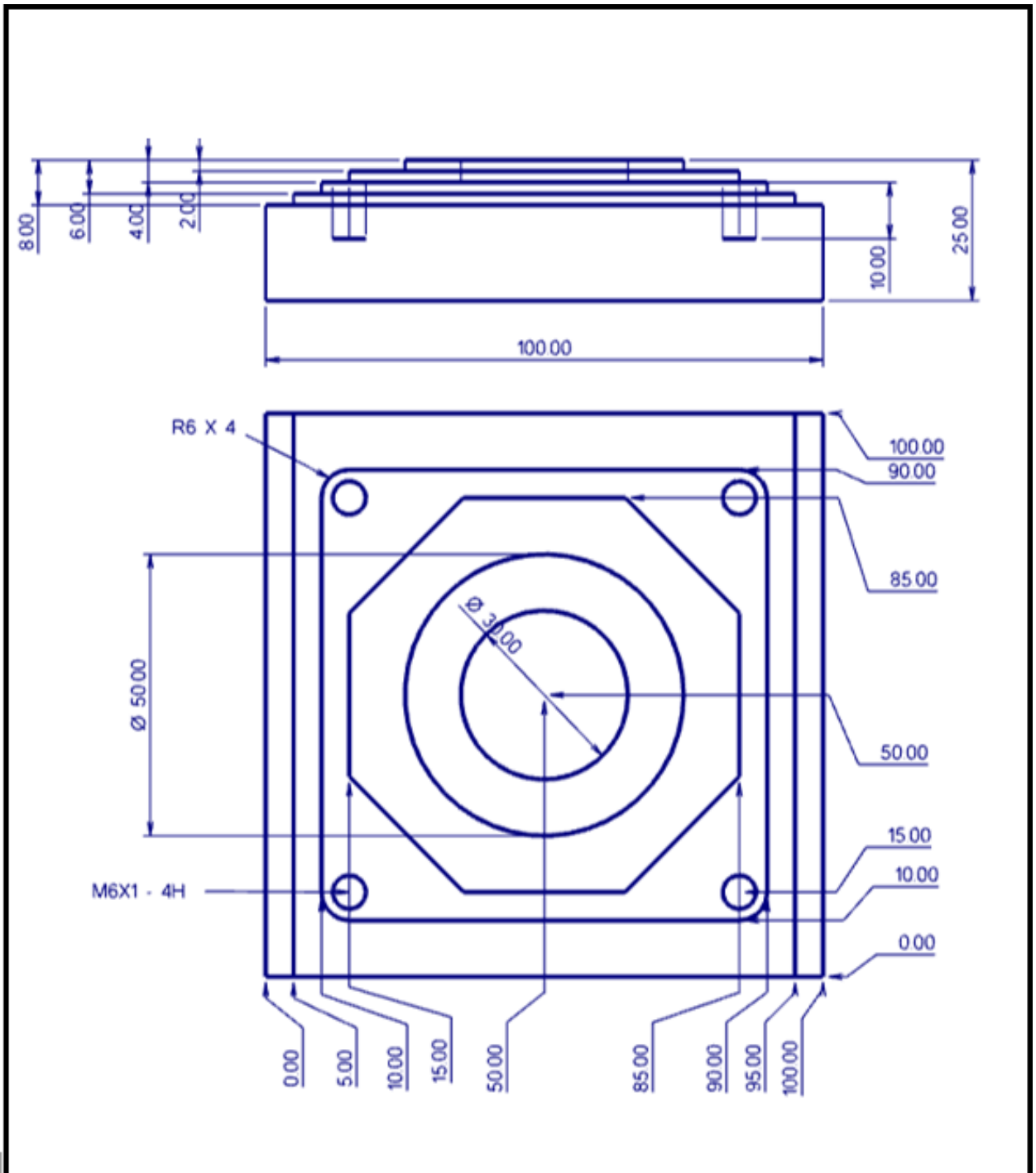
1. Draw a drafting task by using MasterCam software.
2. Set up the roughing, semi-finishing, and finishing processes, including parameters and tool selection.
3. Generate the NC codes from the task.

Scan the QR Codes:



Or select the link to watch the video how to construct the exercise Practical Task 3.

<https://drive.google.com/file/d/1TzjUSgkmBqVu6ITJnw1BKlygDRc9rsUK/view?usp=sharing>



TITLE	MILLING: PRACTICAL TASK 3	DEPT	MECHANICAL ENGINEERING
POLITEKNIK PORT DICKSON	COURSE	DJF41042 (CAD CAM)	
	MATERIAL	ALUMINIUM 6061	
	UNIT	METRIC (MM)	



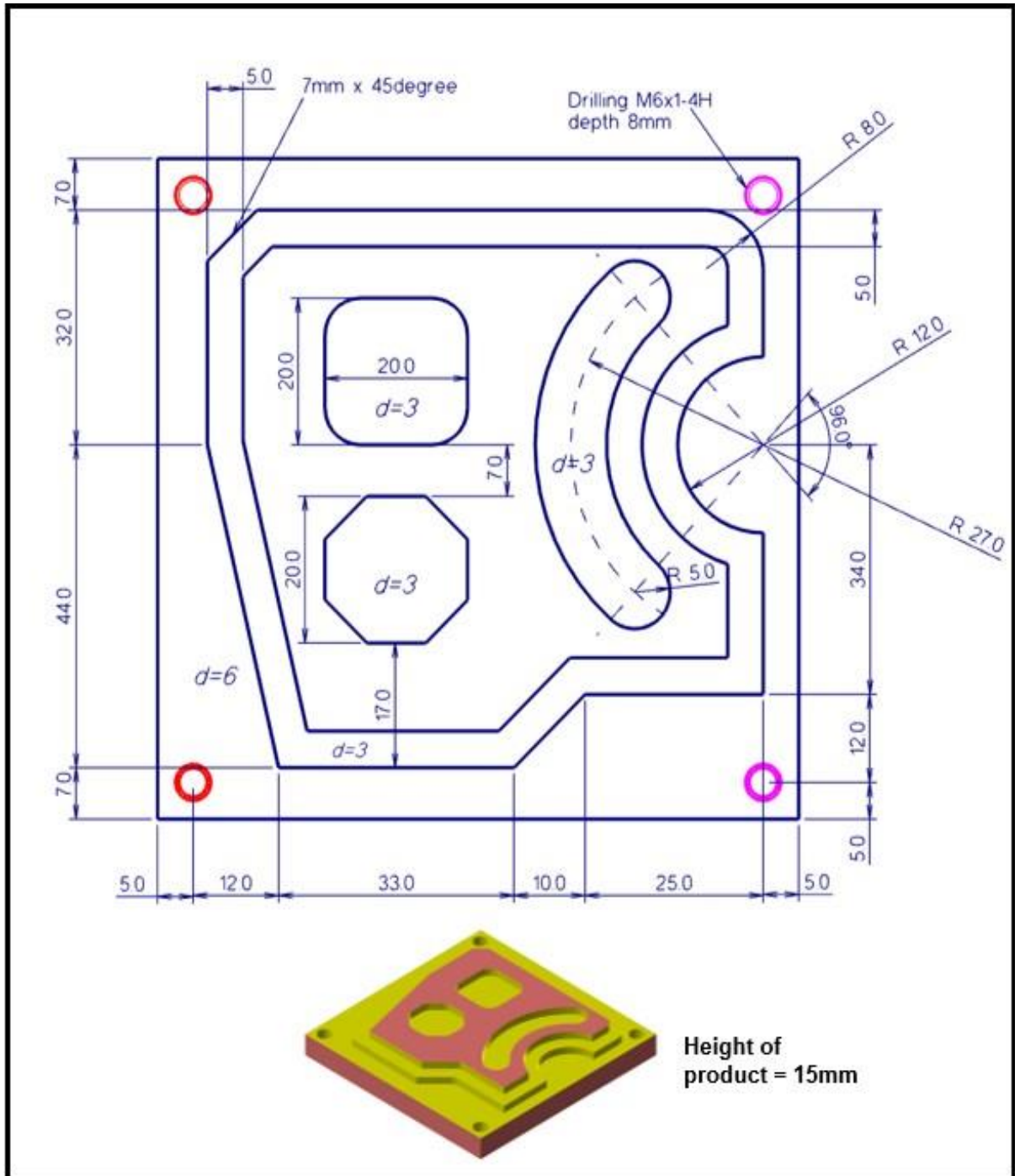
PRACTICAL TASK 4

Overview of making part in milling process :

1. The students will check the drafting and construct the part of geometry.
2. Once the geometry is complete, the student needs to set up the toolpath.
3. It is also included to set up the tool settings, stock size, contour or pocketing toolpath, and the display of geometry will be created by machine.

Instruction :

1. Draw a drafting task by using MasterCam software.
2. Set up the roughing, semi-finishing, and finishing processes, including parameters and tool selection.
3. Generate the NC codes from the task.

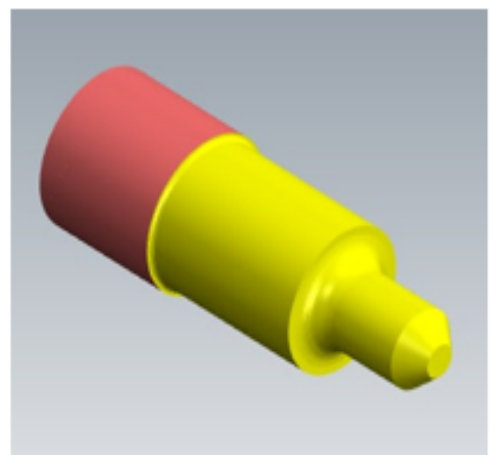
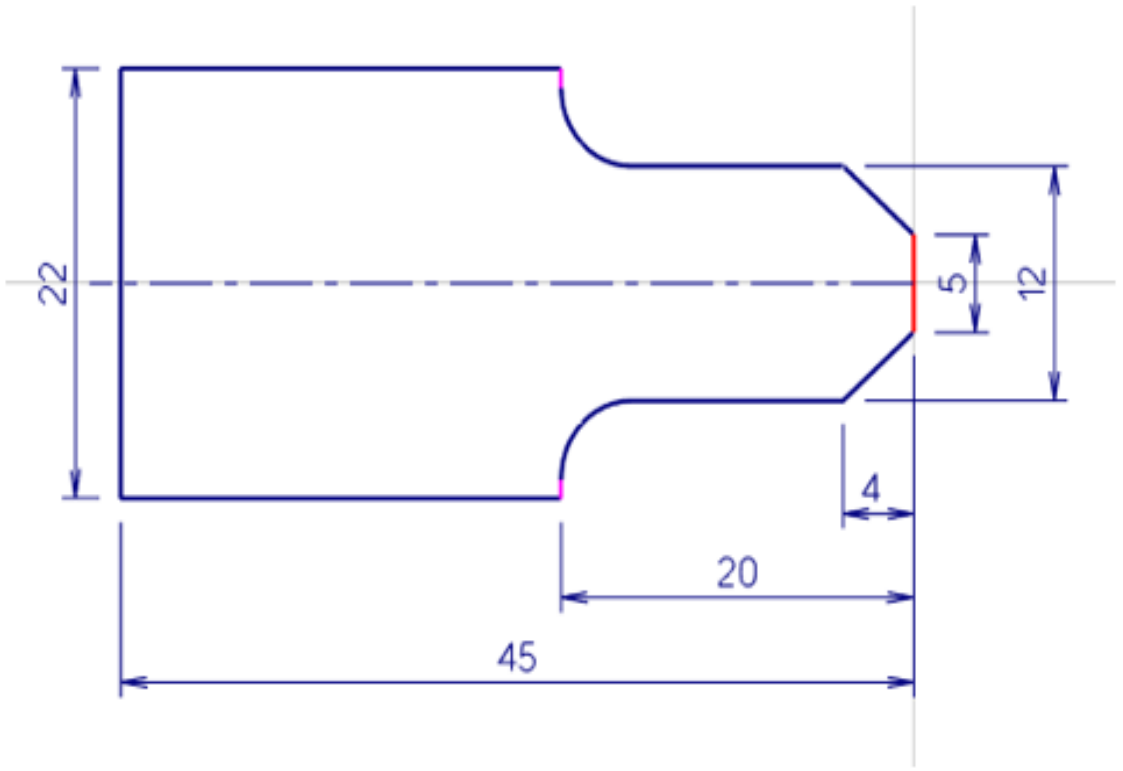


TITLE	MILLING: PRACTICAL TASK 4	DEPT	MECHANICAL ENGINEERING
POLITEKNIK PORT DICKSON	COURSE	DJF41042 (CAD CAM)	
	MATERIAL	ALUMINIUM 6061	
	UNIT	METRIC (MM)	



CHAPTER 3

TURNING PROCESS



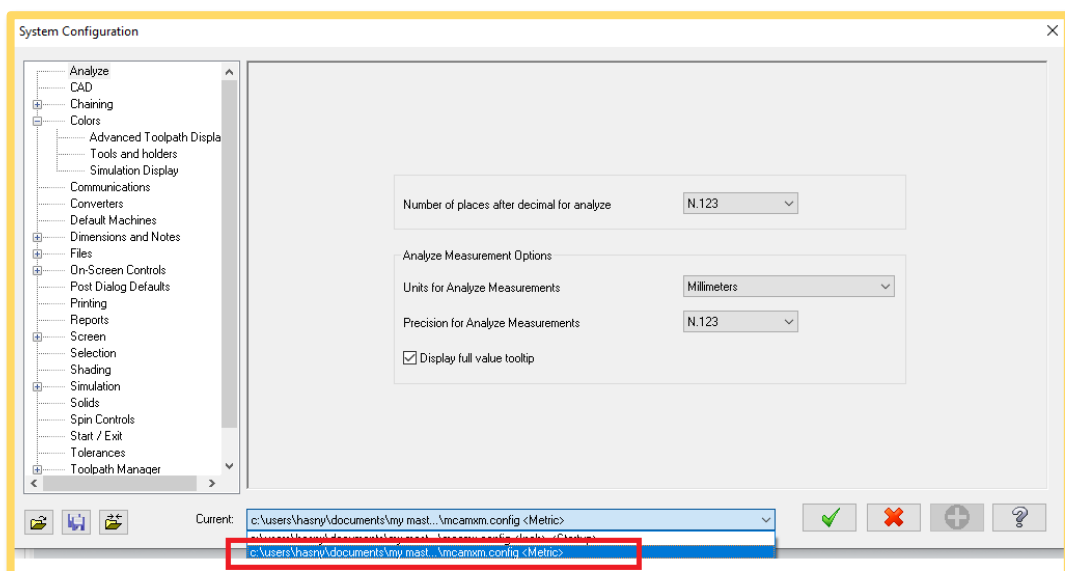
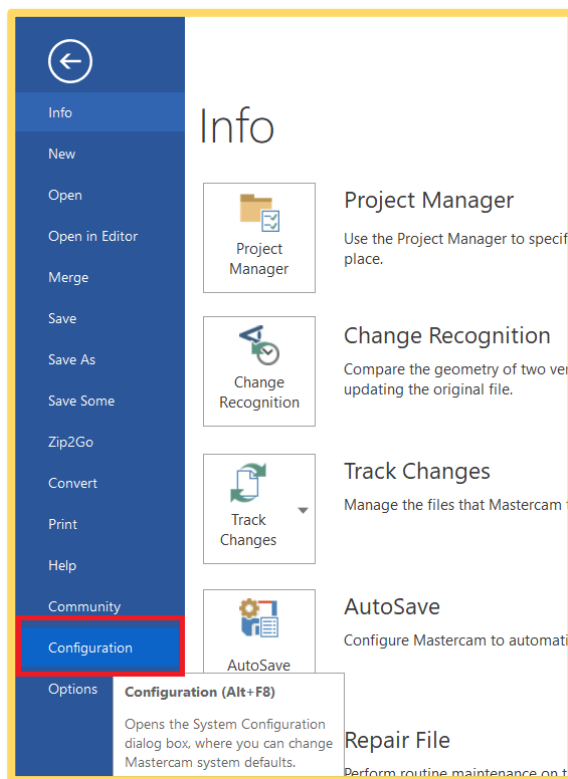
TITLE	TURNING: PRACTICAL TASK 1	DEPT	MECHANICAL ENGINEERING
POLITEKNIK PORT DICKSON		COURSE	DJF41042 (CAD CAM)
		MATERIAL	ALUMINIUM 6061
		UNIT	METRIC (MM)



STEP 1: CREATE A GEOMETRY

1

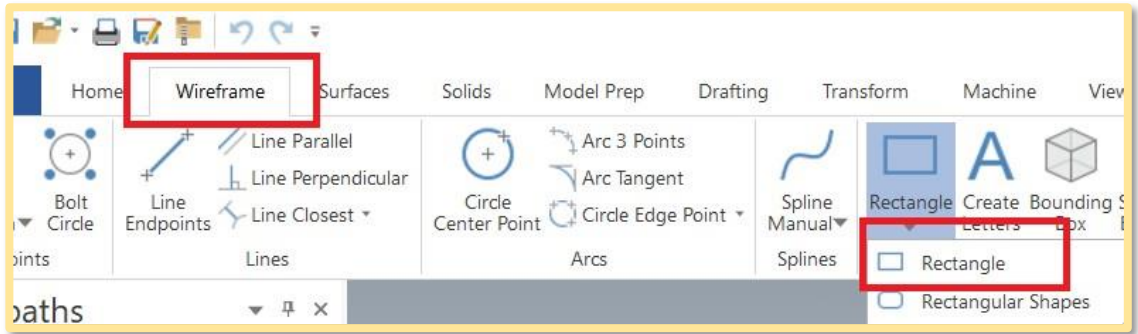
- Open Mastercam software
- From toolbar, click “SETTING”
 - Click “**Configuration**” or click **(Alt+8)**
 - Select Current <metric> and units is “**millimeters**”
- Click **ok**
- Then select shortcutkey “**F9**” for show coordinates axis.



2

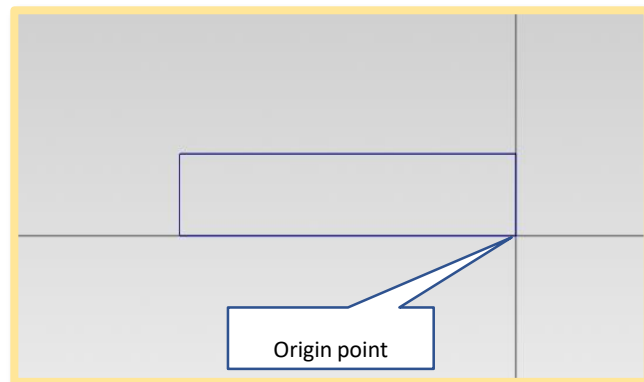
Draw a picture that has been given.

- From toolbar, select **WIREFRAME**
 - Click **Rectangle**



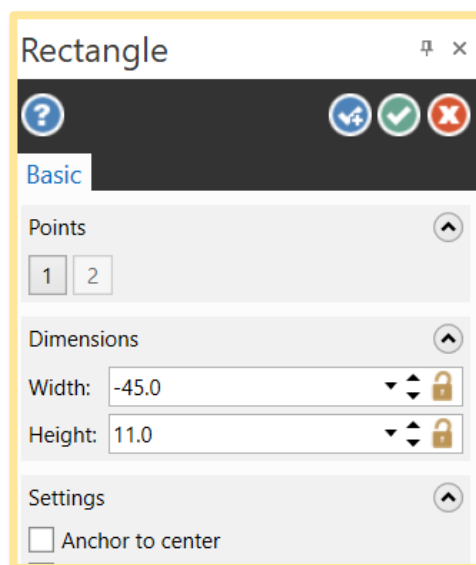
3

- Select position of corner
 - Bring the cursor to the origin point and click.



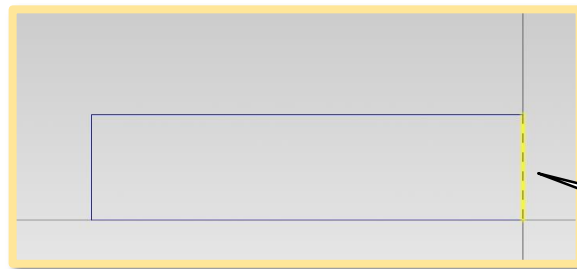
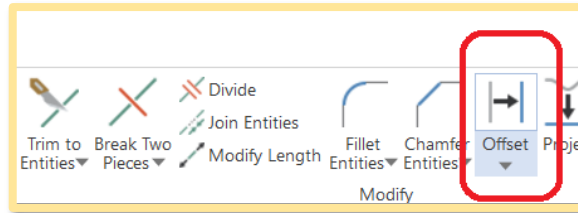
4

- In a ribbon bar, setting the **width :-45** and **height :11.0**
- Click **OK**



5

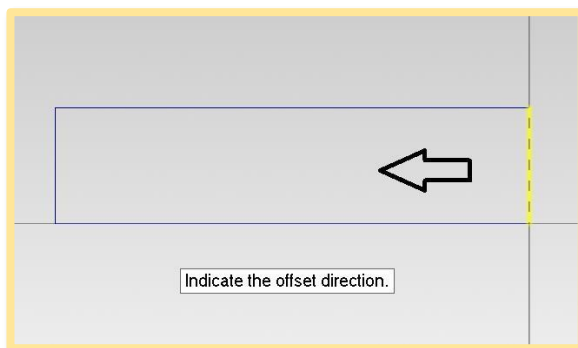
- Select **WIREFRAME** feature
 - Click **Offset**
 - **Select Entity A** as shown in the figure below.



Select Entity A

6

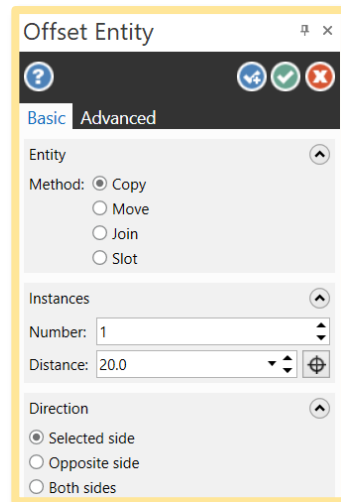
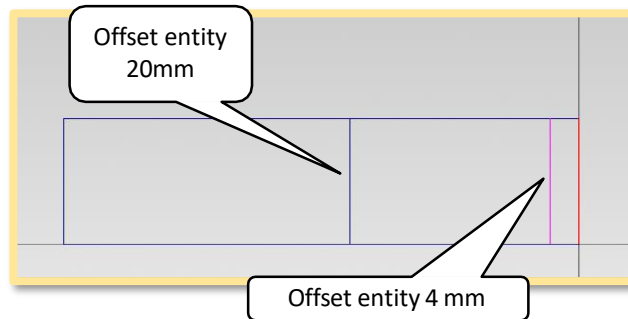
- Choose a point that is left of the chosen line.



Indicate the offset direction.

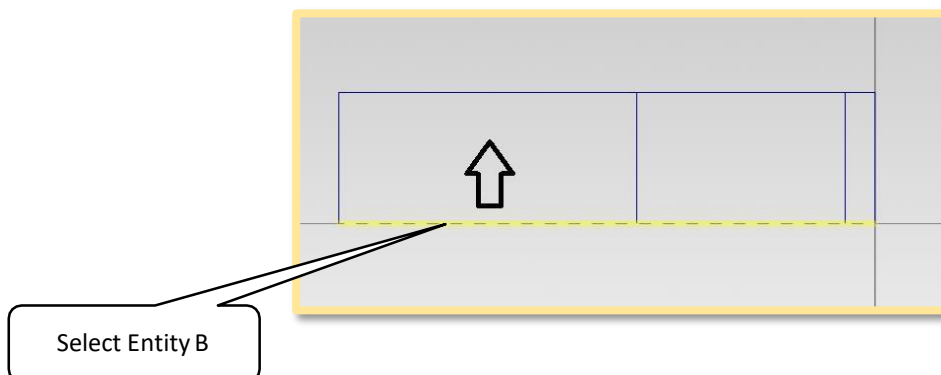
7

- Type the distance :20mm
- Enter
- **Pick a same entities to the left again**
- Type the distance: 4 mm
- Enter



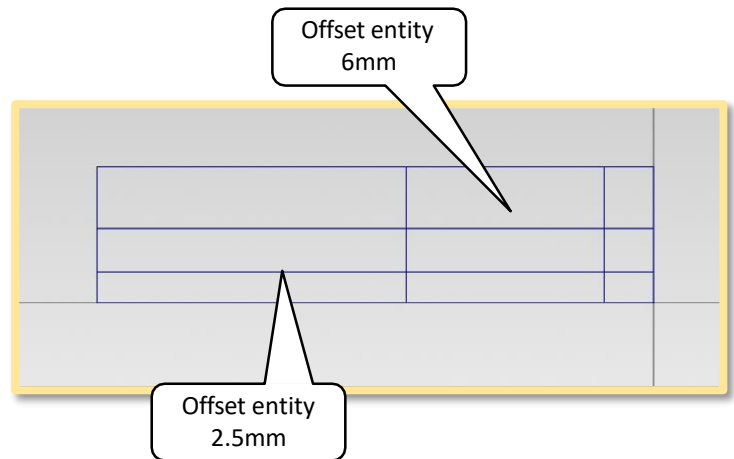
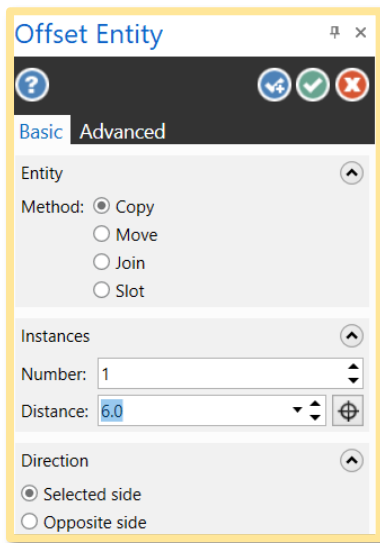
8

- **Select Entity B as shown in the figure.**
- Pick a point to the upper, a parallel line of the selected line.



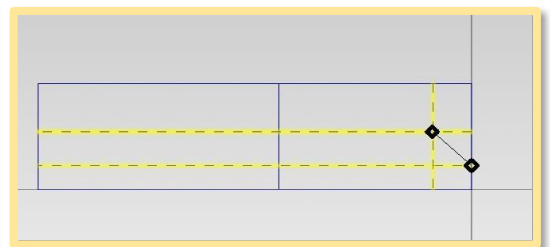
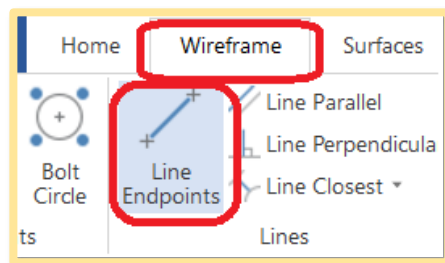
9

- Type the distance : **6 mm**
- Enter
- Pick a point **Entity B** to the upper again
- Type the distance : **2.5mm**
- Enter
- Click **OK** to exit the offset command



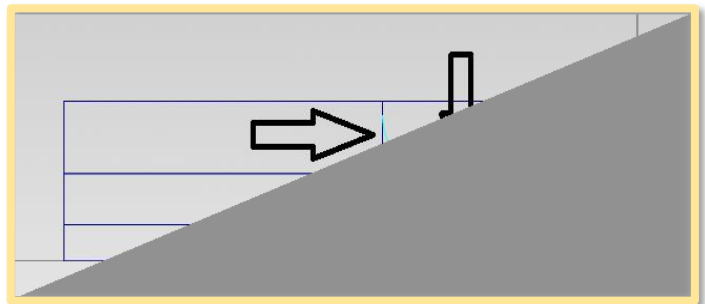
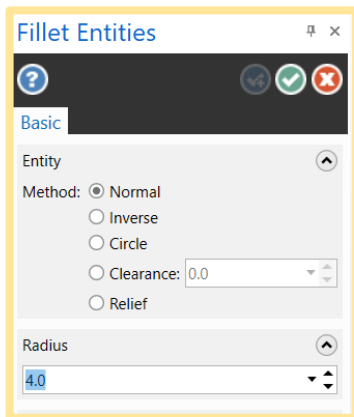
10

- Select **Wireframe**
 - **Line Endpoint**
- Select both points as shown in the figure.
- Click **OK**



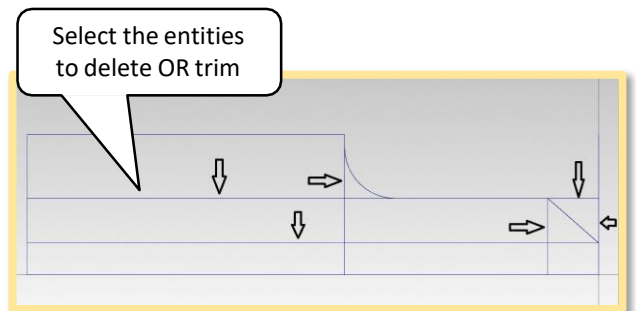
11

- Select **Fillet Entities**
- Set the radius : **4mm**
- Select the both entities
- Click **OK**



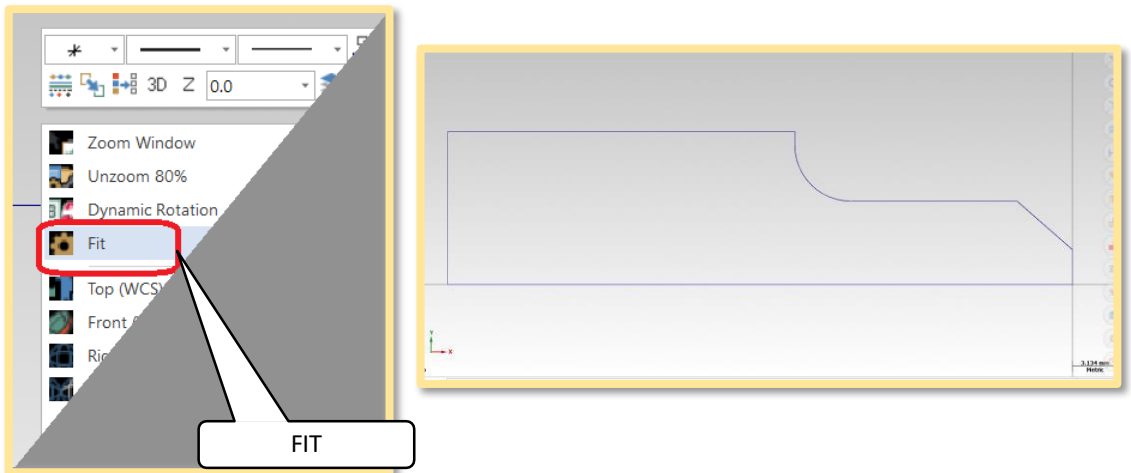
12

- Select **WIREFRAME** feature **Divide** :
 - Click all the entities to trim or delete as shown in the figure



13

- Click the **right of mouse** and choose **FIT** command
- The drawing will be zoom all and fit the window.
- Save the file : **“Tutorial_1”**



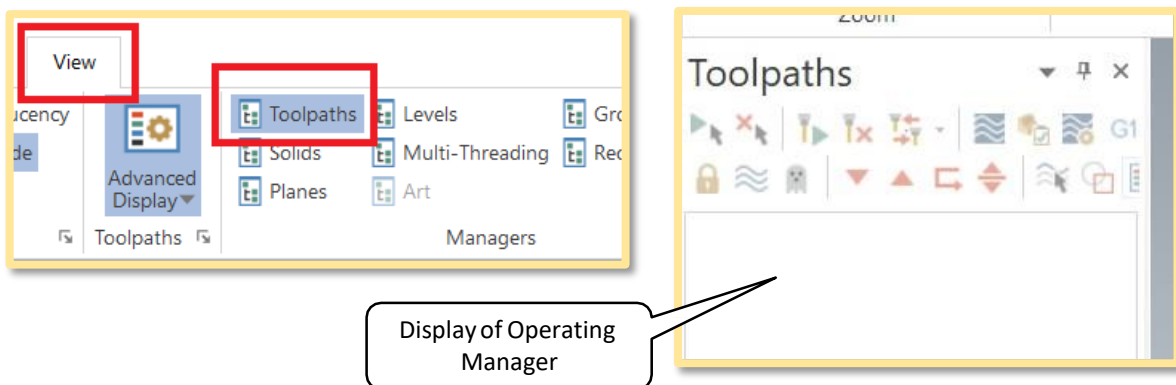


STEP 2: SETUP THE MACHINE DEFINITION AND STOCK

Before we proceed to make any toolpath, we have to select a Machine Definition. The Machine Definition are included mill, lathe, wire router, or mill-turning. It is a template which you can set up the command, features, tool setting, toolpath, spindle speed, feed rate, and plunge rate. It can be converted to the G-code and M-code when you post the processing after finishing the task.

1

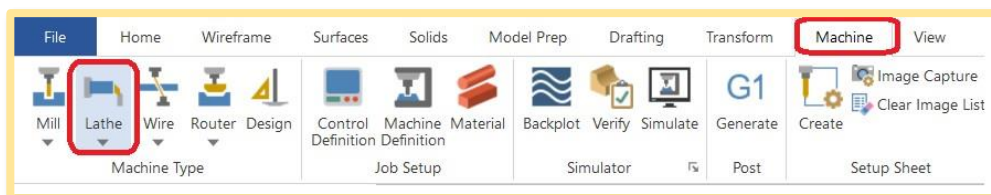
- Select **View** features and click **Toolpaths** for display of **Operating Manager**.
- You can use Type **ALT+O** for display too.
- Select the **Fit** icon in **View Features** to zoom or fit the drawing to the screen



Note : For the purpose of this tutorial, we will be using the **Default Lathe machine**.

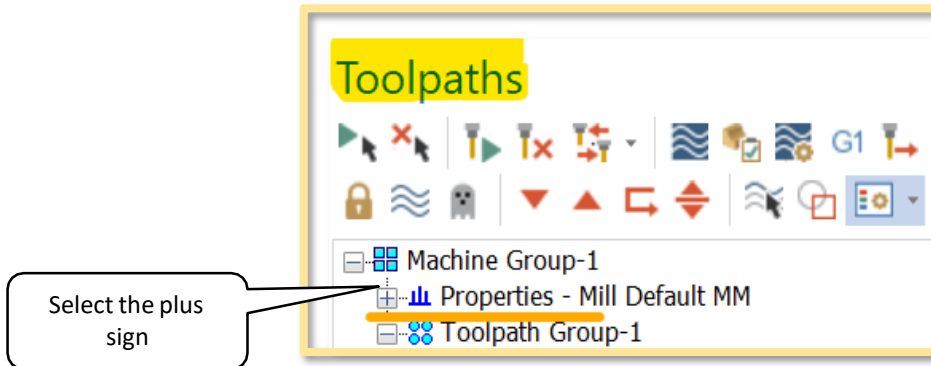
2

- From toolbar, click “**Machine**” features.
 - Lathe
 - Default



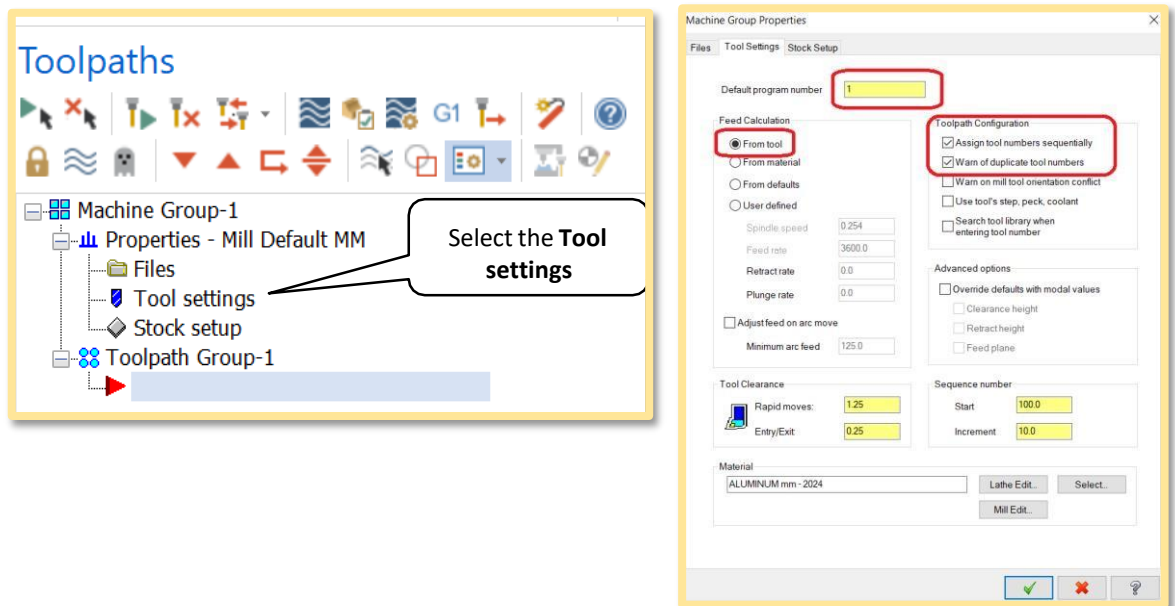
3

- Select the plus sign in front of **Properties** in the **Toolpaths Manager** to expand the **Toolpaths Group Properties** as shown in figure



4

- Select **Tool Settings** to set the tool parameters to match figure below



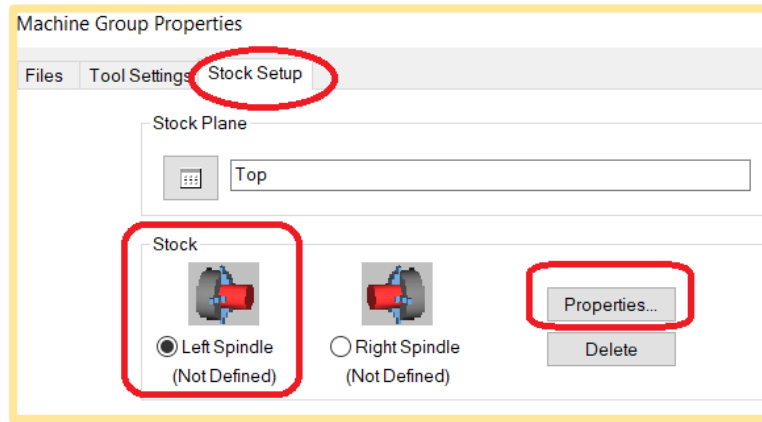
Program # is sequence number in series that used for any CNC machine consists the movements of cutting tool, to command the spindle speed, and feed rate and external M code in the command. The program is an order which the machine coded in a block programmed.

Assign tool numbers sequentially (setting as a default for your machining group which allows student to overwrite the tool number from the library.

Warn of duplicate tool numbers (The warning will be given when you enter the same number of two tools).

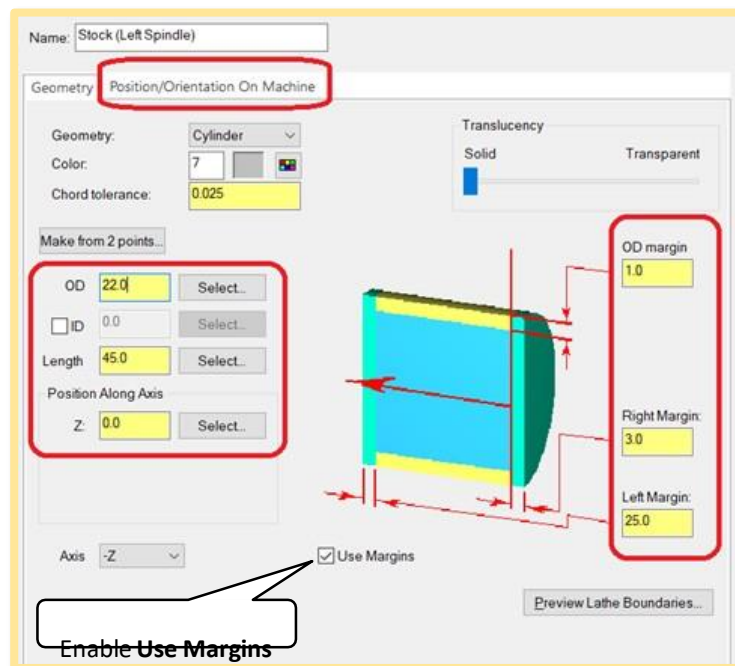
5

- Select **Stock Setup: Left Spindle** and choose **Properties**.



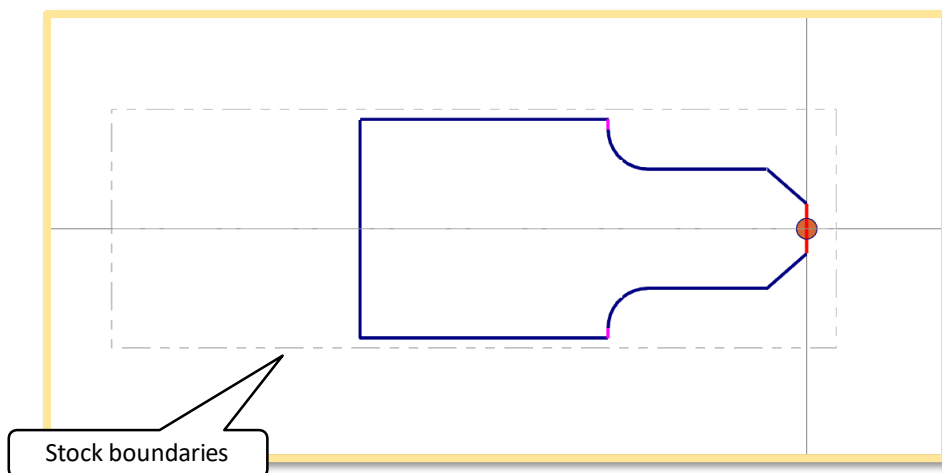
6

- Define the stock.
- Ensure you must enable **Use Margins** before setup the values.
- Setting the stock position and orientation values as shown in the figure.



7

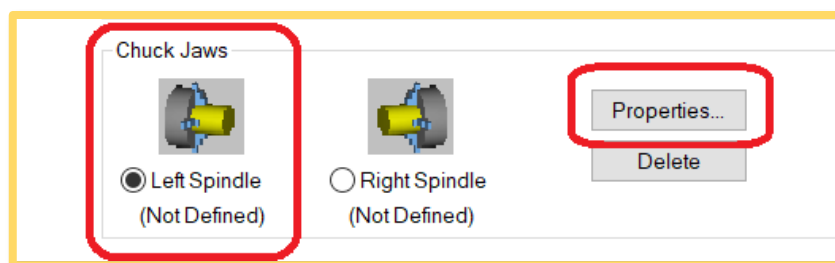
- Select **Preview Lathe Boundaries** to view the stock setup as shown in the figure.
- Click **Esc** to go back the position/orientation on machine of stock setup.



Note : You can create and choose on the left or right spindle. The stock model that you setting and create can be displayed as a preview lathe boundaries.

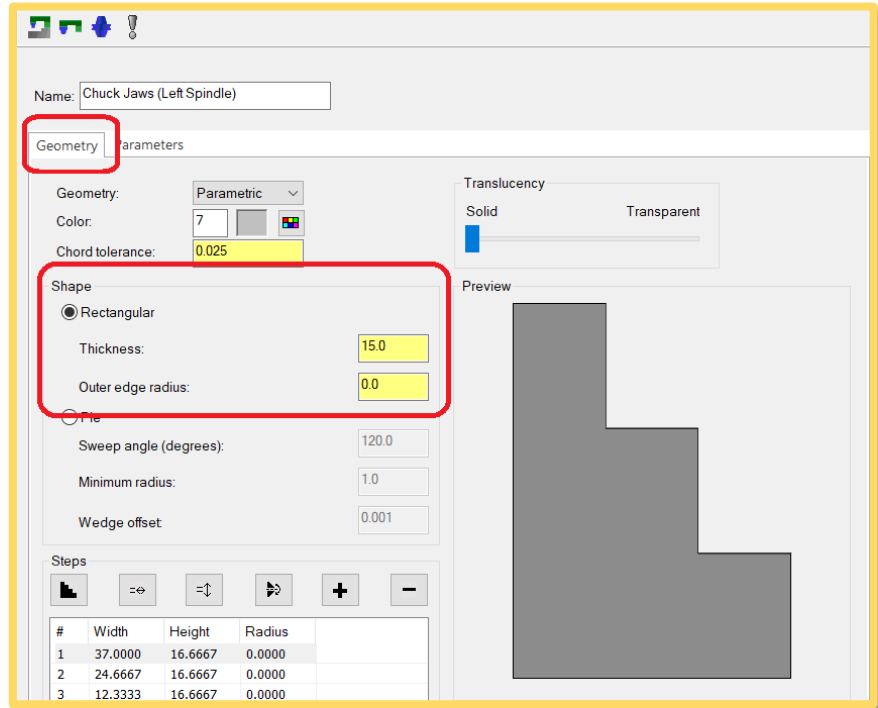
8

- Select **Chuck Jaws: Left Spindle** and choose **Properties**.



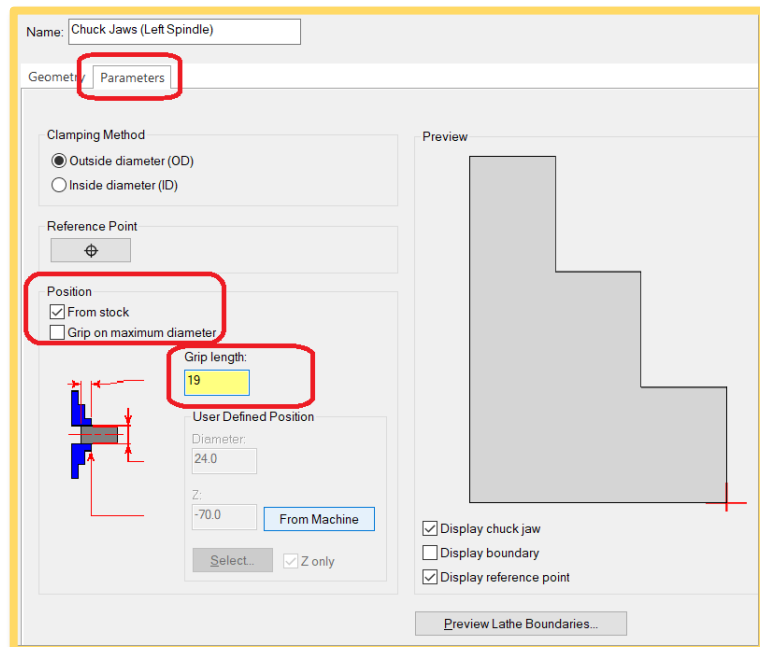
9

- Define the geometry.
- Setting the shape values as shown in the figure.



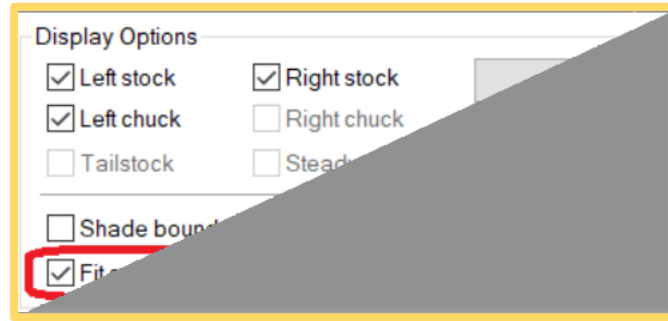
10

- Define the parameters of chuck jaws. .
- Setting the shape values as shown in the figure.
- Select the **Preview Lathe Boundaries** to view the chuck boundaries setup.



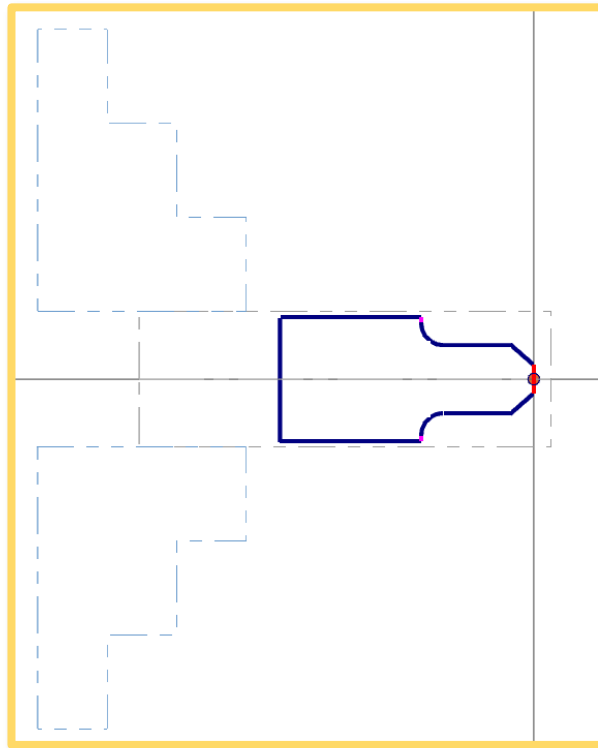
11

- Select OK button to exit the Chuck Jaws setup.
- In the **Display Options**, enable **Fit Screen to Boundaries** as shown.



12

- Select OK button to exit **Machine Group Properties**.
-



13

- Save the file : "Turning Task_1"

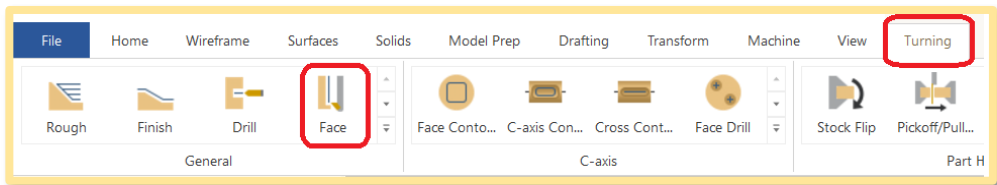


STEP 3: MACHINING THE STOCK USING TURNING TOOL PATH – FACE THE PART

Face toolpaths is allows to quickly clear the stock from the one end of the part and create an even surface for future operations.

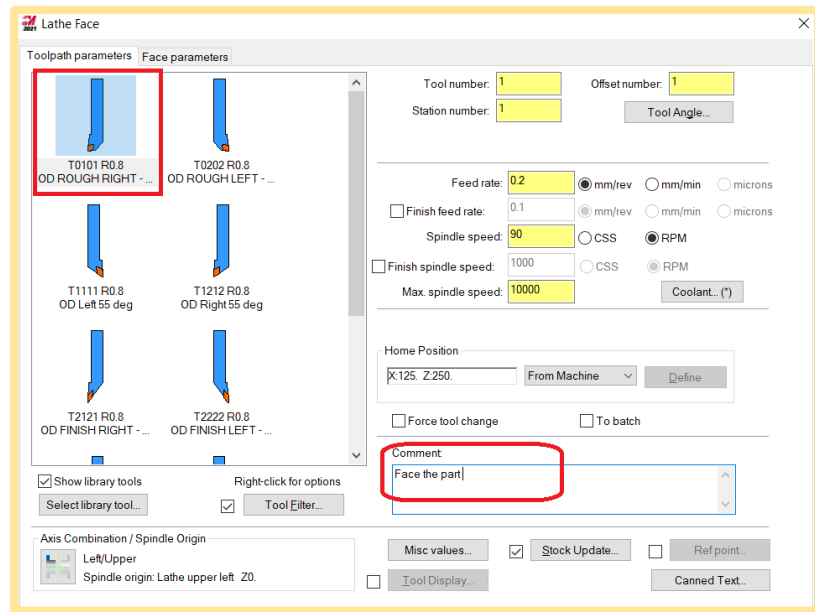
1

- Select features of **Turning** and choose **Face** command.



2

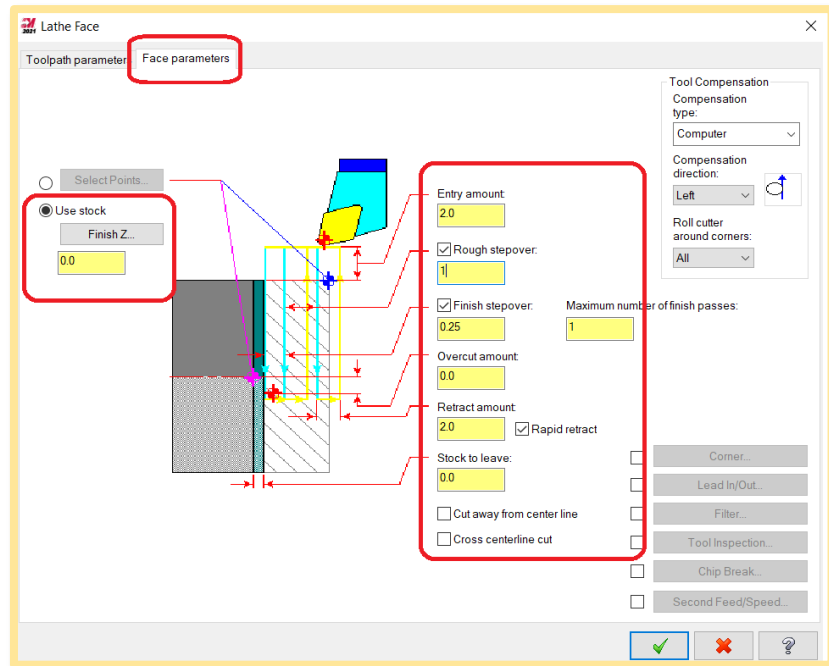
- Select the **T0101 R0.8 Rough Right – 80 degree**.
- Fill the comment as shown in the figure.



Note: When you are modify the pages, the Mastercam will updates the them (in the Tree View).

3

- Setting **Face parameters** and setting all the necessary as shown in Figure.
- Click **OK**



The amount of material to be removed during each pass is determined by the **depth of cut**.

To ensure equal passes, **Equal Steps** sets the depth of cut value to the maximum amount of material that the tool can remove at each pass.

The minimum cut that can be made per pass is determined by the **minimum cut depth**.

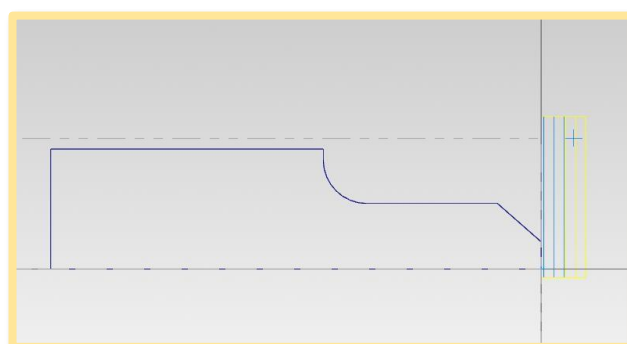
After the tool has made all necessary passes, **stock to leave in X** determines how much stock is still in the X axis.

After the tool has made all necessary passes, **stock to leave in Y** determines how much stock is still in the Y axis.

Entry Amount sets the height at which the tool retracts too or from the part.

4

- The toolpath will appear as shown in the figures.



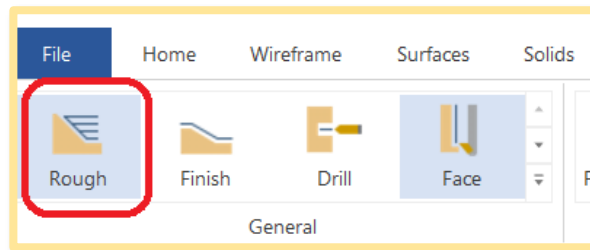


STEP 4: MACHINING THE STOCK USING TURNING TOOL PATH – ROUGH THE PART

Rough toolpaths is allows to remove a large amounts of stock quickly for preparation a finish pass. Roughing passes cuts usually a straight cuts parallel to the Z-axis.

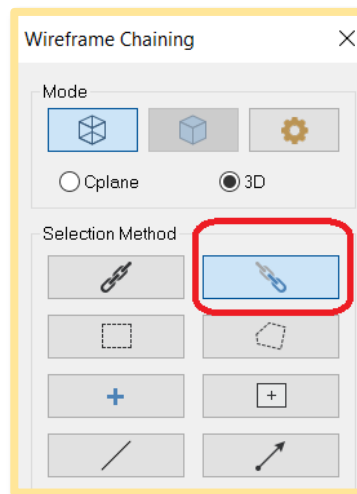
1

- Select the **Rough** operation in **Turning** features.



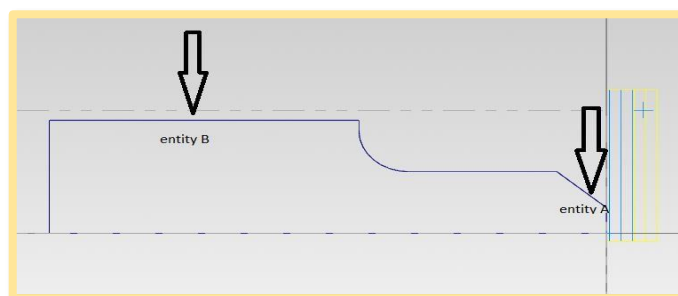
2

- The chaining mode is **Partial** is setting by default.
- For partial, you have to select first entity and last entity of the contour.



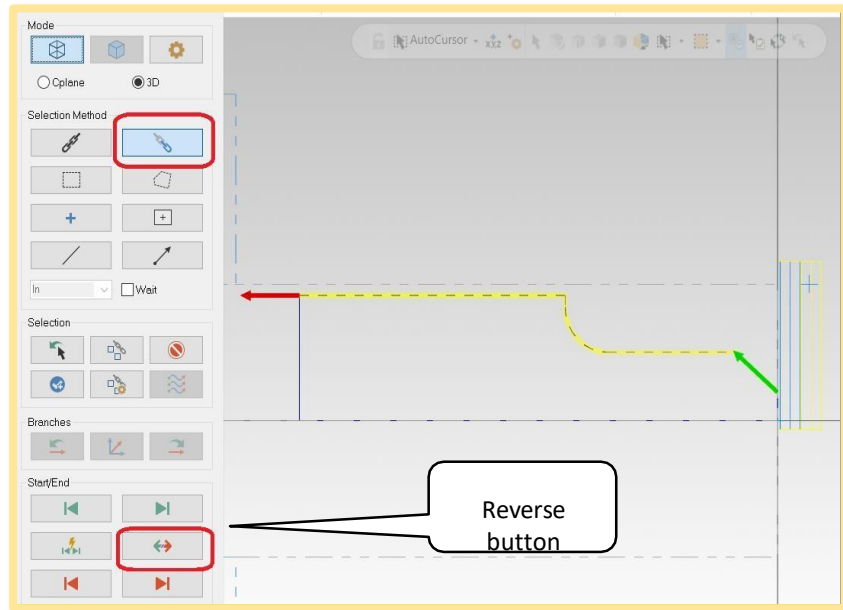
3

- Select **entities A** dan **entities B** as shown in the figure.



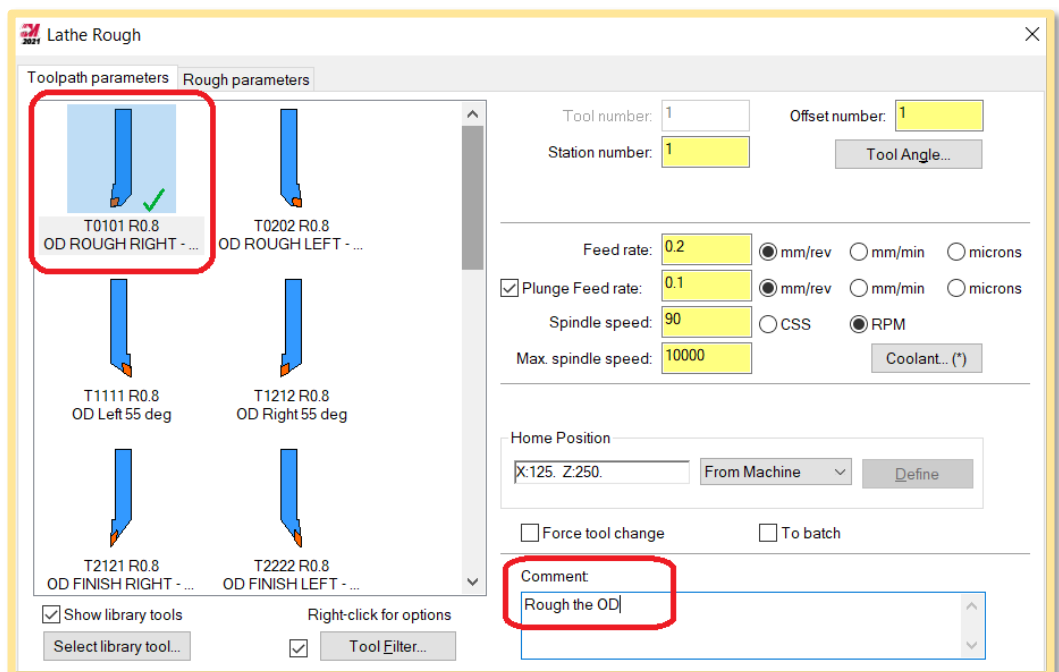
4

- Make sure the chaining direction as shown in the figure (CCW).
- If not, select the **reverse button** in the Chaining dialog box.
- Select the Ok button to exit



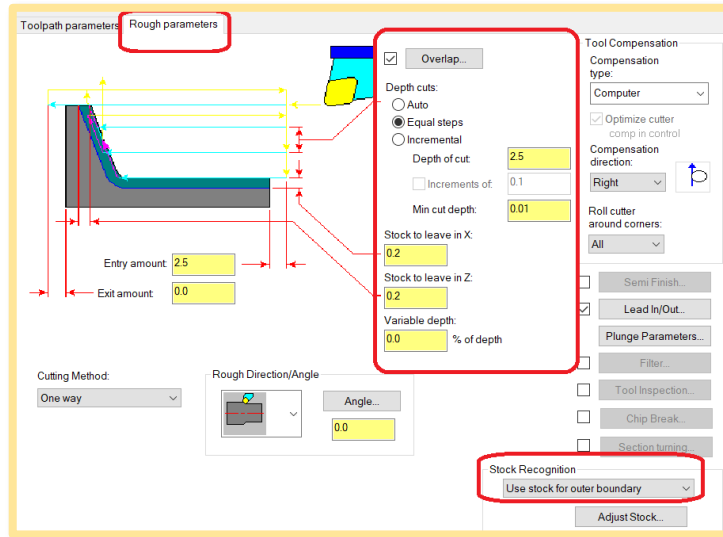
5

- In the Toolpath Parameters, use the same cutting tool that we have used in facing operation.
- Fill the comment as shown in the figure.



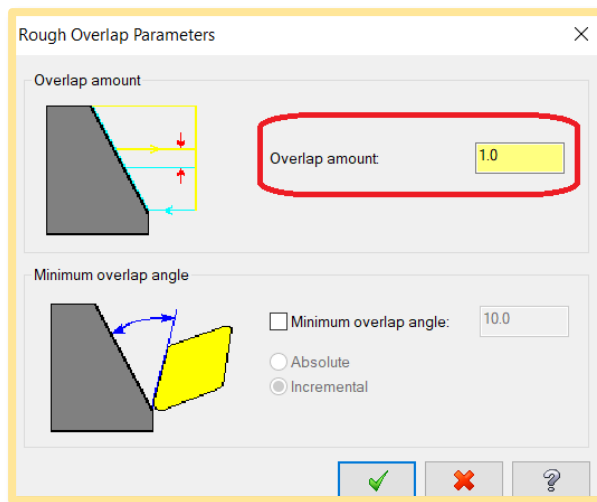
6

- In the **Rough Parameters** tab , makes any necessary changes as shown in the figure.



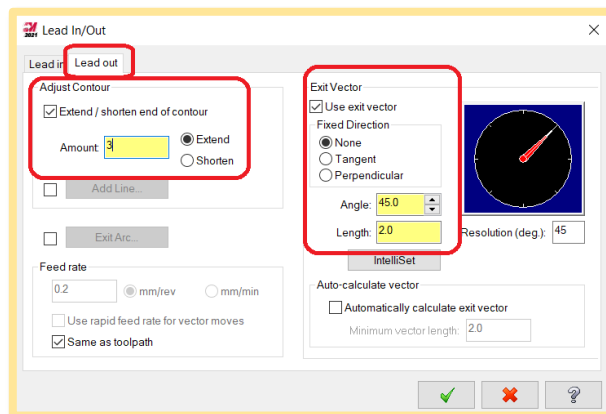
7

- Select the **Rough Overlap Parameters** to establish how much the tool overlaps the previous cut and specify an **Overlap amount** of 1.0 as shown.



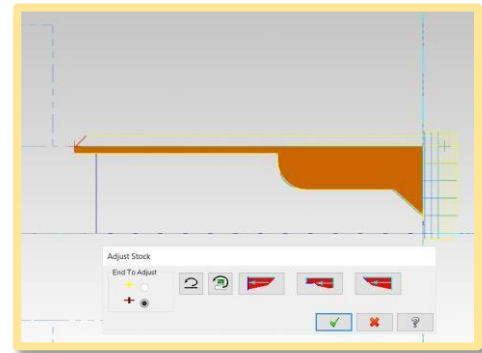
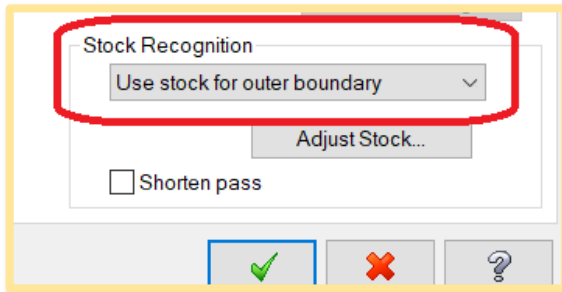
8

- Select the **Lead In/Out** button and choose the **Lead Out** tab to extend the end of the contour as shown in the figure.



9

- In the rough parameters tab, change the **Stock Recognition** to **Use stock for outer boundary**
- You will have option available to **Adjust Stock**.
- When you click on the button these options will appear.



Adjust Stock is used to tell Mastercam how to integrate the contour that you've chained for the toolpath with the stock boundary.

10

- Select the **Ok** button to exit the **Adjust Stock** dialog box
- Select the **Ok** button to exit the **Roughing Toolpath Parameters**

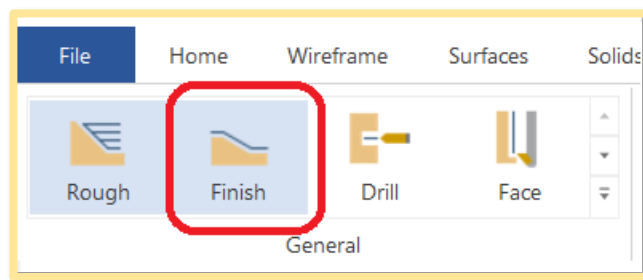


STEP 5 : MACHINING THE STOCK USING TURNING TOOL PATH – FINISH THE PART

Finish toolpaths follow the contour of the chained geometry. Typically a finish toolpath follows a roughing toolpath.

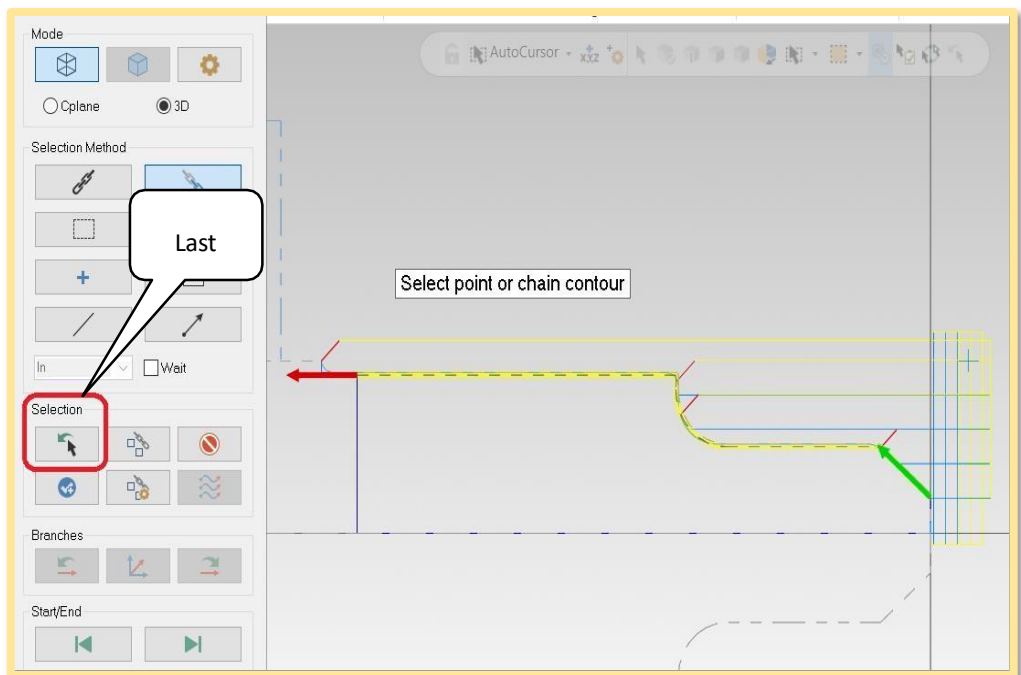
1

- Select the **Finish** operation in **Turning** features.



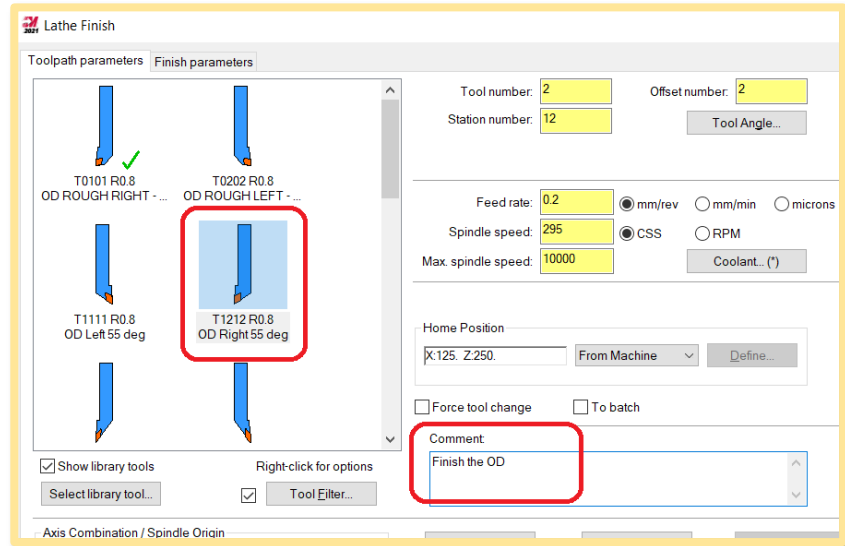
2

- Select the **Last** button in the Chaining dialog box as shown.
- Click **Ok** to exit Chaining Dialog box



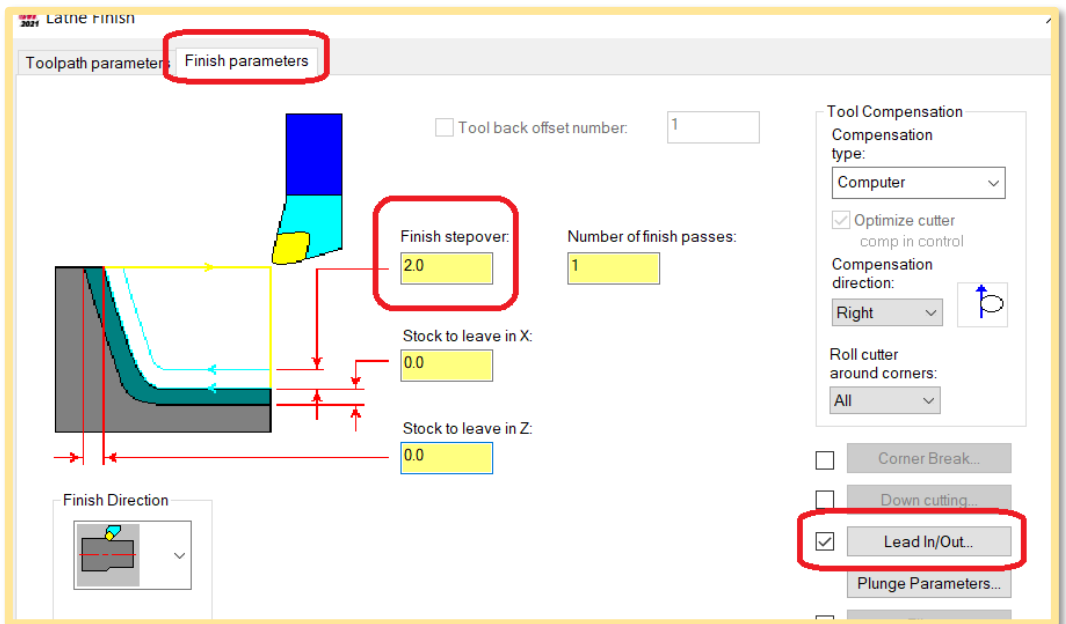
3

- Select the **T1212 OD 55 Degree Right** tool from the tool list.
- Fill the comment as shown in the figure.



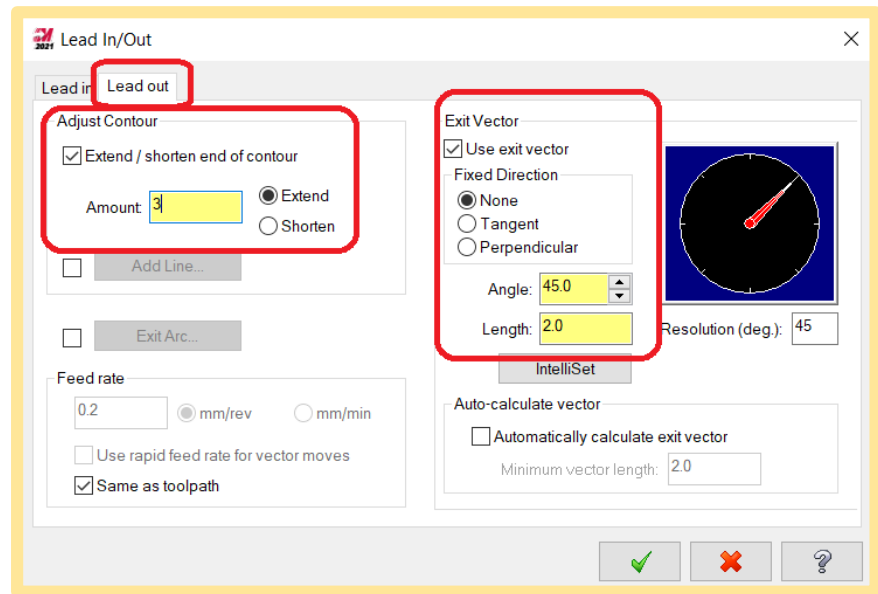
4

- Select the **Finish Parameter** tab and make all of the necessary changes as shown in the figure.



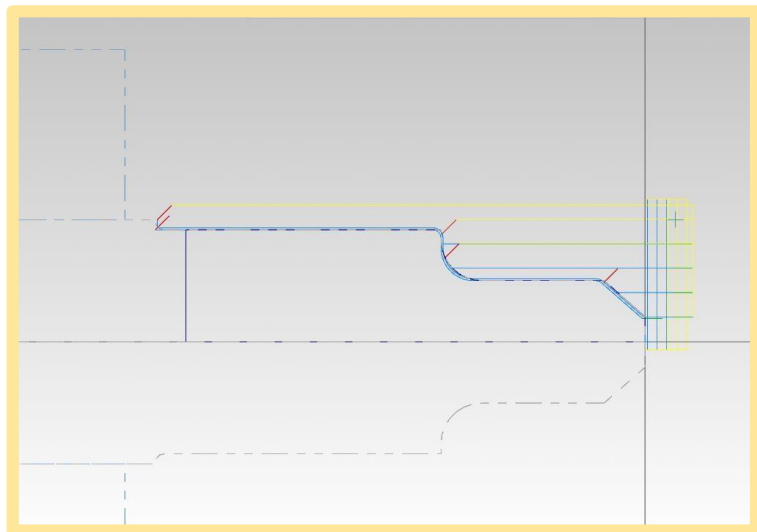
5

- Select the button of “**Regenerate all dirty operations**”.
- This features is for regenerate the modification that you have made to the toolpaths .



6

- Select the **Ok** button twice to exit the **Finish Parameter**.



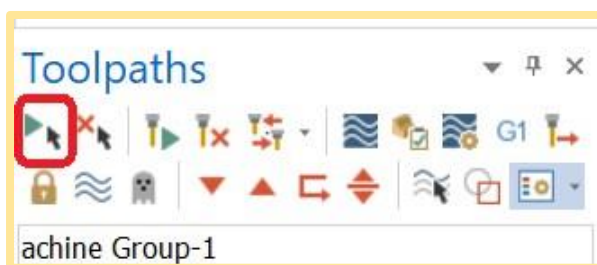


STEP 6 : BLACKPLOT THE TOOLPATHS

Blackplotting shows the path the tools take to cut the part. This display lets you spot errors in the program before you machine the part. As you backplot toolpaths, Mastercam displays the current X, Y and Z coordinates in the lower left corner of the screen.

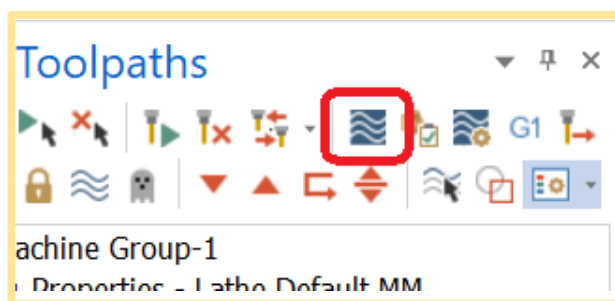
1

- Make sure that you **Select all operations** (signified by the green check mark on the folder icon).



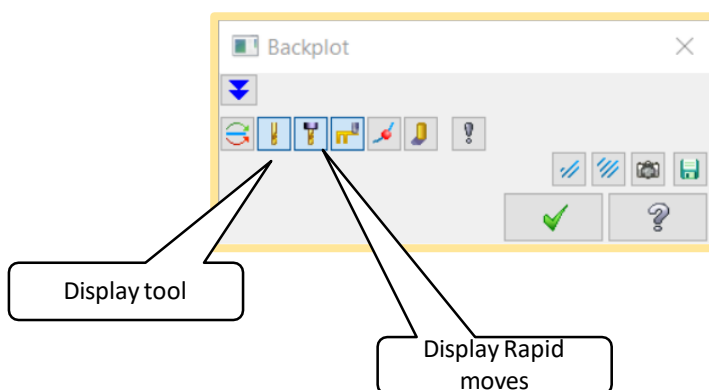
2

- Select the **Backplot selected operations** button as shown.



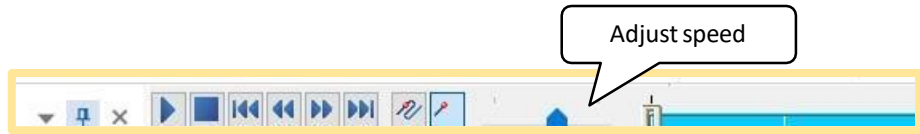
3

- Make sure that you have the following buttons turned on (they will appear pushed down) to see the tool and the rapid moves).



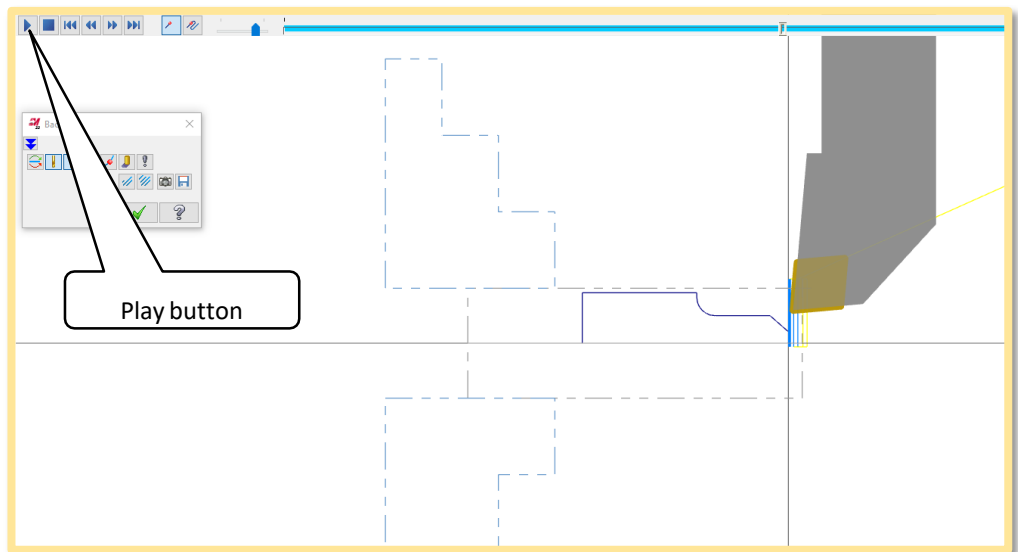
4

- Select the isometric view from the view toolbar to see the stock.
- Select the Fit button



5

- You can adjust the speed of the **blackplot**
- You can step through the **Backplot** by using the **Step Forward** or **Step Back** buttons.
- Select the **Play** button in the **VCR** bar.



6

- Select the **Ok** button to exit the **Backplot**.

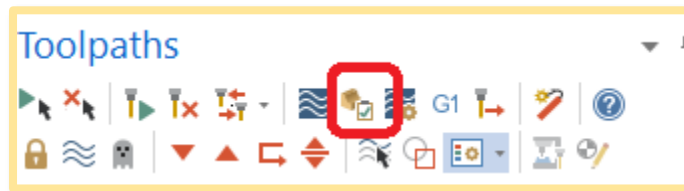


STEP 7 : TOOLPATH VERIFICATION

Toolpath Verification allows you to use solid models to simulate the machining of the part. The model created by the verification represents surface finish and shows collisions if any exist.

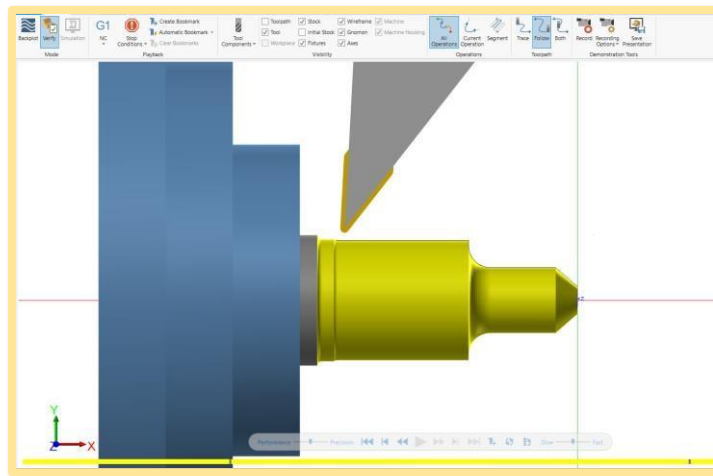
1

- Select the Verify all the operations button as shown.



2

- Use the **Fit** icon to fit the solid to the screen.
- Set the **Verify Speed** by moving the slider bar in the speed control bar
- Select the **Play** button to start the simulation.



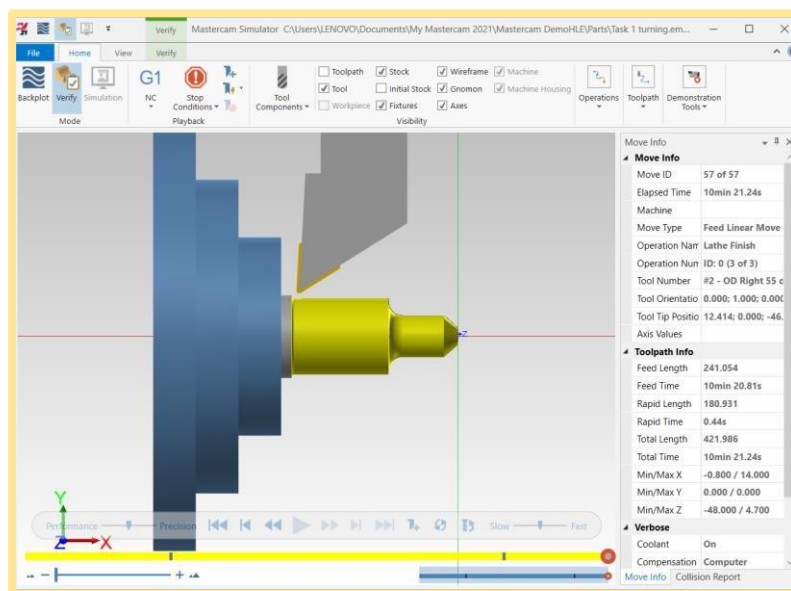
3

- The finished part should appear as shown in the following picture.



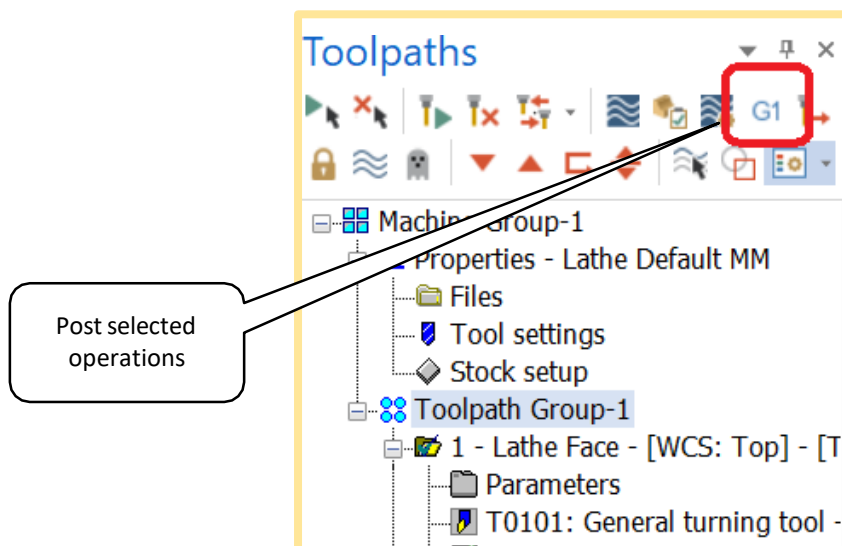
STEP 8 : POST THE PROCESSOR FOR G-CODE FILE

Post Processing refers to the process by which the toolpaths in your Mastercam part files are converted to a format that can be understood by your machine tool's control. A special program reads your Mastercam file and writes the appropriate NC code.



1

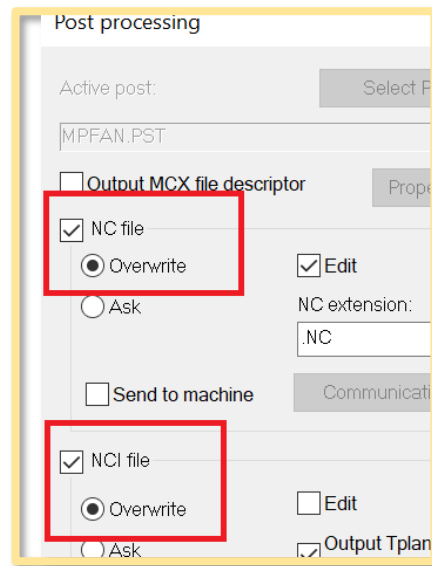
- Click the **Post selected operation** button in the Operation Manager.
- Make the necessary changes as shown in **Post Processing** window in the figure above.



- Note: Make sure all the operations are selected. **Select Overwrite** in the **Operations Manager**.

2

- Set the Post Processing like the Figure.
- Select OK button to continue.



3

- After generates NC codes, copy all the codes and save in as **Notepad** version.
- You can use this post in the NC code at CNC machine.

```

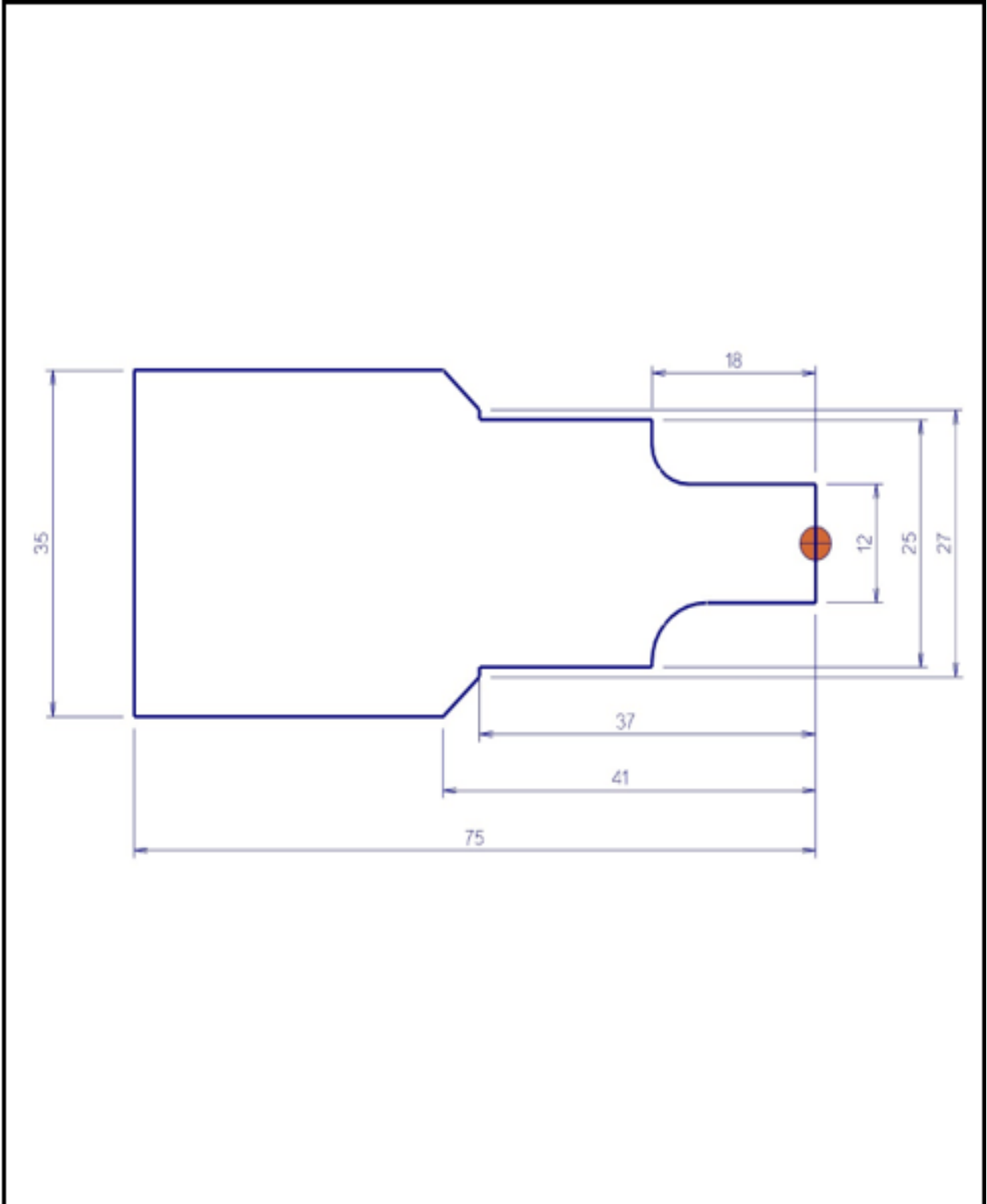
1  %
2  O0001]
3  (PROGRAM NAME - TURNING 1)
4  (DATE=DD-MM-YY - 02-08-22 TIME=HH:MM - 10:35)
5  (MCX FILE - C:\USERS\HASNY\DOCUMENTS\MY MCMX9\MCX\TURNING 1.MCX-9)
6  (NC FILE - C:\USERS\HASNY\DOCUMENTS\MY MCMX9\LATHE\NC\TURNING 1.NC)
7  (MATERIAL - ALUMINUM MM - 2024)
8  G21
9  (TOOL - 1 OFFSET - 1)
10 (OD ROUGH RIGHT - 80 DEG.  INSERT - CNMG 12 04 08)
11 ( FACE THE PART )
12 G0 T0101
13 G18
14 G97 S3126 M03
15 G0 G54 X28. Z2.083 M8
16 G50 S3600
17 G96 S275
18 G99 G1 X-1.6 F.25
19 G0 Z4.083
20 X28.
21 Z1.167
22 G1 X-1.6
23 G0 Z3.167
24 X28.
25 Z.25
26 G1 X-1.6
27 G0 Z2.25
28 X28.

```



PRACTICAL TASK 2

- **Create the geometry** from Practical Task 2 exercise use commands rectangle, line endpoint, offset, fillet entities, chamfer entities, trim/divide and delete entities.
- **Set the machine properties** including stock and chuck setup. **Define** the facing, roughing, and finishing operation to the OD of the part.
- **Setup** lead in/out and extend the toolpath so there **is no collision** between chuck, stock and cutting tool.

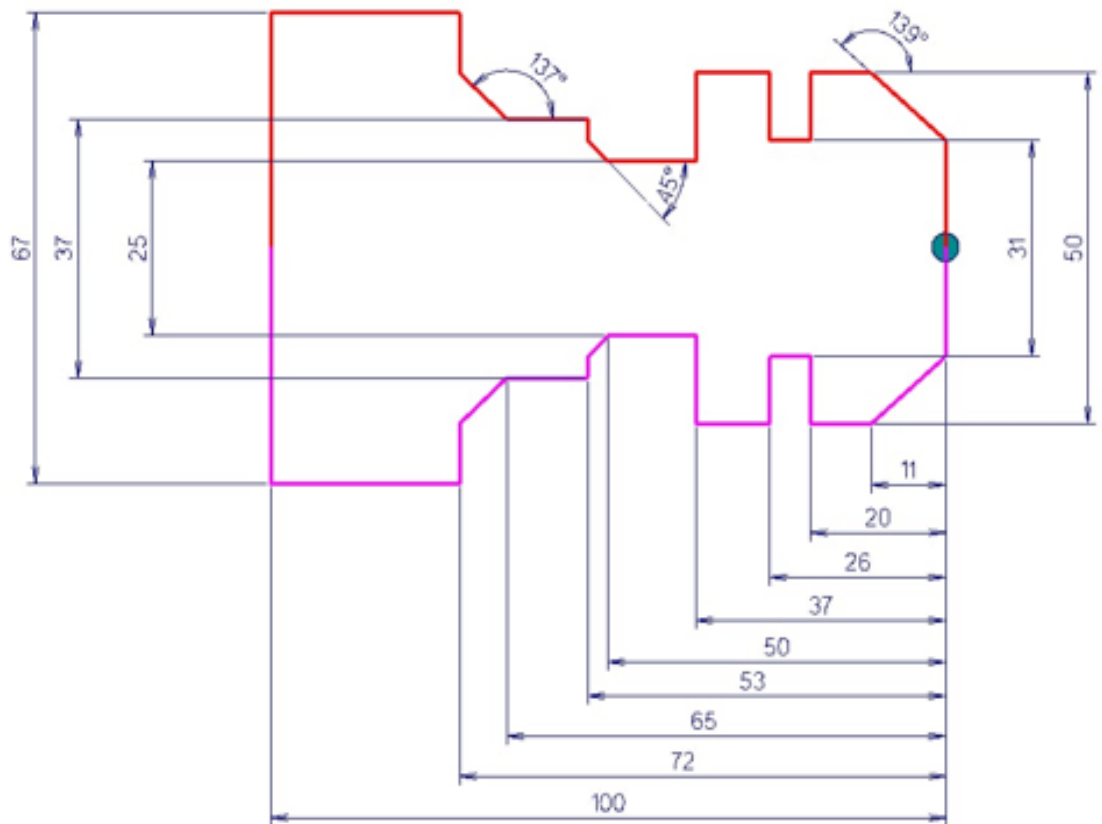


TITLE	TURNING: PRACTICAL TASK 2	DEPT	MECHANICAL ENGINEERING
POLITEKNIK PORT DICKSON		COURSE	DJF41042 (CAD CAM)
		MATERIAL	ALUMINIUM 6061
		UNIT	METRIC (MM)



PRACTICAL TASK 3

- **Create the geometry** from Practical Task 2 exercise use commands rectangle, line endpoint, offset, fillet entities, chamfer entities, trim/divide and delete entities.
- **Set the machine properties** including stock and chuck setup. **Define** the facing, roughing, and finishing operation to the OD of the part.
- **Setup** lead in/out and extend the toolpath so there **is no collision** between chuck, stock and cutting tool.



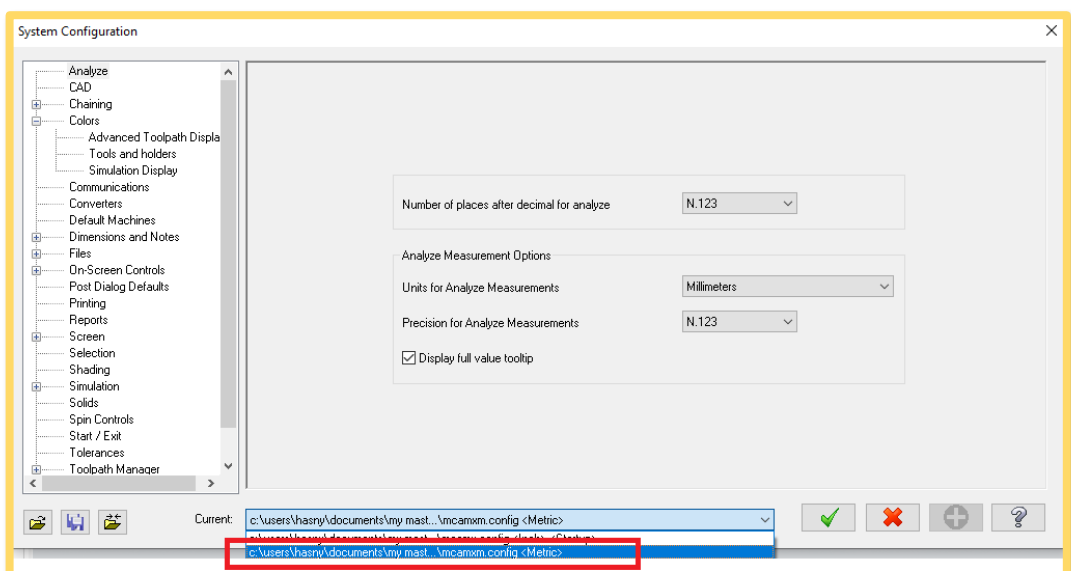
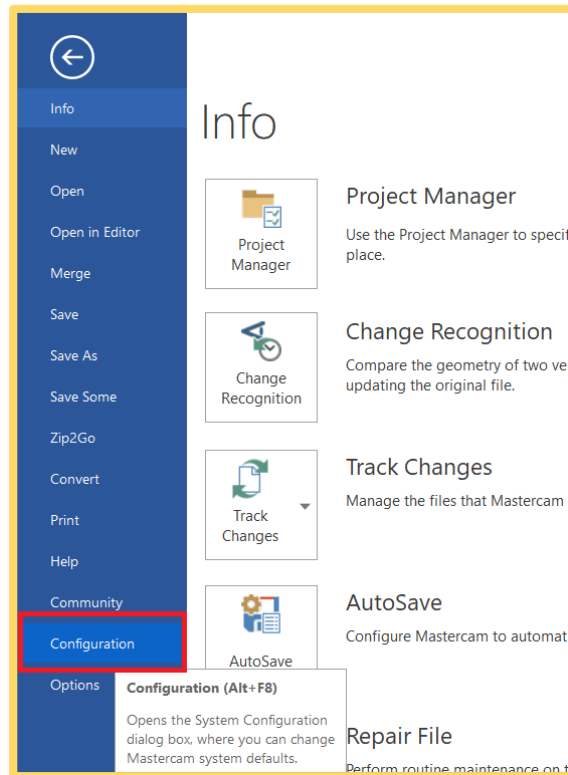
TITLE	TURNING: PRACTICAL TASK 3	DEPT	MECHANICAL ENGINEERING
POLITEKNIK PORT DICKSON		COURSE	DJF41042 (CAD CAM)
		MATERIAL	ALUMINIUM 6061
		UNIT	METRIC (MM)



STEP 1: CREATE A GEOMETRY

1

- Open Mastercam software
- From toolbar, click “SETTING”
 - Click “Configuration” or click (Alt+8)
 - Select Current <metric> and units is “millimeters”
- Click ok
- Then select shortcutkey “F9” for show coordinates axis.



2

- Scan the QR code.
- Watch the tutorial video how to drafting the task in Mastercam 2022.
- Draw a picture that has been given.
- Save the file : “**Task_3**”

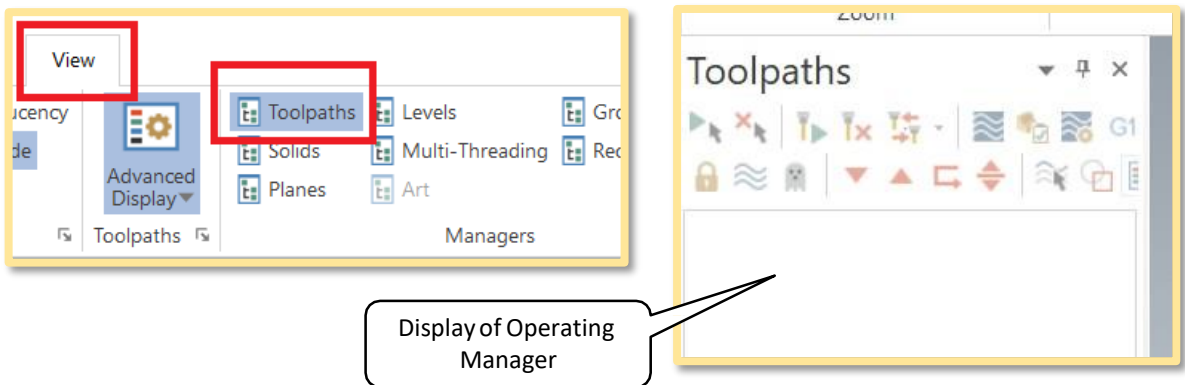


STEP 2: SETUP THE MACHINE DEFINITION AND STOCK

Before we proceed to make any toolpath, we have to select a Machine Definition. The Machine Definition are included mill, lathe, wire router, or mill-turning. It is a template which you can set up the command, features, tool setting, toolpath, spindle speed, feed rate, and plunge rate. It can be converted to the G-code and M-code when you post the processing after finishing the task.

1

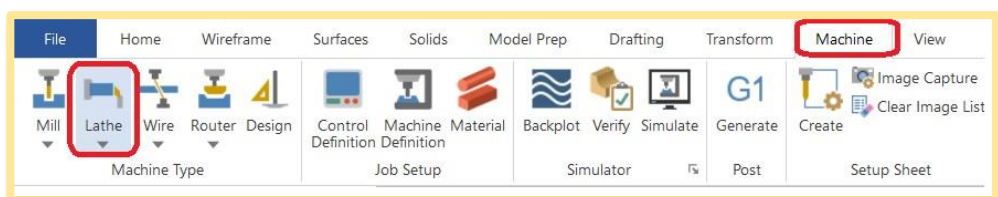
- Select **View** features and click **Toolpaths** for display of **Operating Manager**.
- You can use Type **ALT+O** for display too.
- Select the **Fit** icon in **View Features** to zoom or fit the drawing to the screen



Note : For the purpose of this tutorial, we will be using the **Default Lathe machine**.

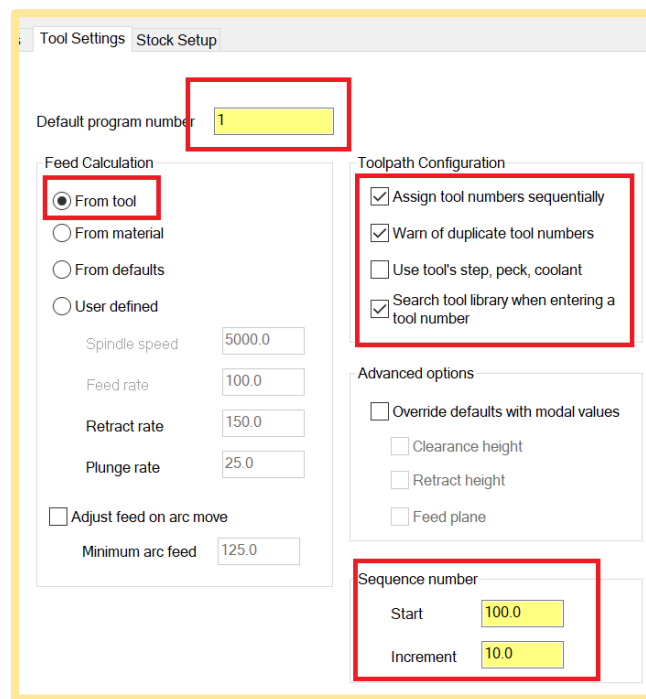
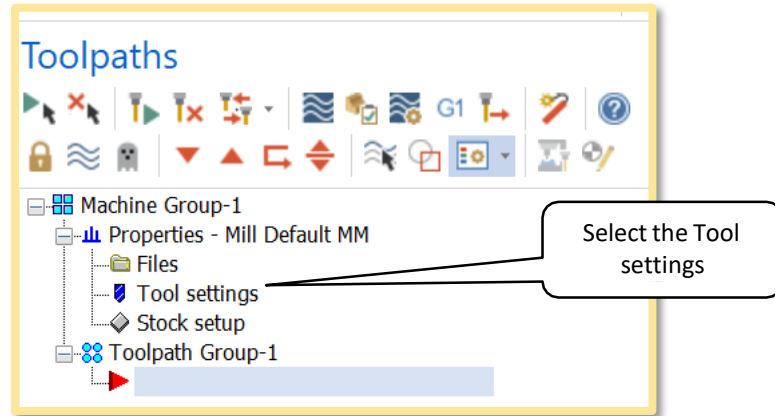
2

- From toolbar, click “**Machine**” features.
- Lathe
- Default



3

- Select **Tool Settings** to set the tool parameters to match Figure.



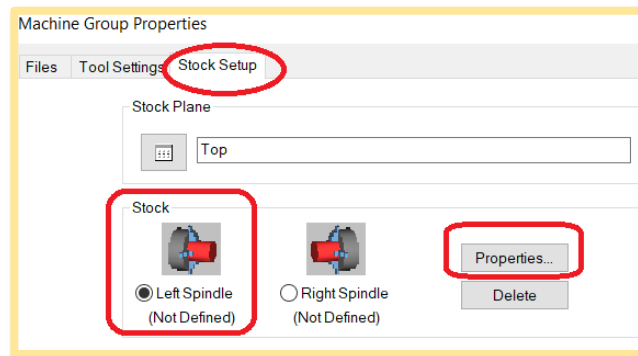
Program # is sequence number in series that used for any CNC machine consists the movements of cutting tool, to command the spindle speed, and feed rate and external M code in the command. The program is an order which the machine coded in a block programmed.

Assign tool numbers sequentially (setting as a default for your machining group which allows student to overwrite the tool number from the library).

Warm of duplicate tool numbers (The warning will be given when you enter the same number of two tools).

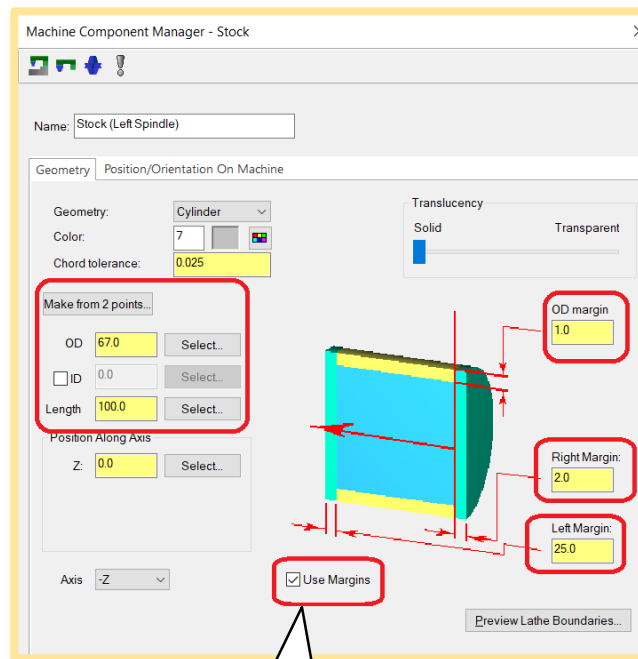
4

- Select **Stock Setup: Left Spindle** and choose **Properties**.



5

- Define the stock.
- Ensure you must enable **Use Margins** before setup the values.
- Setting the stock position and orientation values as shown in the figure.

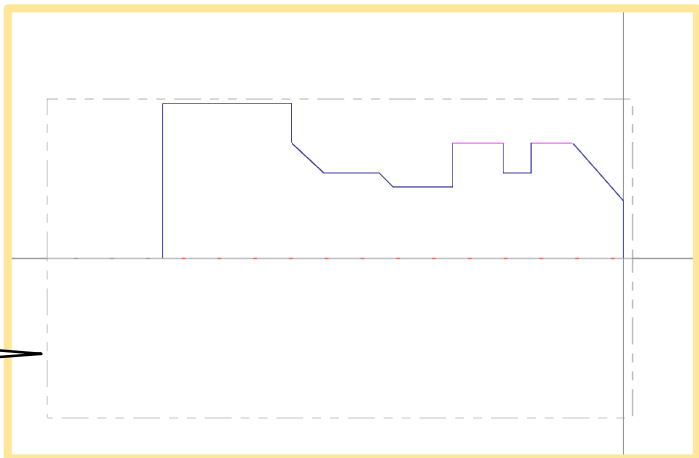


Note : The stock model that you create can be displayed with the part geometry when viewing the file or the toolpaths, during backplot or while verifying toolpaths. You can create stock on the left or right spindle.

6

- Select **Preview Lathe Boundaries** to view the stock setup as shown in the figure.
- Click **Esc** to go back the position/orientation on machine of stock setup.

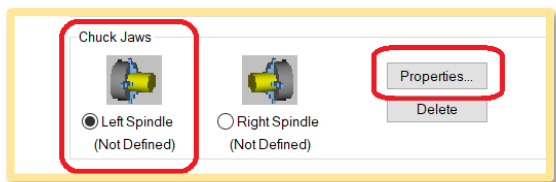
Stock boundaries



Note : You can create and choose on the left or right spindle. The stock model that you setting and create can be displayed as a preview lathe boundaries.

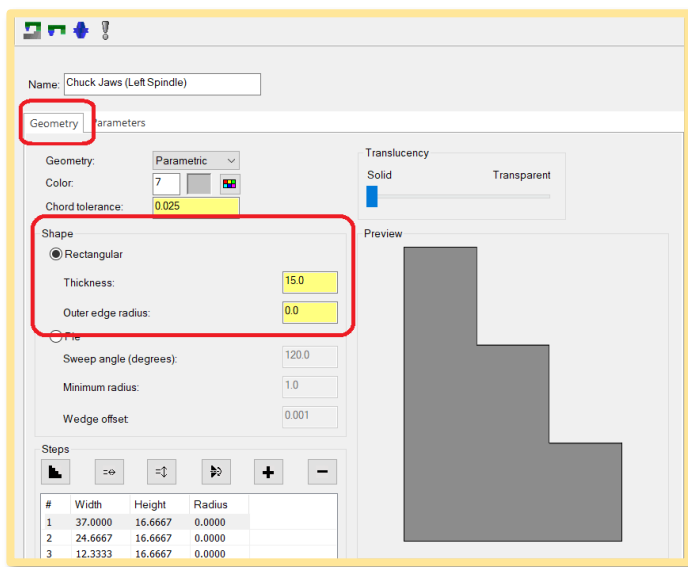
7

- Select **Chuck Jaws: Left Spindle** and choose **Properties**.



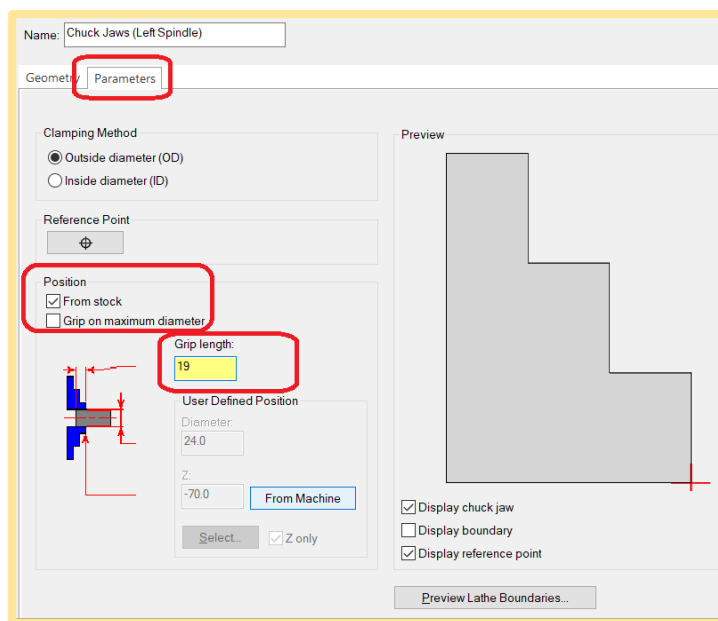
8

- Define the geometry.
- Setting the shape values as shown in the figure



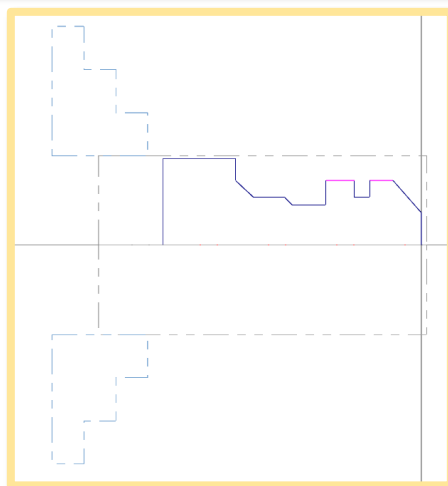
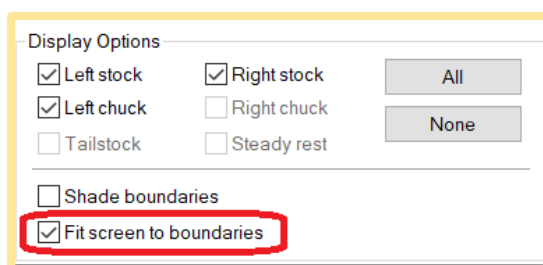
9

- Define the parameters of chuck jaws. .
- Setting the shape values as shown in the figure.
- Select the **Preview Lathe Boundaries** to view the chuck boundaries setup.



10

- Select OK button to exit the Chuck Jaws setup.
- In the **Display Options**, enable **Fit Screen to Boundaries** as shown.
- **Select OK** button to exit **Machine Group Properties**.



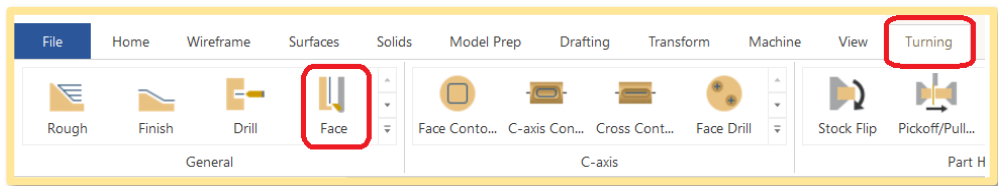


STEP 3 : MACHINING THE STOCK USING 2D CONTOUR TOOLPATH_FACING

Face toolpaths is allows to quickly clear the stock from the one end of the part and create an even surface for future operations.

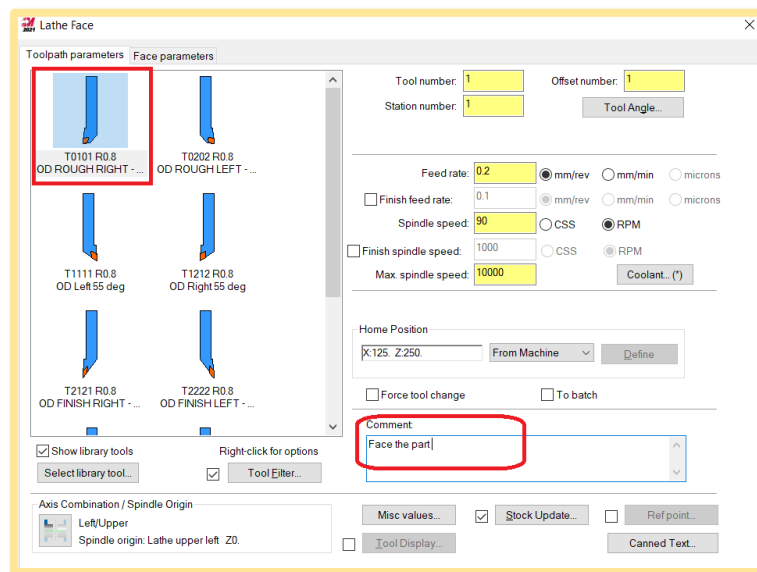
1

- Select features of **Turning** and choose **Face** command.



2

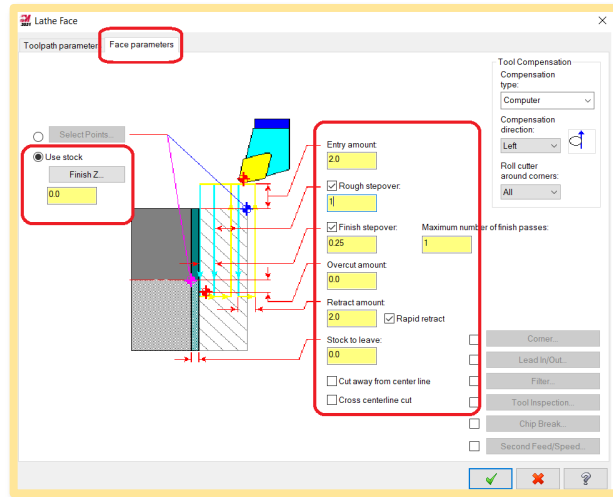
- Select the **T0101 R0.8 Rough Right – 80 degree**.
- Fill the comment as shown in the figure.



Note: When you are modify the pages, the Mastercam will updates the them (in the Tree View).

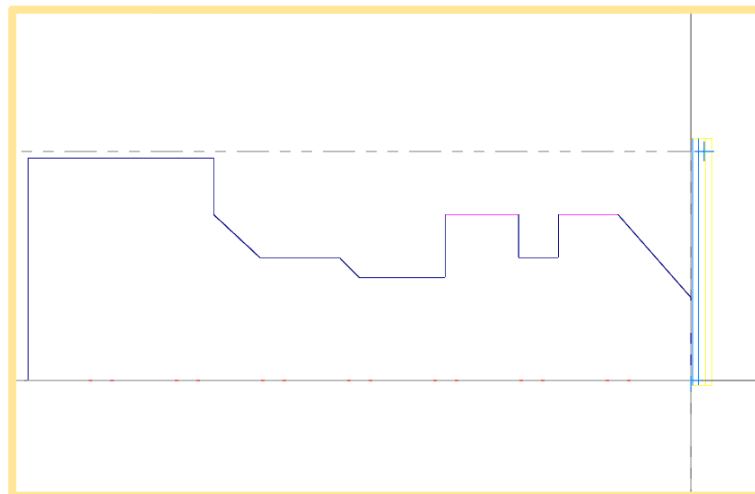
3

- Setting **Face** parameters and setting all the necessary as shown in Figure.
- Click **OK**



4

- The toolpath will appear as shown in the figures.



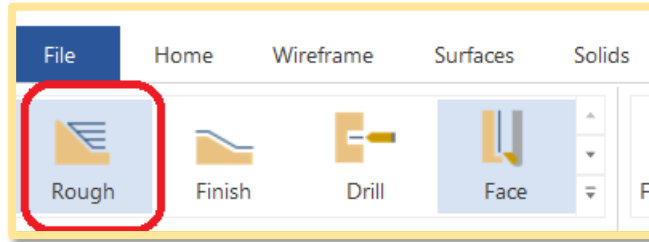


STEP 4 : MACHINING THE STOCK USING 2D CONTOUR TOOLPATH_ROUGHING

Rough toolpaths is allows to remove a large amounts of stock quickly for preparation a finish pass. Roughing passes cuts usually a straight cuts parallel to the Z-axis.

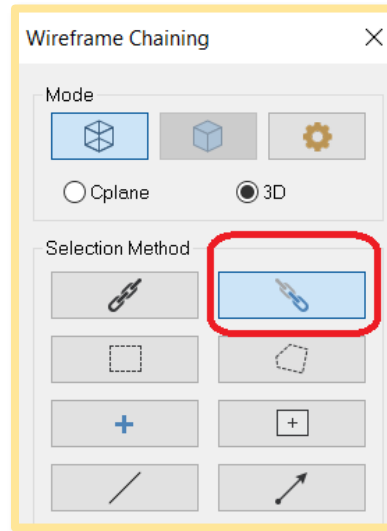
1

- Select the **Rough** operation in **Turning** features.



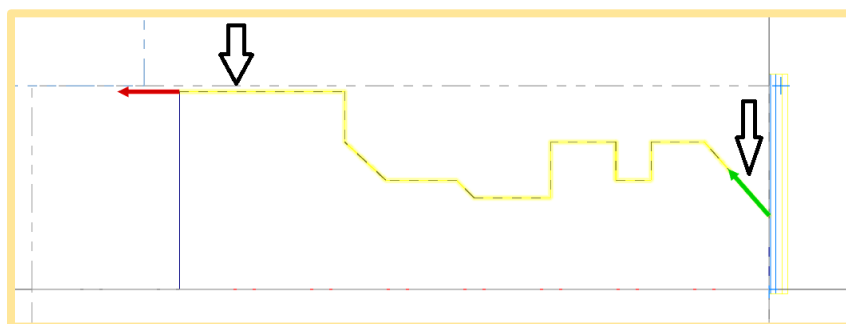
2

- The chaining mode is **Partial** is setting by default.
- For partial you have to select first entity and last entity of the contour.



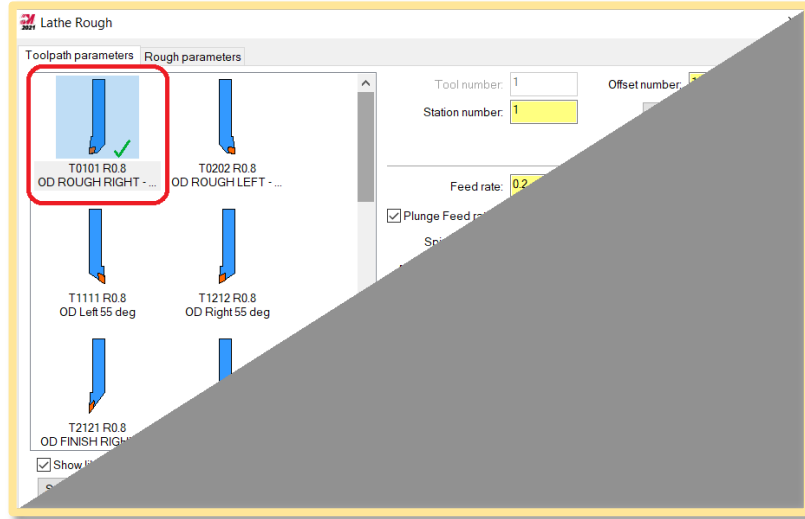
3

- Select **entities A** dan **entities B** as shown in the figure.
- Make sure the chaining direction as shown in the figure (CCW).
- If not, select the **reverse button** in the Chaining dialog box



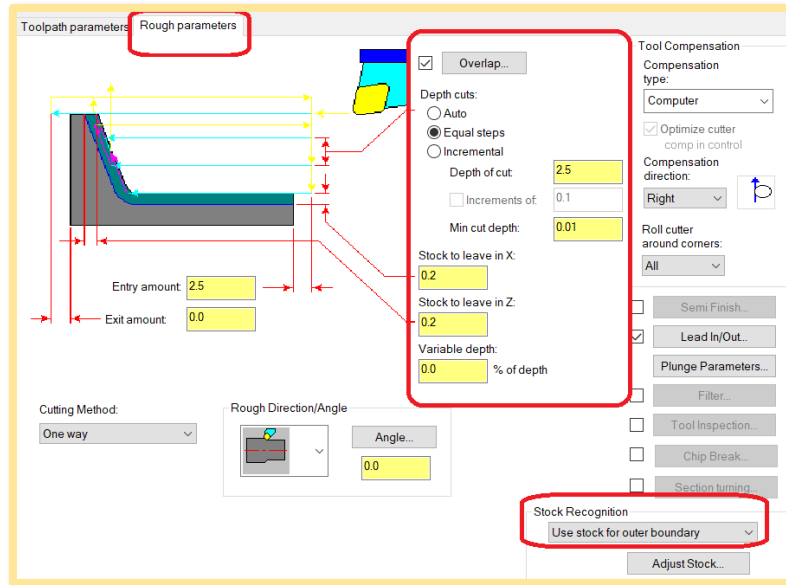
4

- In the Toolpath Parameters, uses the same cutting tool that we have used in facing operation.
- Fill the comment as shown in the figure.



5

- In the **Rough Parameters** tab, makes any necessary changes as shown in the figure.



Depth of cut is sets the amount of material to be removed during each passes.

Minimum cut depth is the sets of minimum cut that can be taken per pass.

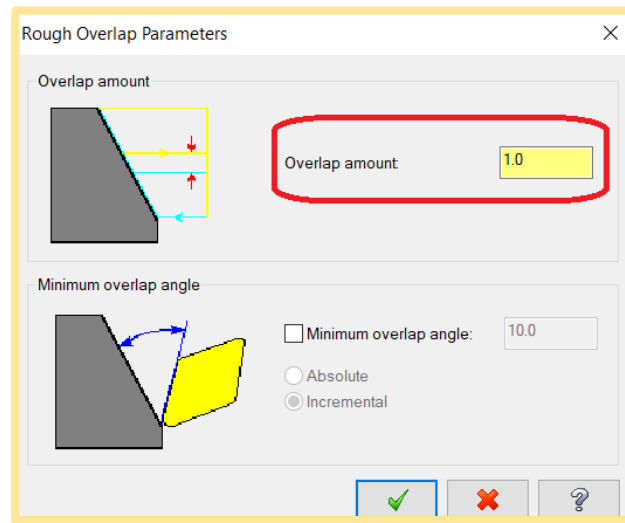
Stock to leave in X sets the remaining stock in the X axis after the tool completed all passes.

Stock to leave in Y sets the remaining stock in the Y axis after the tool completed all passes.

Entry amount sets the height at which the tool rapids too or from the part.

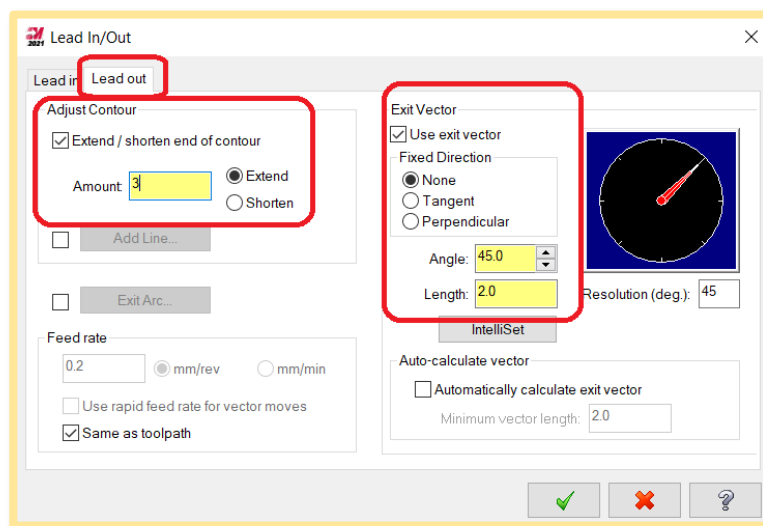
6

- Select the Rough Overlap Parameters to establish how much the tool overlaps the previous cut and specify an Overlap amount of 1.0 as shown.



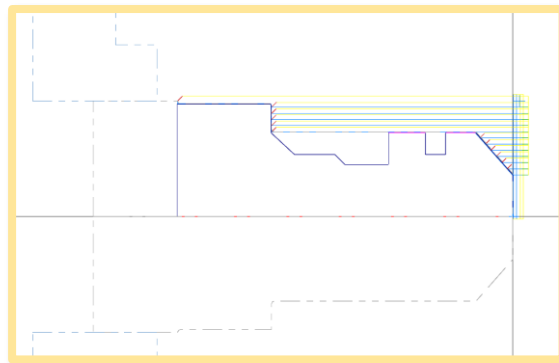
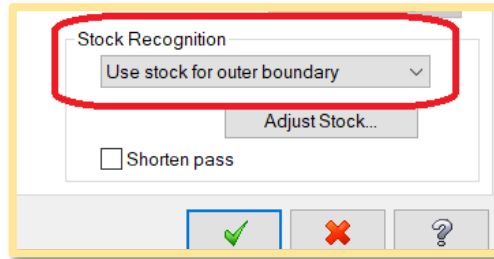
7

- Select the **Lead In/Out** button and choose the **Lead Out** tab to extend the end of the contour as shown in the figure.



8

- In the rough parameters tab, change the **Stock Recognition** to **Use stock for outer boundary**



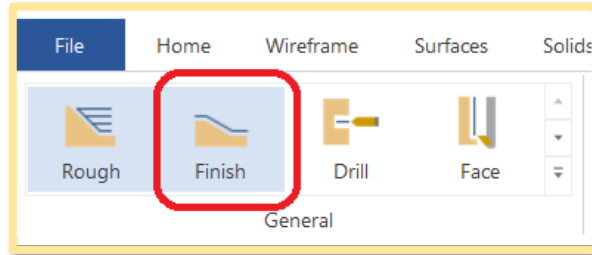


STEP 5 : MACHINING THE STOCK USING 2D CONTOUR TOOLPATH_FINISHING THE PART

Finish toolpaths is typically follow a roughing toolpath contour of the chained geometry.

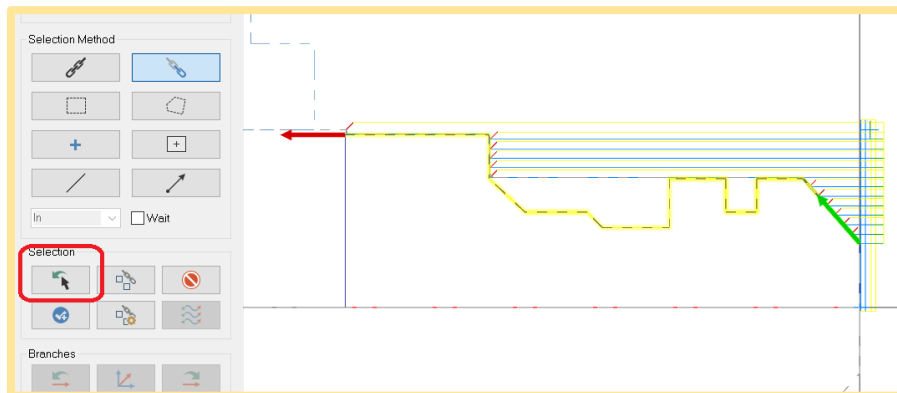
1

- Select the Toolpaths tab in the Operation Manager as shown.



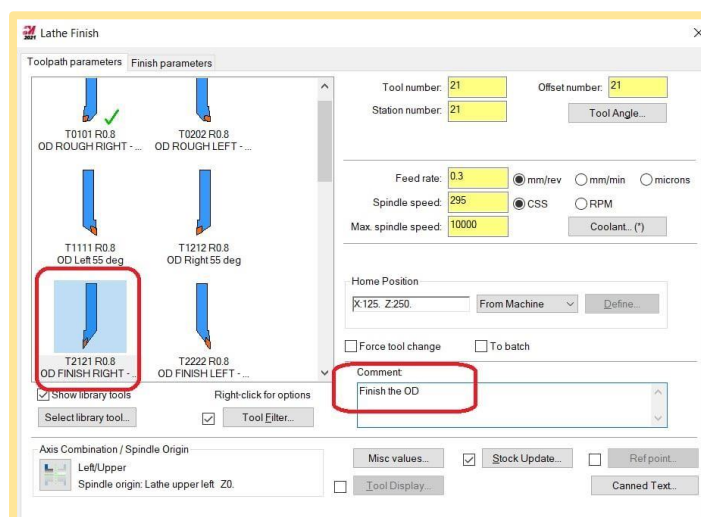
2

- Select the **Last** button in the Chaining dialog box as shown.
- Click **Ok** to exit Chaining Dialog box



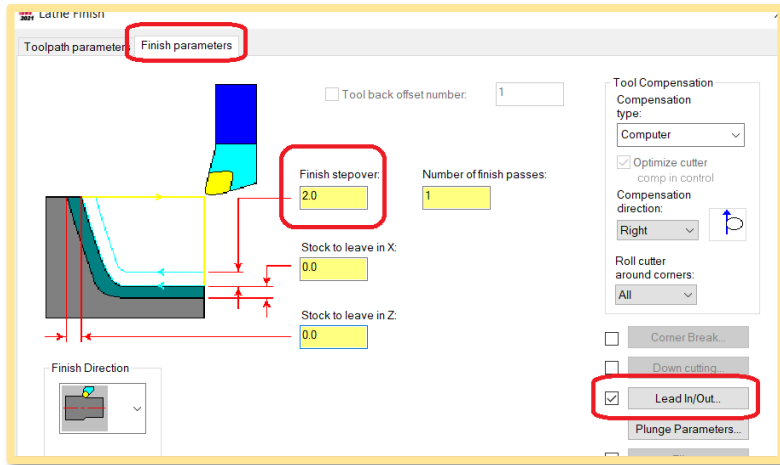
3

- Select the **T2121 OD 55 Degree Right** tool from the tool list.
- Fill the comment as shown in the figure.



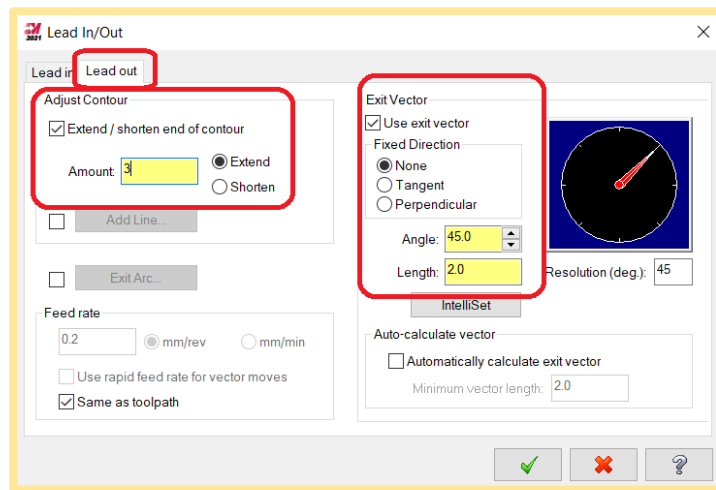
4

- Select the **Finish Parameter** tab and make all of the necessary changes as shown in the figure.



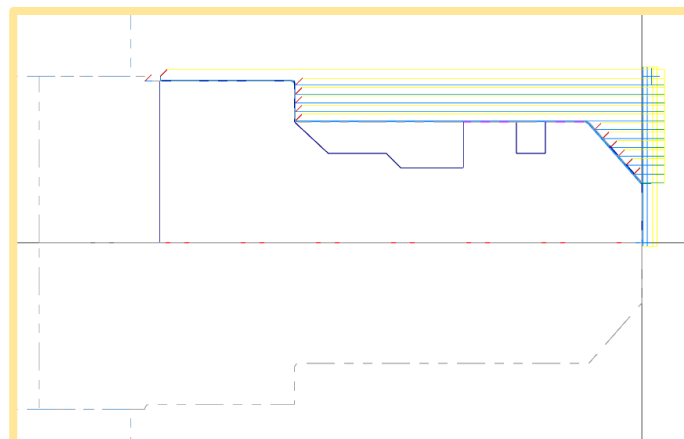
5

- Select the LEAD In/Out, choose the Lead out tab and extend the end of the contour by 3.0 as shown in figure.



6

- Select the Ok button twice to exit the Finish Parameters. The toolpath should appear as shown in figure.





STEP 6 : MACHINING THE STOCK USING 2D CONTOUR TOOLPATH_ GROOVING THE PART

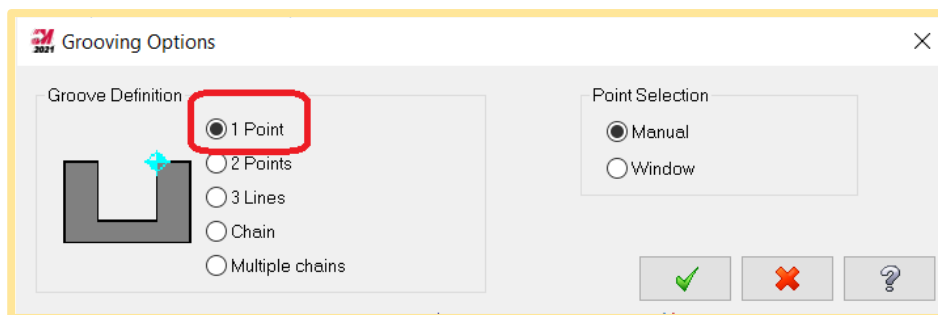
1

- Select the **Groove** in Toolpaths tab in the Operation Manager



2

- Choose 1 Point when the grooving Options dialog box opens up as shown.



1 point allows the user to select points from the graphics area to identify a groove.

2points allows you to specify a groove by indicating the top corner of the groove and the point in the lower opposite corner.

3lines allows you to select three lines from the graphics screen to define the groove shape.

Chain allows you to chain a more complex shape to define a groove by chaining on screen geometry.

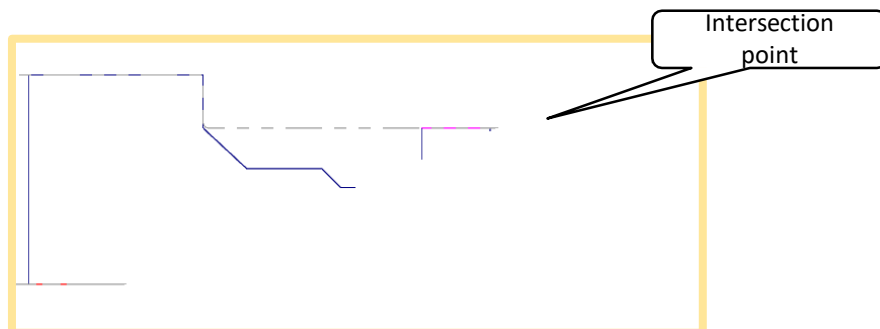
Multiple chains allows you to chain multiple grooves by chaining on screen geometry.

Manual Point Selection allows you to manually select points from the graphics area.

Window Point Selection allows you to create a window in the graphics area and chains all of the points within the window.

3

- Select the **Ok** button to exit the **Grooving Options** box
- Select the **Intersection Point** as shown in figure.

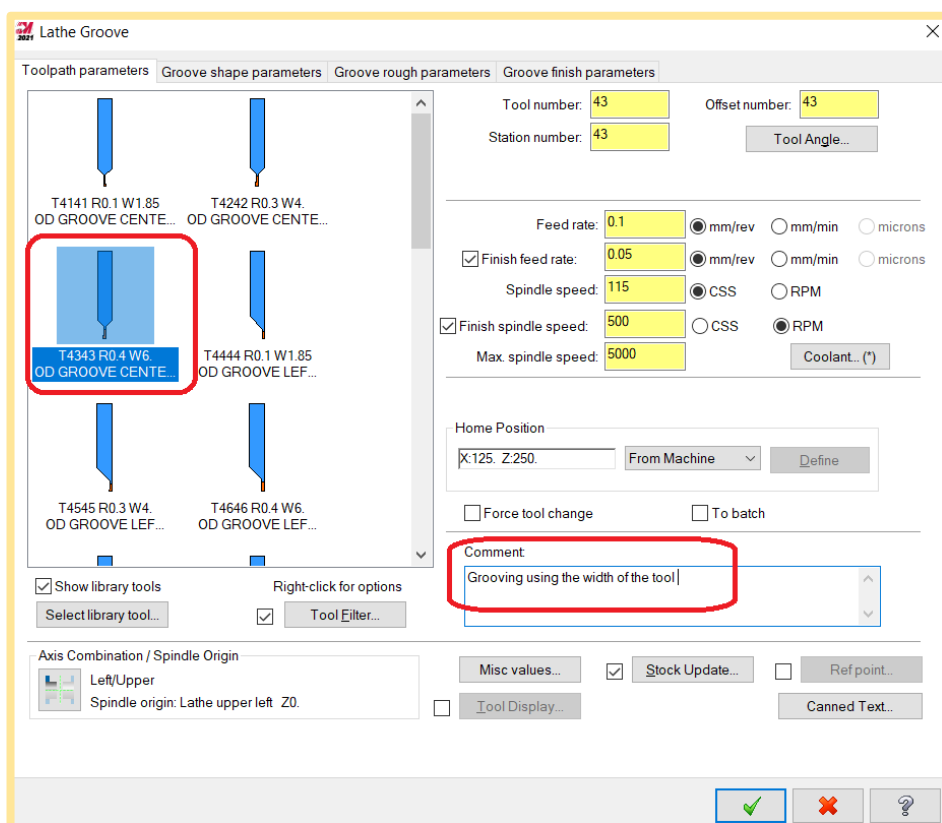


4

- Press **Enter** on your keyboard after you have selected the point.

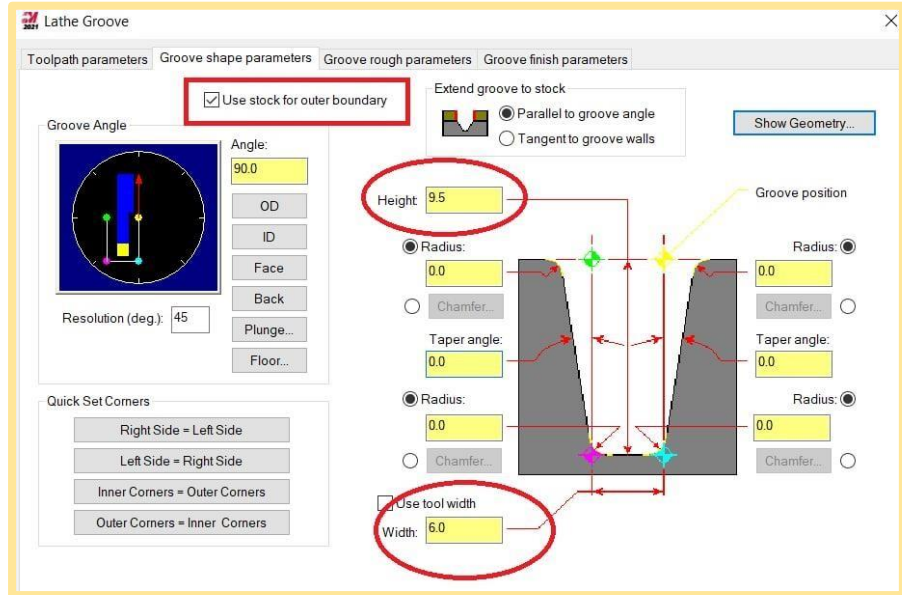
5

- Select the **T4343 : OD Groove Center Wide** tool from the tree list and enter in the comment as shown in Figure.



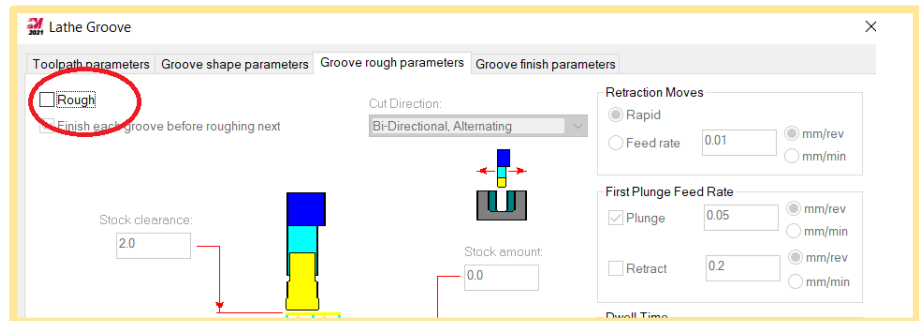
6

- Select the **Groove Shape Parameter** tab and make necessary changes as shown in Figure.



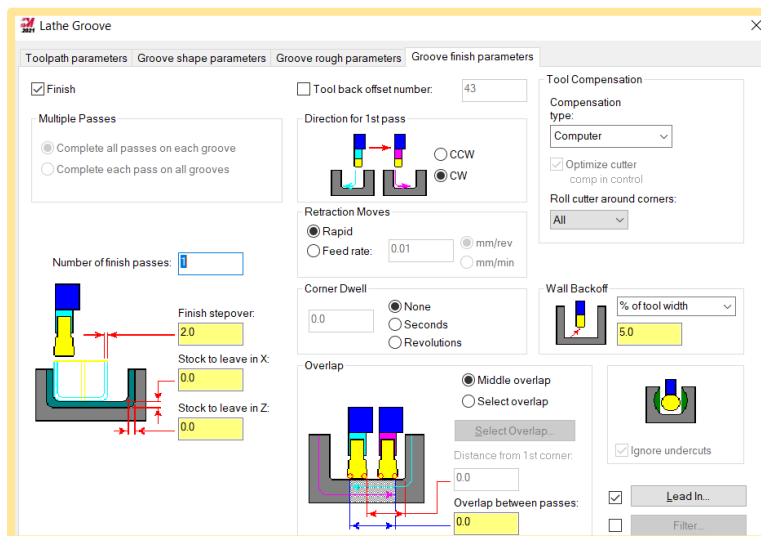
7

- Select the **Groove Rough Parameter** tab and disable the option to **Rough the groove** as shown in the figure..



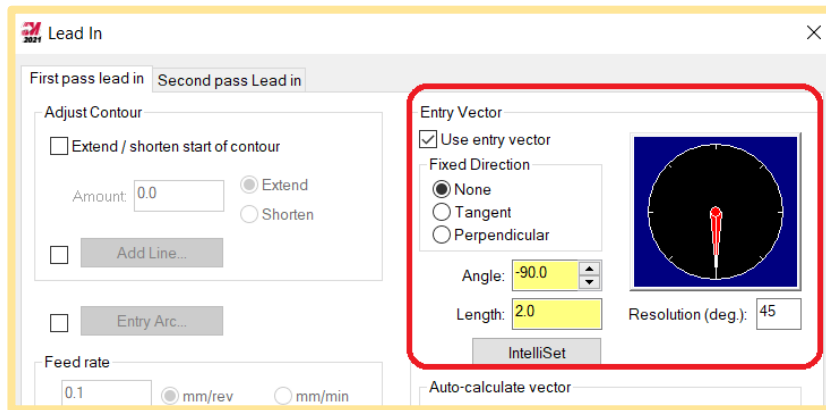
8

- Select the **Groove Rough Parameter** tab and disable the option to **Rough the groove** as shown in the figure..



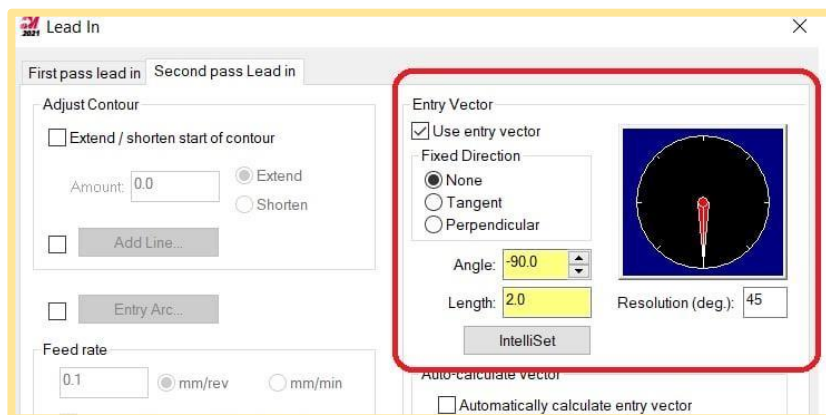
9

- Select the **Lead In** button and change the **First pass lead in** entry vector to **-90** as shown in the figure.



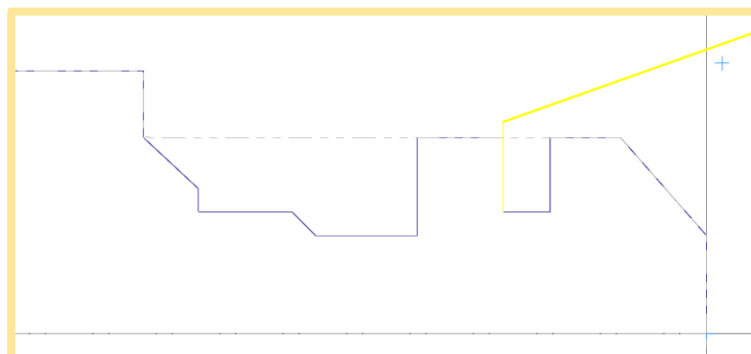
10

- Select the **Second pass lead in** tab change the entry vector to **-90** as shown in the figure.



11

- Select the **Ok** button to exit the **Lead In** dialog box.
- Select the **Ok** button to exit the **Lathe Groove** dialog box and generate the toolpath.
- The toolpath should appear as shown in Figure.





STEP 7 : GROOVE THE PART USING THE SINGLE CHAIN METHOD .

In this step we will create another grooving toolpath, this time defining the groove with a chain and selecting the geometry using the partial chain method.

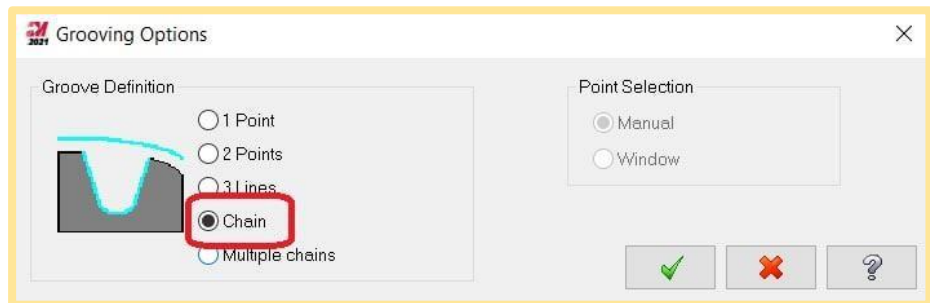
1

- Select the **Groove** in Toolpaths tab in the Operation Manager



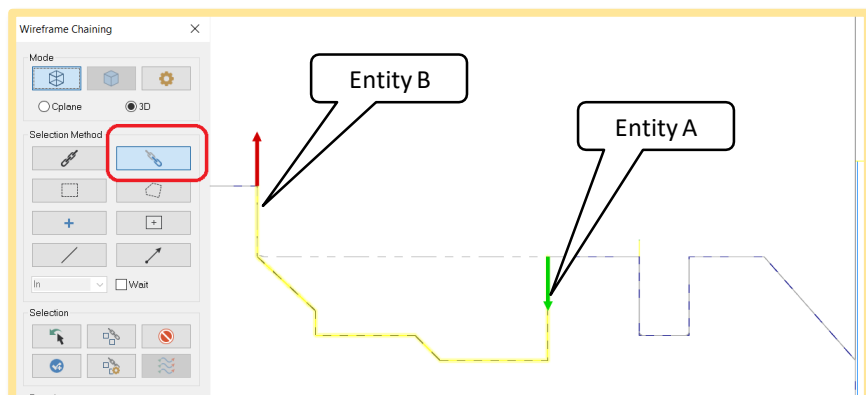
2

- Choose Chain when the Grooving options dialog box opens up as shown below.



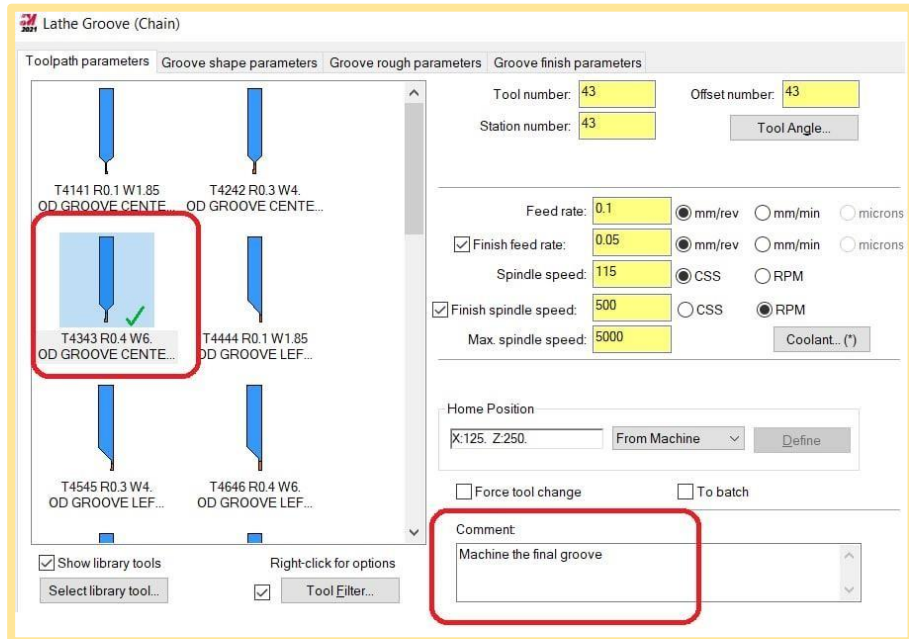
3

- Select the **Ok** button to exit the **Grooving Options** dialog box.
- The **chaining method** should be set to **Partial** by default
- [Select the entry point or chain the inner boundary]. Select **Entity A** as your inner boundary as shown in the figure.
- Select **Entity B** as your last chain as shown.



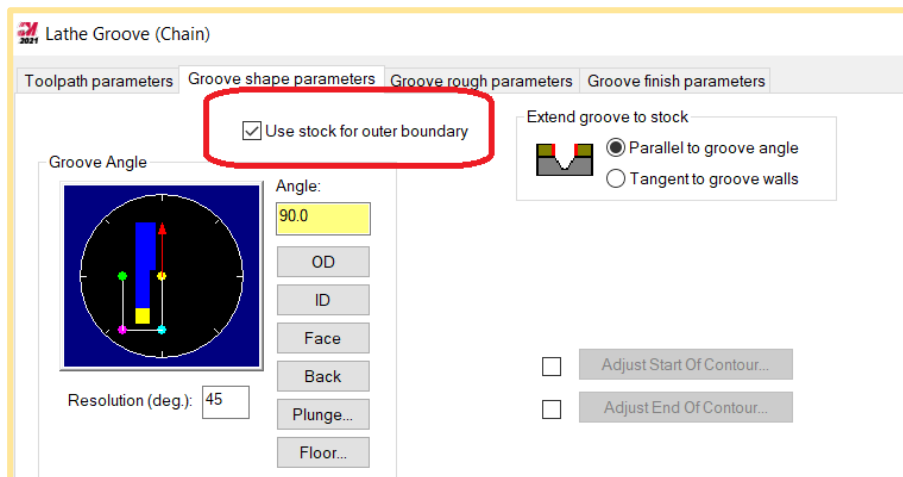
4

- Select the T4343 OD Groove center – Wide tool from the tool list and add a comment as shown in figure.



5

- Select the Groove Parameters tab and enable Use Stock for outer boundary as shown in figure.



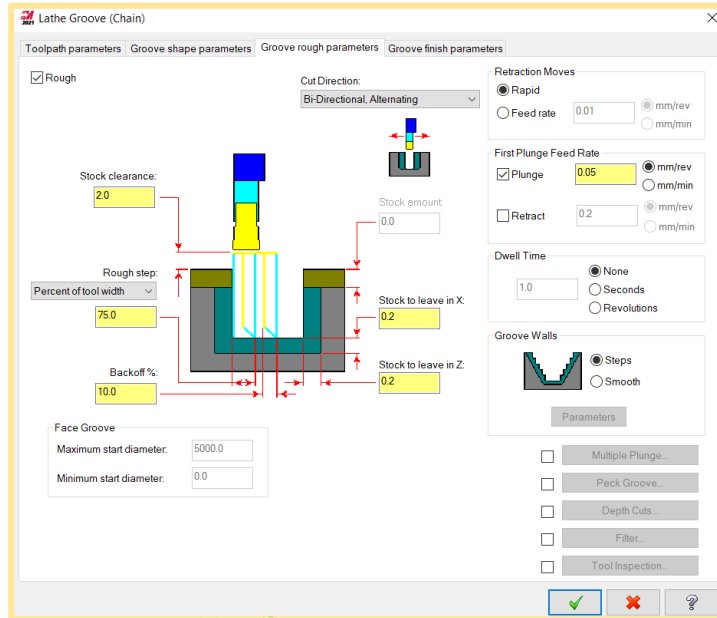
Use Stock for outer boundary when enable Use Stock for outer boundary as shown in Figure.

Parallel to groove angle extends the groove 's inner boundary to the stock , parallel to the groove angle.

Tangent to groove walls extends the groove 's inner boundary to the stock, tangent to the groove walls.

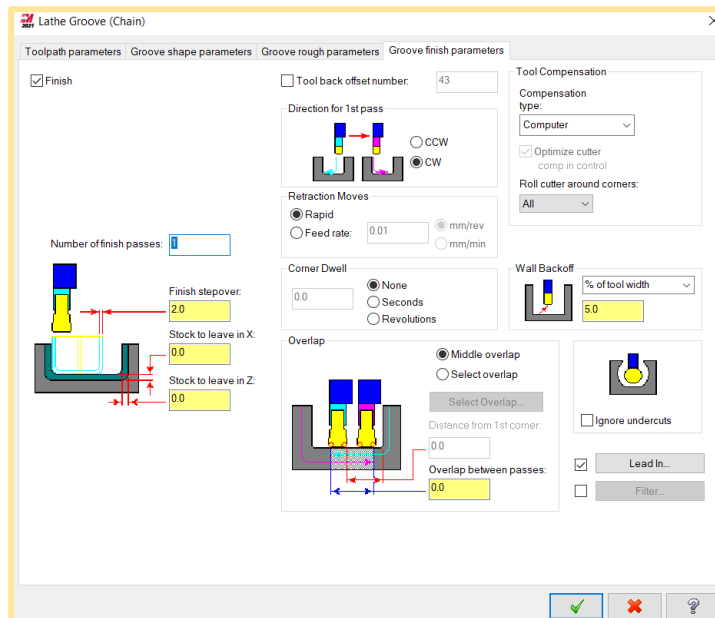
6

- Select the **Groove rough parameters** tab and make any changes as shown in figure.



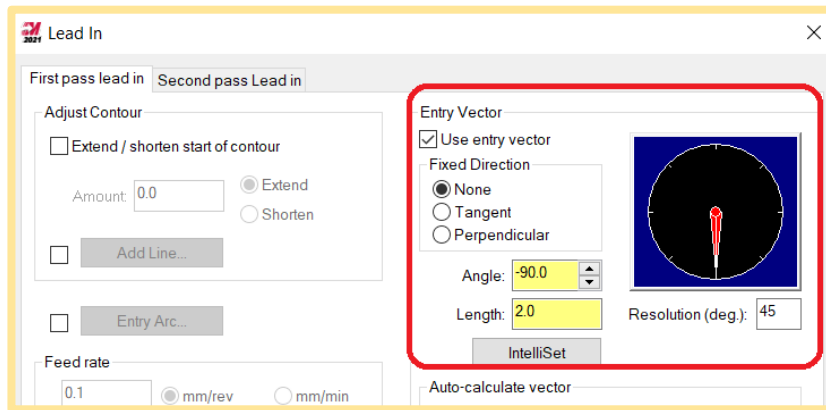
7

- Select the **Groove Finish parameters** tab and make any changes as shown in figure.



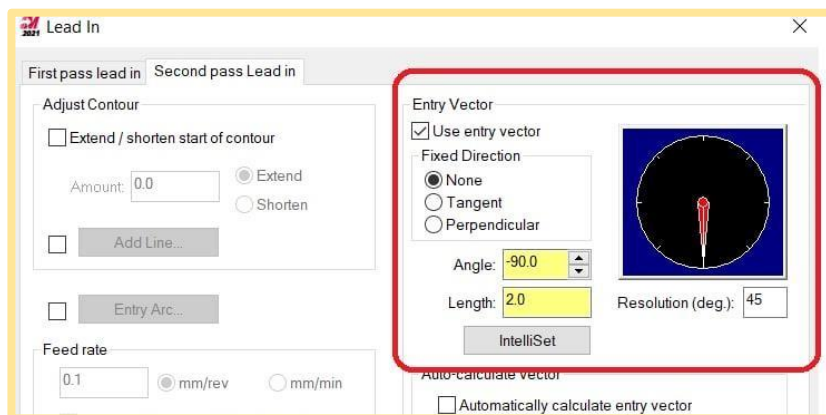
8

- Select the **Lead In** button and change the **First pass lead in** entry vector to **-90** as shown in the figure.



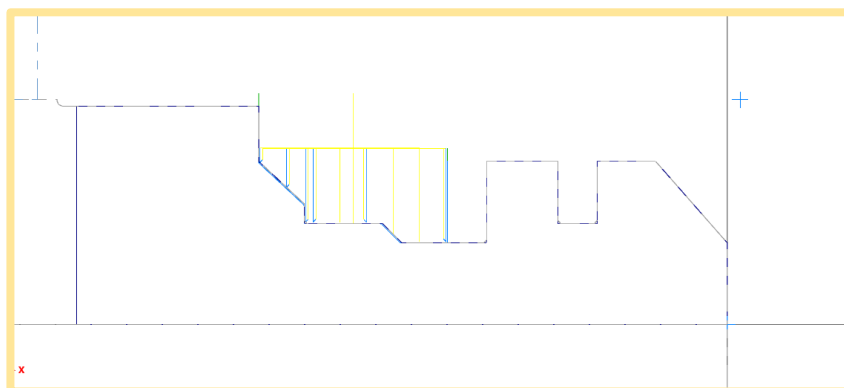
9

- Select the **Second pass lead in** tab change the entry vector to **-90** as shown in the figure.



10

- Select the **Ok** button to exit the **Lead In** dialog box.
- Select the **Ok** button to exit the **Lathe Groove** dialog box and generate the toolpath.
- The toolpath should appear as shown in Figure.

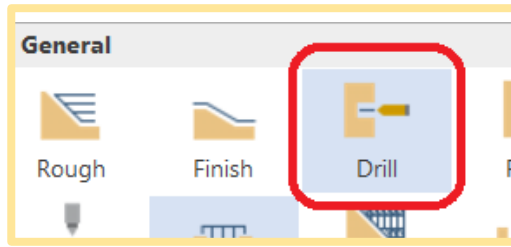




STEP 8 : CENTER DRILL THE PART

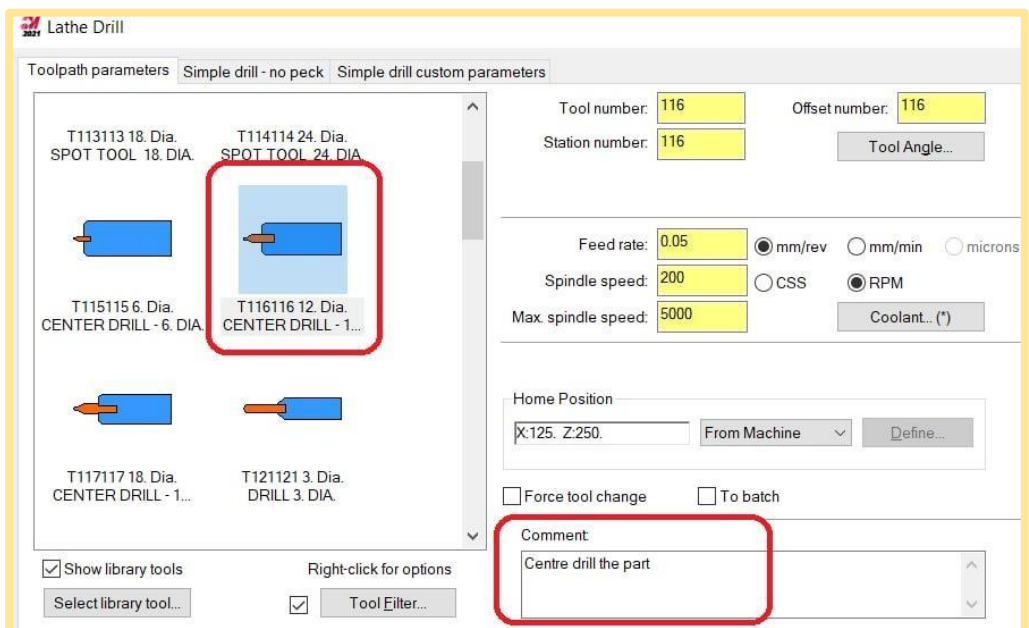
Drill Toolpaths create a drilling toolpath on the face of the part along the centerline. In this step, we will center drill the face before drilling the part to finish size.

- 1 • Select the Drill in Tollpaths.



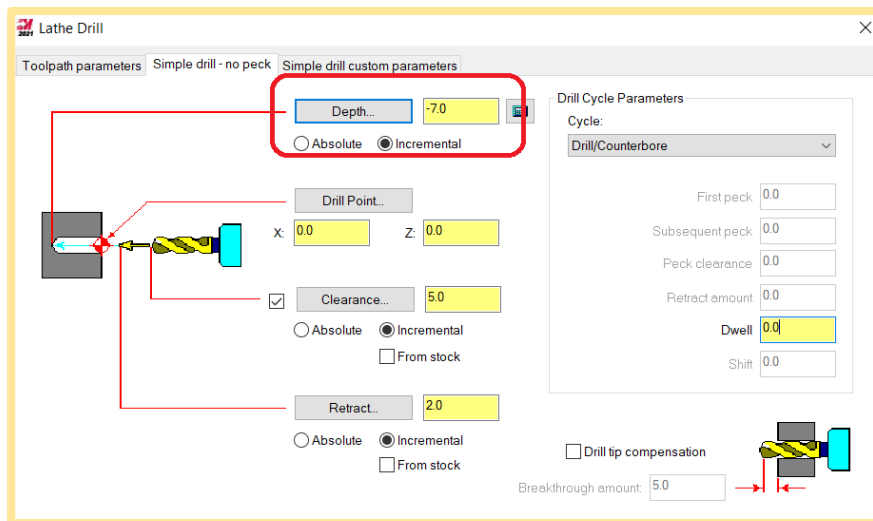
Note : The **Lathe Drill Parameter** dialog box will automatically open, no chaining is needed, because Mastercam Drilling along the centerline to create the toolpath. The drill depth are specified within the dialog box.

- 2 • Select the **T116116 :12 mm Diameter Center Drill** from the tool list and enter in the comment as shown in figure.



3

- Select the **Simple Drill** tab and change the parameters to match the screenshot as shown in figure. .



Depth sets the location of the bottom of the hole.

Drill points allows you to choose the point location where you want to drill

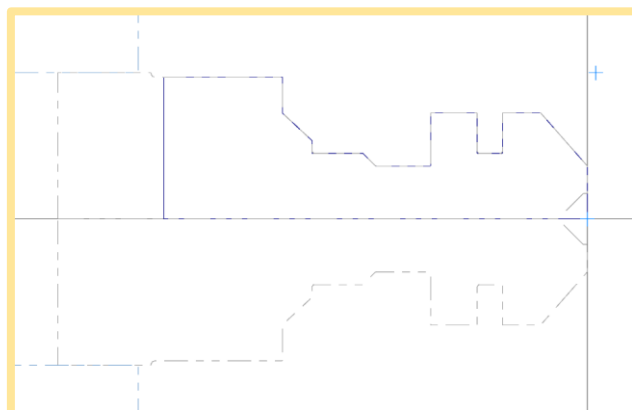
Retract value determines the point at which the tool starts to move with the feed rate towards the stock.

Drill tip compensation automatically adjusts the depth value adding the tip of the drill to it.

Breakthrough amount allows you to add an extra amount that Mastercam adds to the depth for the through holes.

4

- Select the Ok button to exit the Lathe Drill dialog box
- The toolpath should appear as shown in Figure.

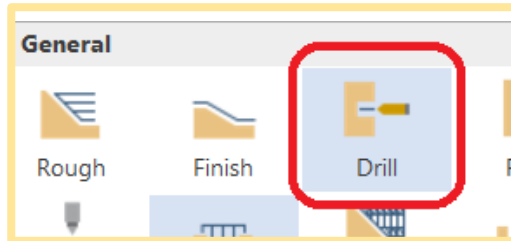




STEP 8 : CENTER DRILL THE PART

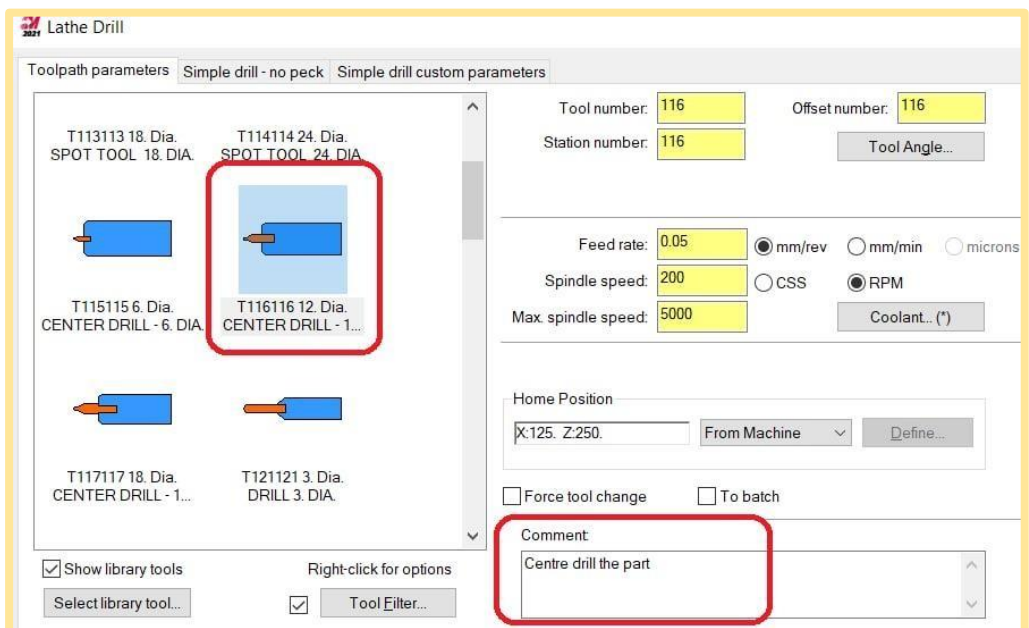
In this step we will create another drilling operation to finish the part

- 1 • Select the Drill in Tollpaths.



Note : The **Lathe Drill Parameter** dialog box will automatically open, no chaining is needed, because Mastercam Drilling along the centerline to create the toolpath. The drill depth are specified within the dialog box.

- 2 • Select the **T116116 :12 mm Diameter Center Drill** from the tool list and enter in the comment as shown in figure.



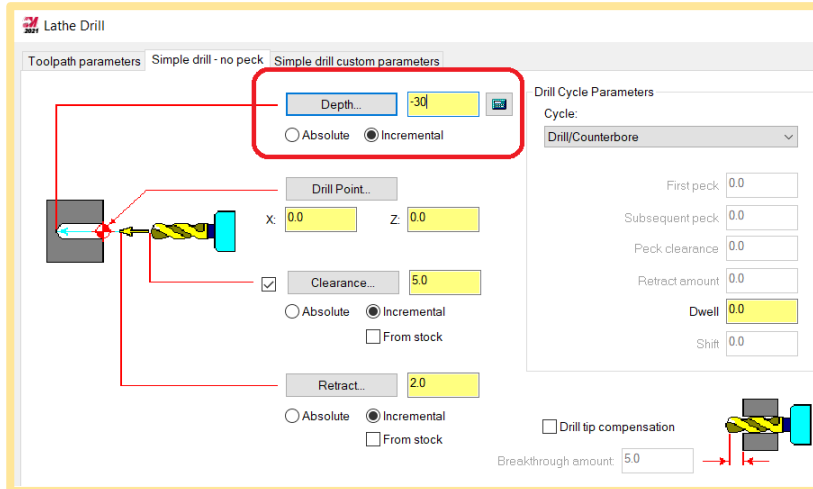


STEP 9 : DRILL THE ANOTHER PART

In this step we will create another drilling operation to finish the part

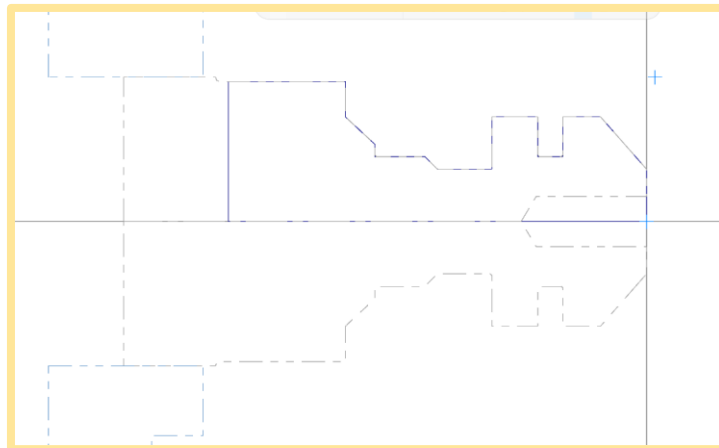
1

- Select the **Simple Drill** tab and change the parameters to match the screenshot as shown in figure.



2

- Select the Ok button to exit the Lathe Drill dialog box.
- The toolpath should appear as shown in Figure



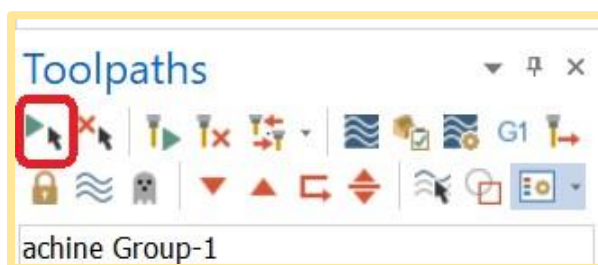


STEP 10 : BLACKPLOT THE TOOLPATHS

Blackplotting shows the path the tools take to cut the part. This display lets you spot errors in the program before you machine the part. As you backplot toolpaths, Mastercam displays the current X, Y and Z coordinates in the lower left corner of the screen.

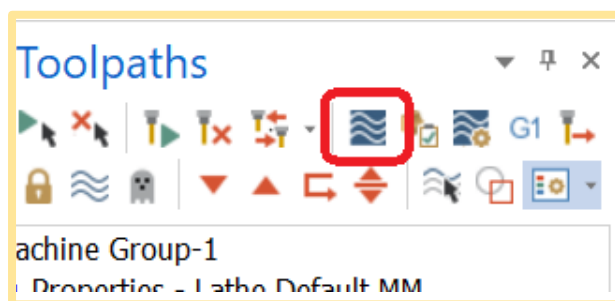
1

- Make sure that you **Select all operations** (signified by the green check mark on the folder icon).



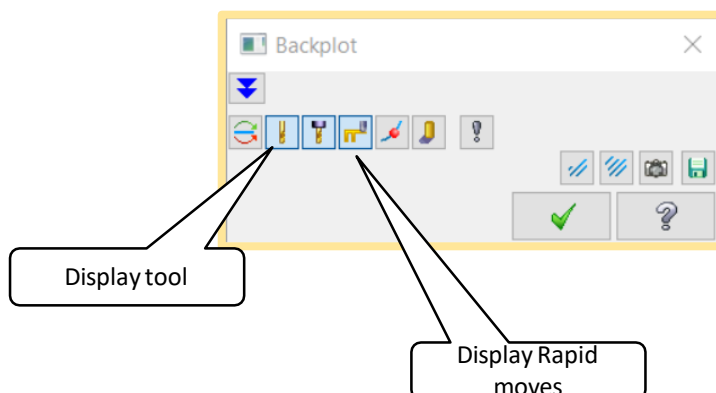
2

- Select the **Backplot selected operations** button as shown.



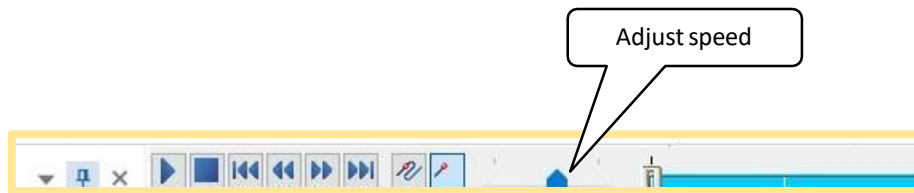
3

- Make sure that you have the following buttons turned on (they will appear pushed down) to see the tool and the rapid moves).



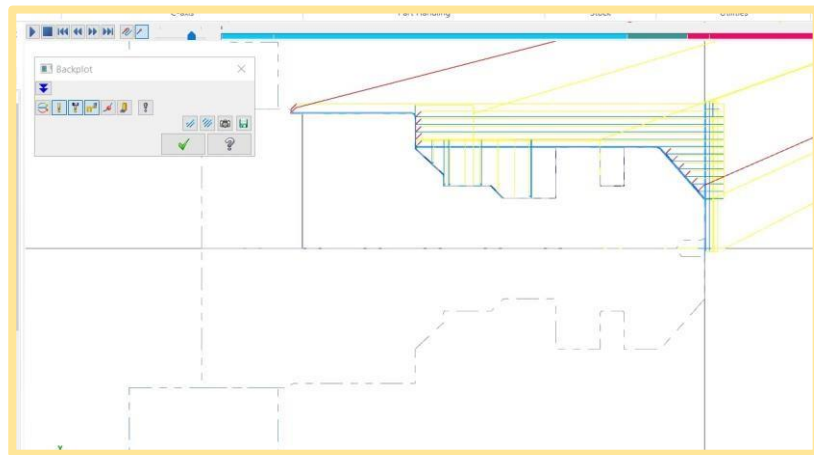
3

- Select the isometric view from the view toolbar to see the stock.
- Select the Fit button



4

- You can adjust the speed of the **blackplot**
- You can step through the **Backplot** by using the **Step Forward** or **Step Back** buttons.
- Select the **Play** button in the **VCR** bar.



5

- Select the **Ok** button to exit the **Backplot**.

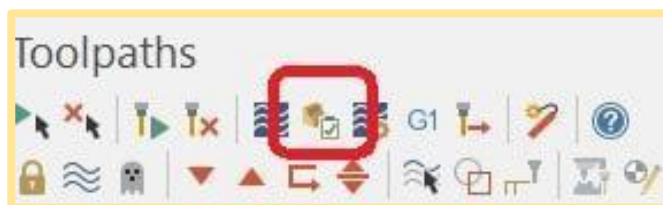


STEP 11 : TOOLPATH VERIFICATION

Toolpath Verification allows you to use solid models to simulate the machining of the part. The model created by the verification represents surface finish and shows collisions if any exist.

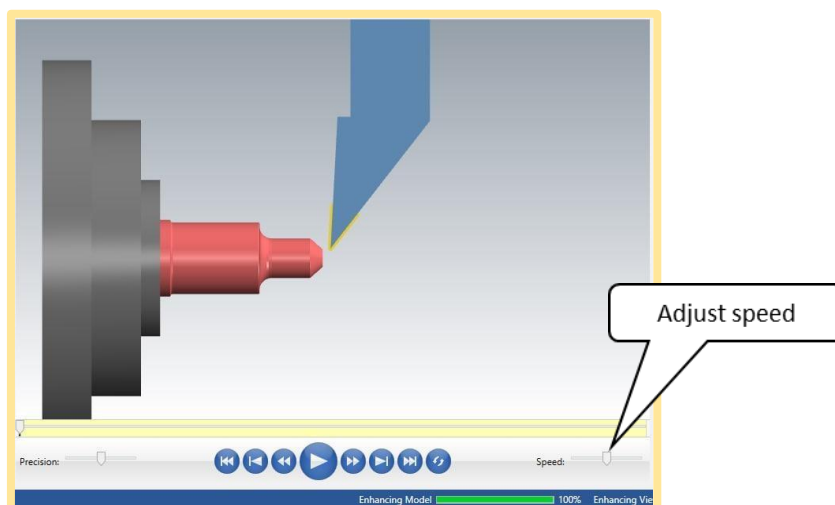
1

- Select the Verify all the operations button as shown.



2

- Use the **Fit** icon to fit the solid to the screen.
- Set the **Verify Speed** by moving the slider bar in the speed control bar
- Select the **Machine** button to start the simulation.



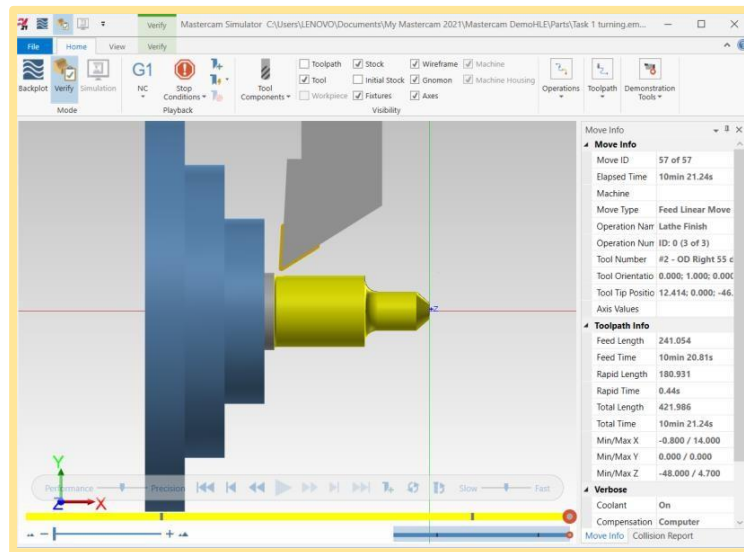
3

- The finished part should appear as shown in the following picture.



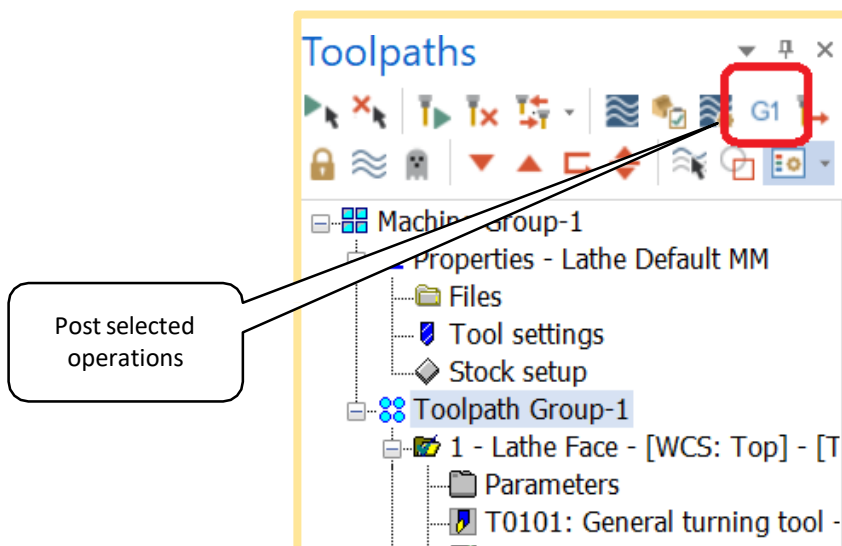
STEP 12 : POST THE PROCESSOR FOR G-CODE FILE

Post Processing refers to the process by which the toolpaths in your Mastercam part files are converted to a format that can be understood by your machine tool's control. A special program reads your Mastercam file and writes the appropriate NC code.



1

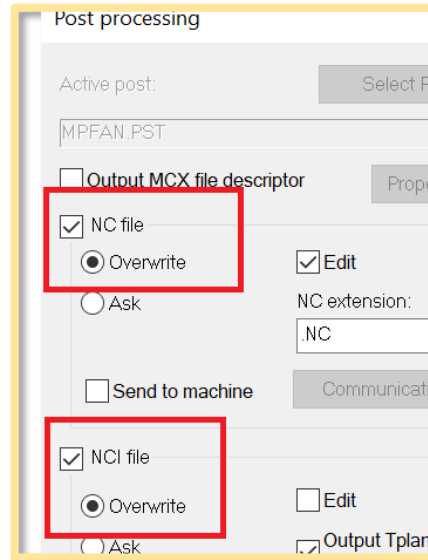
- Click the **Post selected operation** button in the Operation Manager.
- Make the necessary changes as shown in **Post Processing** window in the figure above.



- Note: Make sure all the operations are selected. **Select Overwrite** in the **Operations Manager**.

1

- Set the Post Processing like the Figure.
- Select OK button to continue.



2

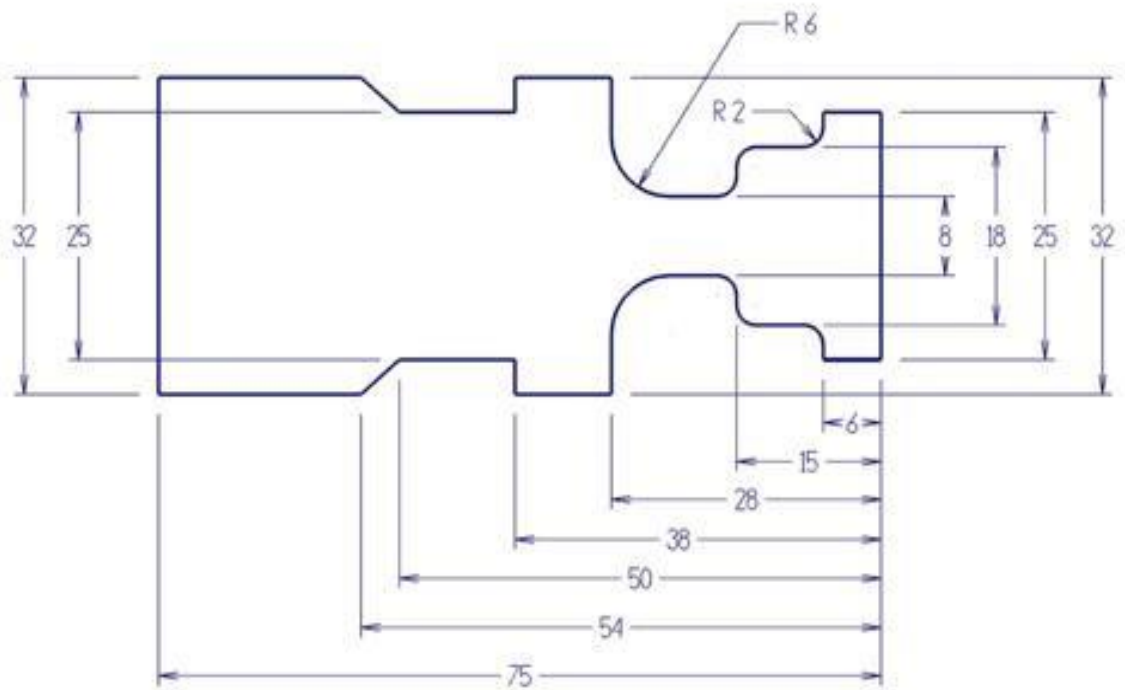
- After generates NC codes, copy all the codes and save in as **Notepad** version.
- You can use this post in the NC code at CNC machine.

```
1  %
2  O0001|
3  (PROGRAM NAME - TURNING 1)
4  (DATE=DD-MM-YY - 02-08-22 TIME=HH:MM - 10:35)
5  (MCX FILE - C:\USERS\HASNY\DOCUMENTS\MY MCAMX9\MCX\TURNING 1.MCX-9)
6  (NC FILE - C:\USERS\HASNY\DOCUMENTS\MY MCAMX9\LATHE\NC\TURNING 1.NC)
7  (MATERIAL - ALUMINUM MM - 2024)
8  G21
9  (TOOL - 1 OFFSET - 1)
10 (OD ROUGH RIGHT - 80 DEG.  INSERT - CNMG 12 04 08)
11 ( FACE THE PART )
12 G0 T0101
13 G18
14 G97 S3126 M03
15 G0 G54 X28. Z2.083 M8
16 G50 S3600
17 G96 S275
18 G99 G1 X-1.6 F.25
19 G0 Z4.083
20 X28.
21 Z1.167
22 G1 X-1.6
23 G0 Z3.167
24 X28.
25 Z.25
26 G1 X-1.6
27 G0 Z2.25
28 X28.
29 Z0.
30 G1 X-1.6
31 G0 Z2.
32 ( ROUGH THE OD )
33 M9
34 X19.226
35 Z4.7
36 G1 Z2.7
37 Z-19.8
38 X20.4
39 G18 G3 X22.4 Z-20.8 K-1.
40 G1 Z-47.8
41 X24.
42 X26.828 Z-46.386
43 G0 Z4.7
44 X14.453
45 G1 Z2.7
46 Z-19.059
47 G2 X18.4 Z-19.8 I1.973 K2.259
48 G1 X19.726
49 X22.555 Z-18.386
```



PRACTICAL TASK 4

- **Create the geometry** from Practical Task 2 exercise use commands rectangle, line endpoint, offset, fillet entities, chamfer entities, trim/divide and delete entities.
- **Set the machine properties** including stock and chuck setup. **Define** the facing, roughing, and finishing operation to the OD of the part.
- **Setup** lead in/out and extend the toolpath so there **is no collision** between chuck, stock and cutting tool.



TITLE	TURNING: PRACTICAL TASK 4	DEPT	MECHANICAL ENGINEERING
POLITEKNIK PORT DICKSON		COURSE	DJF41042 (CAD CAM)
		MATERIAL	ALUMINIUM 6061
		UNIT	METRIC (MM)



QUESTION AND ANSWER

CAD designer	DNC	geometric model
assemblies	Numerical Control	Internal Micro Processor
physical	Solid Modeling (SW)	

Question A: Select the suitable answer for all the statements.

1. _____ can be defined as an operation of machine tools by the means of specifically coded instructions to the machine control system in which the machine operated automatically.
2. _____ is not a part of the control unit.
3. CNC system uses an _____ that contains memory registers storing a variety of routines that are capable of manipulating logical functions.
4. A _____ describes the shape of _____ or mathematical object by means of geometric concepts.
5. _____ is a natural extension from the use of essentially 1D entities (curves) or 2D entities (surface) to the modeling of shape using 3D solids.
6. _____ must expert to create curve, surface and solid in order to create complex solid models.
7. Computerized environment is a designer's info can be explain comprehensively and clearly by CAM systems be able to describe the parts, _____, raw material & manufacturing requirements



QUESTION AND ANSWER

Questions B: State the statement which is true or false.

1. Drafting is about - planar cross sectioning, production of shaded images, creation of exploded views for technical illustration.

2. Solid or volume model is a figure on the left shown of a wire frame model.

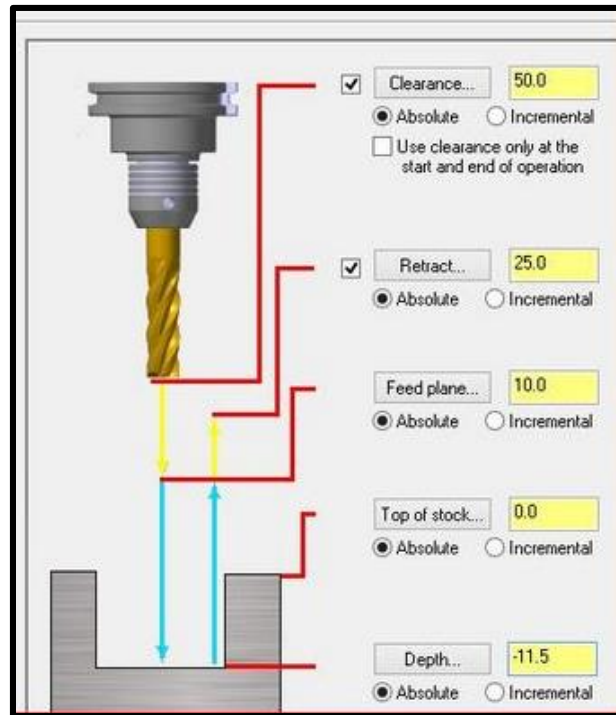
3. 2-dimensional drafting is ability to provide all the information required for manufacturing applications.

4. Projection is represented by a coordinate system, consisted of world coordinate system (WCS) and user coordinate system (UCS).



QUESTION AND ANSWER

Questions C: Based on the figure below, categorize the descriptions with the correct answers.



Descriptions	Linking parameters
<ul style="list-style-type: none"> ○ Set the heights of the material in the Z-axis 	<p><i>Top of stock</i></p>
<ul style="list-style-type: none"> ○ Determines the final machining depth that the tool descends into the stock 	
<ul style="list-style-type: none"> ○ Set the height that the tool moves up to before changing to the plunge rate to enter the part 	
<ul style="list-style-type: none"> ○ Set the height at which the tool moves to and from the part 	
<ul style="list-style-type: none"> ○ Sets the height that the tool moves up to before the next tool pass 	



KEMENTERIAN PENDIDIKAN TINGGI
JABATAN PENDIDIKAN POLITEKNIK DAN KOLEJ KOMUNITI

e ISBN 978-967-2897-92-7



9 789672 897927