KEMENTERIAN PENGAJIAN TINGGI

POLITEKNIK MALAYSIA
PORT DICKSON

CADCAM

BY MASTERCAM

MASTERCAM X9

POLYTECHNIC APPROACH

1ST EDITION

POLITEKNIK
MALAYSIA
PORT DICKSON

POLITEKNIK MALAYSIA
PORT DICKSON

DJF41042
CAD CAM

ACKNOWLEDGEMENT

PATRON
Mohamad Isa Bin Azhari
Director, Politeknik Port Dickson

ADVISORS
Dr. Nor Haniza Binti Mohamad
Deputy Director (Academic), Politeknik Port Dickson
Razali bin Bakri
Head of Mechanical Engineering Department, Politeknik Port Dickson

EDITOR
Mohd Azli bin Kasan
Head of Mechanical Programme, Politeknik Port Dickson

FACILITATORS
Nin Hayati Binti Mohd Yusof
Che Azlina Binti Che Norohoseni
Ragunathan a/l Jayaraman

WRITERS
Hasny Binti Abdul Jalil
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.

Perpustakaan Negara Malaysia Cataloguing-in-Publication Data (after eisbn is received)

Cataloguing Information (to be informed)

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

AUGUST 2020

ACKNOWLEDGEMENT

First of all, we gratefull to Almighty Allah S.W.T for established us to complete this book.

This book is written based on the latest syllabus contents of MasterCam for topic Milling Process. This topic is relevant to student 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 are involved directly in the writing of this book

TEAM OF WRITERS

1. HASNY BINTI ABDUL JALIL (Politeknik Port Dickson)

2. MOHD HAMDAN BIN ABDUL RAZAK (Politeknik Port Dickson)

ABTRACT

In teaching and learning sessions, practical work also serves as a application to the theory presented in class. This study aims to identify the effectiveness by using the eBook Training Tutorial Milling 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.

CADCAM
Mastercam

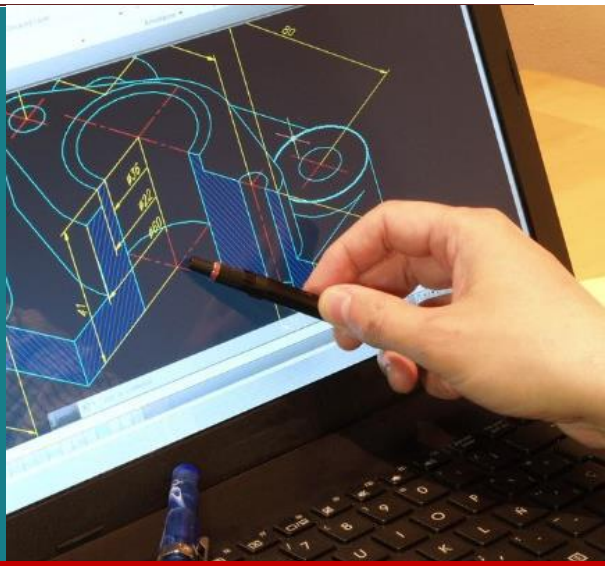
TRAINING TUTORIAL MILLING CONTENTS

TRAINING TUTORIAL MILLING CONTENTS

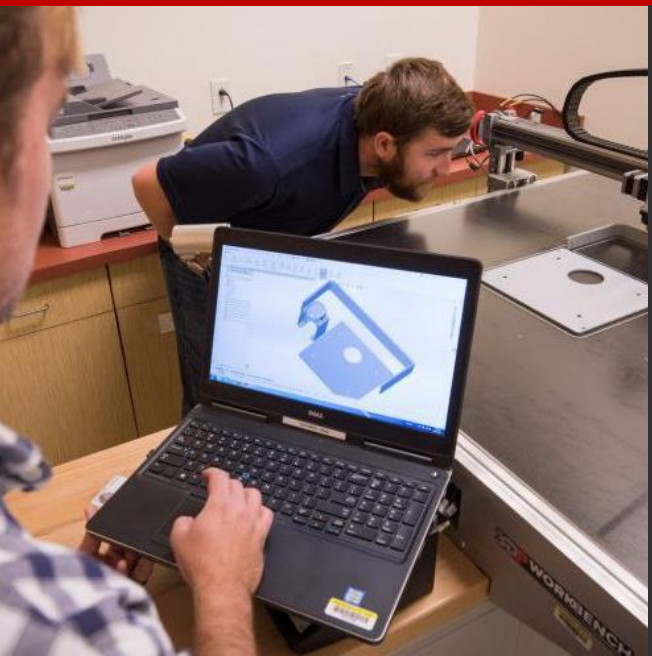
CHAPTER 1
INTRODUCTION
CAD CAM

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.



COMPUTER AIDED DESIGN - CAD



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

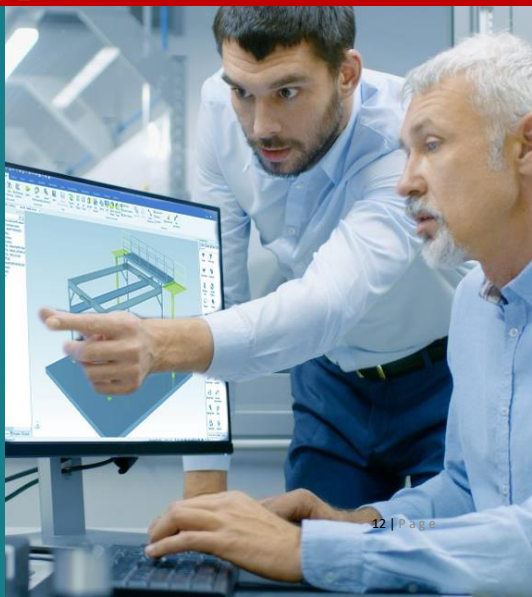
CAM can be defined as the effective utilization of computers in direct manufacturing process control & monitoring or indirect manufacturing operation support.

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.

CAM is the use of computer and computer technology to help in all phases of manufacturing a product, including process and production planning, management, machining, quality control and scheduling.

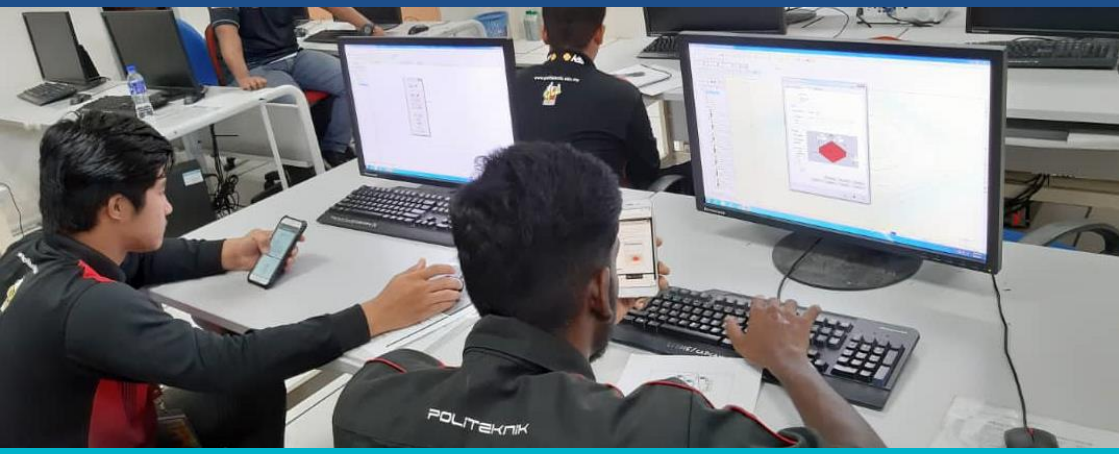
COMPUTER AIDED MANUFACTURING - CAM

Combination of CAD/CAM system allows the transfer of information from design into CAM system. The database elaborated in CAD is kept store and processed further by CAM into the necessary data. This process are included the instructions for operating and controlling production, machinery, automated testing, material handle equipment and inspection for production quality.



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.

CHAPTER 2
NUMERICAL CONTROL

NUMERICAL CONTROL PROGRAMMING

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

CNC

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.

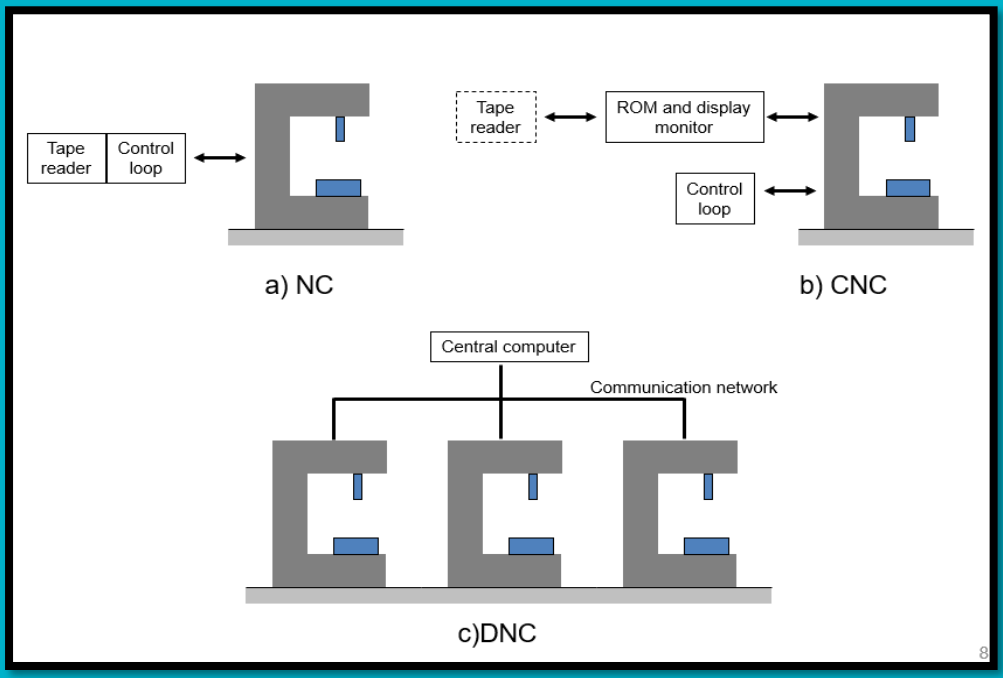
CNC is useful for control the motions of the work piece or tools and the input parameters (feed rate, depth of cut, etc).

DNC

Distributed Numerical Control is not included as a part of the control unit. DNC networking are required when CAM programs are to run in CNC machine control unit. 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.

NC

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


Tape reader
Control loop
a) NC
Tape reader
ROM and display monitor
Control loop
b) CNC
Central computer
Communication network
c)DNC

CNC MACHINE

Numerical Control (NC) is 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

Miscellaneous Functions, M

M00 – Program Stop
M01 – Optional Stop
M02 – End of Program
M03 – Spindle Start Clockwise
M04 – Spindle Start Counter-Clockwise
M05 – Spindle Stop
M06 – Tool Change
M07 – Mist Coolant On
M08 – Flood Coolant On
M09 – Coolant Off
M10 – Clamp
M11 – Unclamp
M13 – Spindle Start Counter Coolant on
M14 – Spindle Start Counter-Clockwise, Coolant on
M30 – End of tape, rewind

CHAPTER 3
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 equipments. 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).

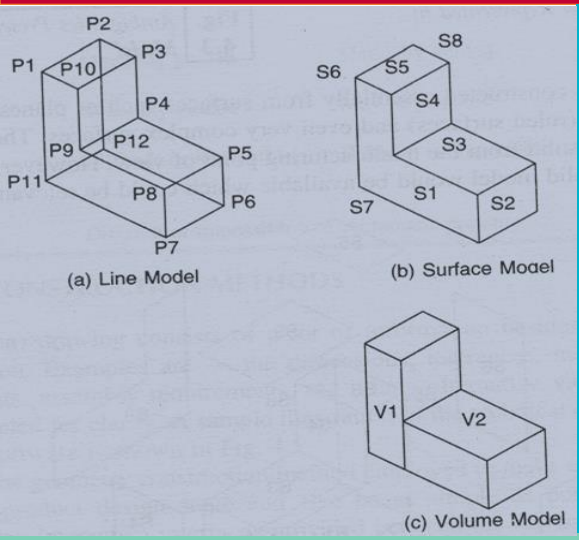
PROCESSING EQUIPMENT

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

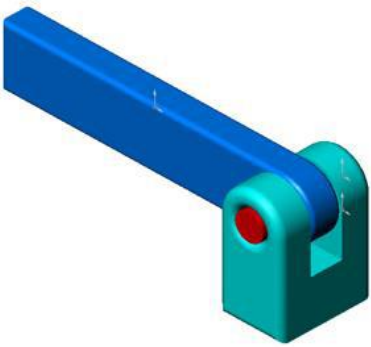
CHAPTER 4
GEOMETRIC MODELLING

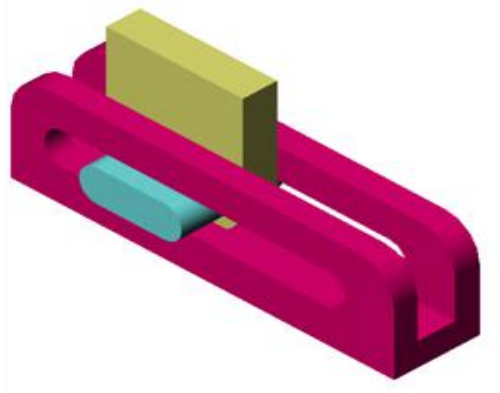
GEOMETRIC MODELLING - GM



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.

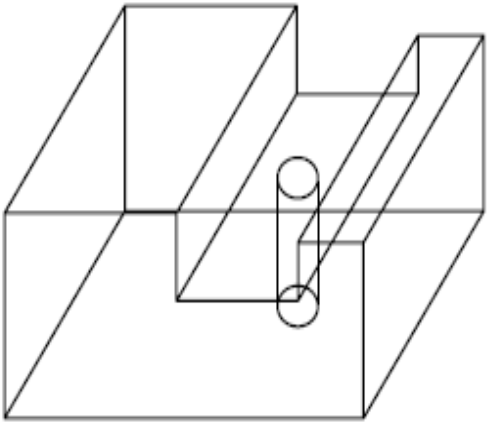
A geometric model should represent its corresponding object, unique and complete to all engineering functions, from documentation to engineering analysis to manufacturing.





Goal of geometric model is to create solid models begin with points, lines, curves, extend the curves to create surfaces and surfaces to create solids..

GEOMETRIC MODELLING - GM

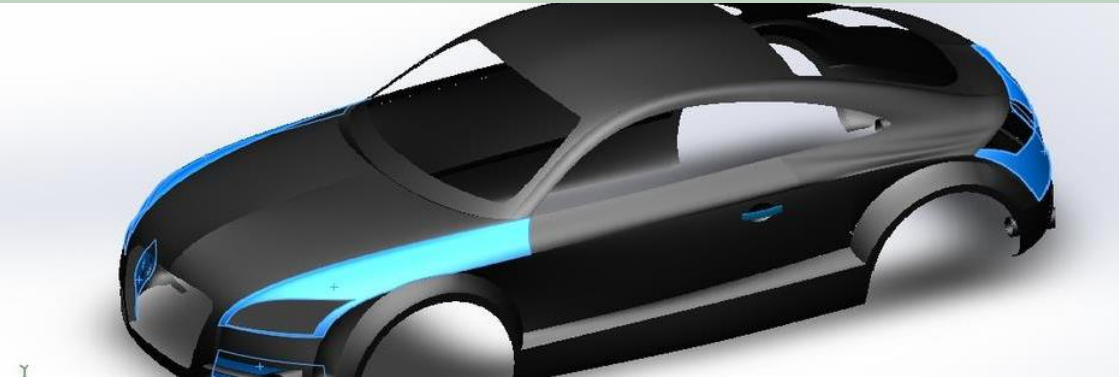


The **wire frame** represents a 3rd dimension of the techniques used for draughting, in view of the simpler manipulation methods.

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 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).

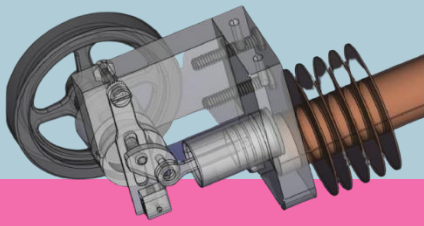


GEOMETRIC MODELLING - GM

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:



PRIMITIVES

Primitives are point and 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.

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:

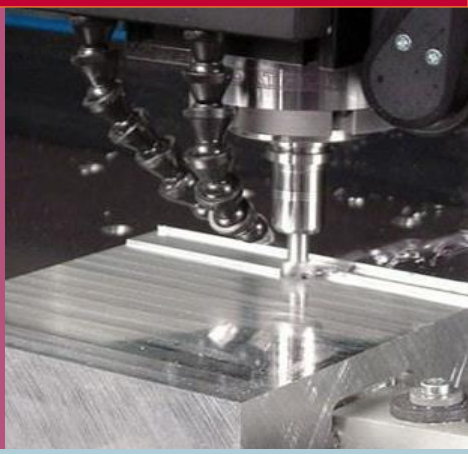
Extruded : use to create solid models of 2.5D objects with uniform thickness.

CHAPTER 5
MILLING PROCESS

CADCAM
Mastercam

MILLING PROCESS

Milling is a machining operation in which a workpart is cutting by a rotating cylindrical tool with multiple cutting edges. The axis of rotation of the cutting tool is perpendicular to the direction of feed.



The machine tool that traditionally performs this operation is a milling machines. Milling is an interrupted cutting operation; the teeth of the milling cutter enter and exit the work during each revolution.

Mastercam is a engineering software that used widely in manufacturing industry. This software is provided CAD and CAM function in one of all. It is used to drive CNC machines for optimized productivity and efficiently.

MILLING PROCESS

Scan me!!

Milling is machining process which a typically used to produce parts by provide a rotating spindle for the cutter and have 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 a 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. Various machine tool design commonly used as a secondary process to add or refine features on parts that were manufactured using a different process. Due to the high tolerances and surface finishes that milling can offer, it is ideal for adding precision features to a part whose basic shape has already been formed.

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

CHAPTER 6
Overview of Practical Task

OVERVIEW OF STEPS FOR MAKING THE FINAL PART OF MILLING :

Drafting the Cad Model:

- The student will checking 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 choosing how to create the geometry by using the Mastercam.

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

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

Determine the necessary Toolpaths to machine the part:

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

Backplot and Verify the task:

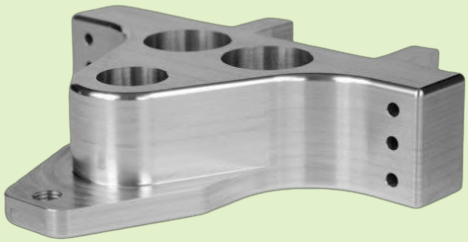
- The Backplot is produce to shows a path of the tools take up to cut the desired part. It is will show the cutting tool motion and toolpath display.

- The Verify it is simulator features that will be used when you attempting to spot the error in the program. It is simulation the toolpath for your analyze and verification before machining the part.

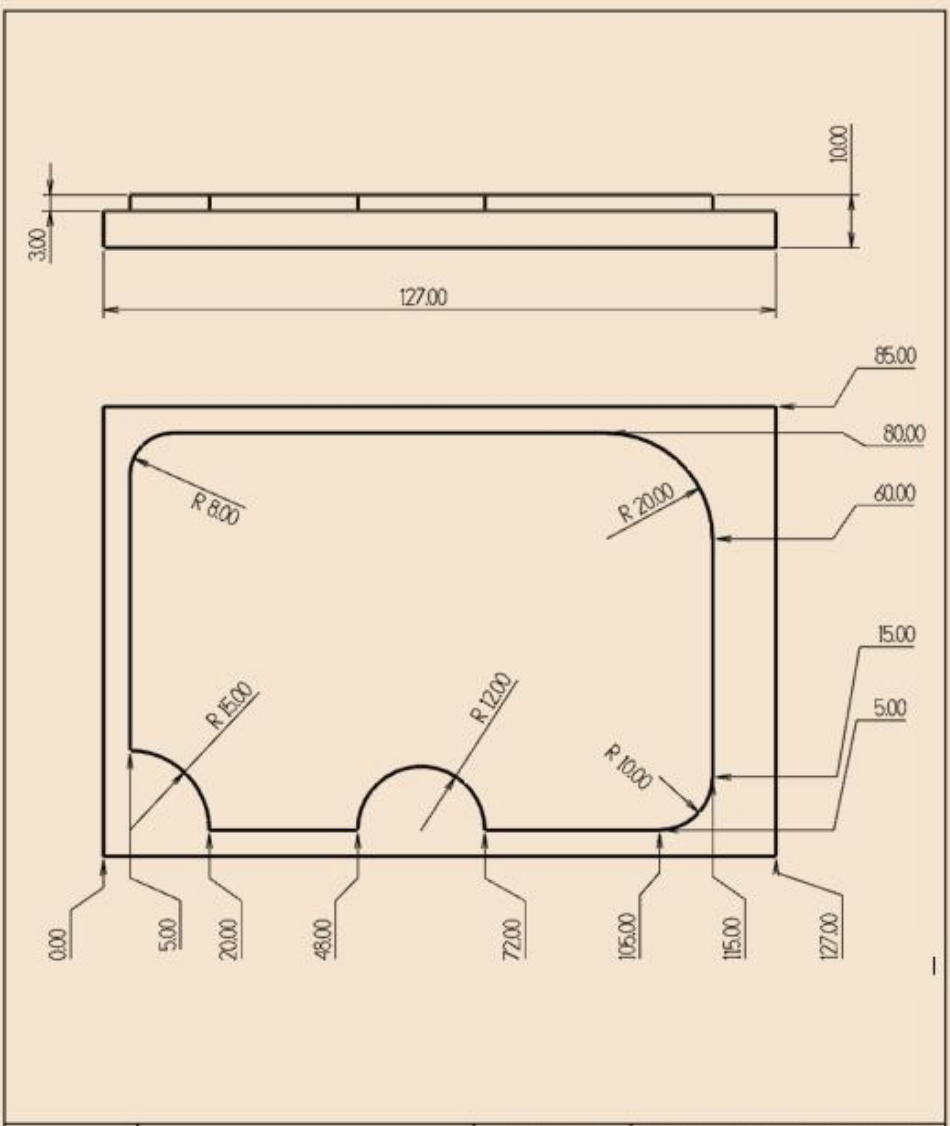
Generate :

- The G and M code will be processing after student choose to posting a file that obtain the NC file coding.

- When the student complete all the process toolpath of operations., the G-code will be generating and ready for machining used.



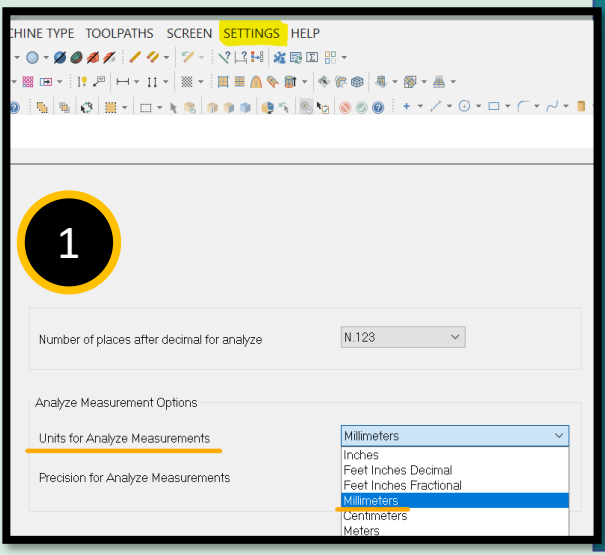
PRACTICAL TASK 1



TITLE	PRACTICAL TASK 1	DEPT	MECHANICAL ENGINEERING
		COURSE	DJF41042
POLITEKNIK PORT DICKSON		LAB	CAD CAM
		UNITS	METRIC (MM)

CADCAM Mastercam

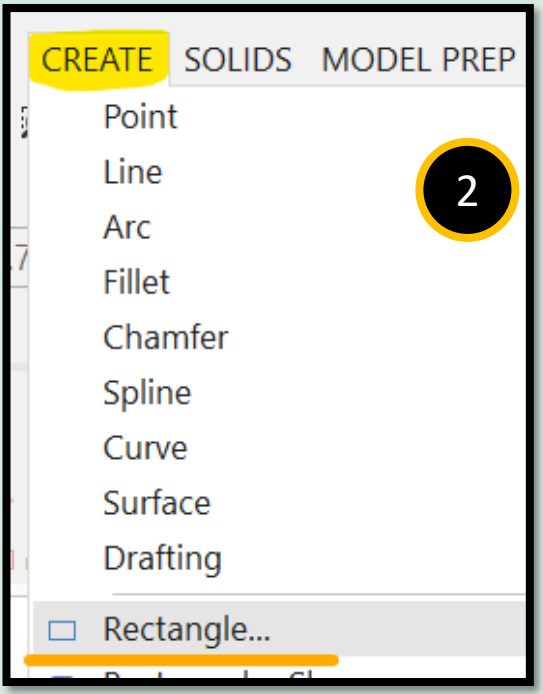
STEP 1: CREATE A GEOMETRY



- Open Mastercam software
- From toolbar, click "SETTING"
  - Click "Configuration"
  - Select unit "milimeters"
- Click ok
- Then select shortkey "F9" for show coordinates axes

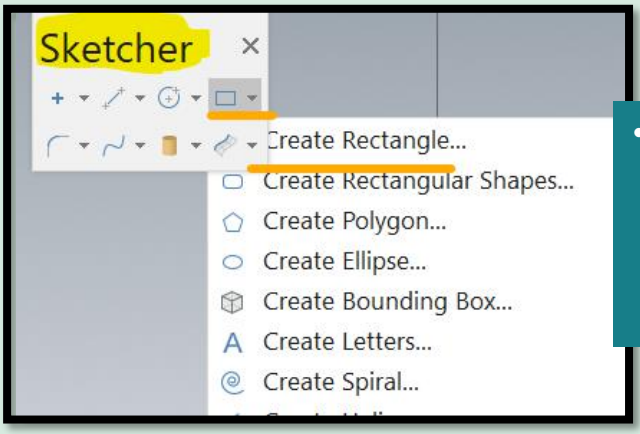
Draw a picture that has been given.

- From toolbar, select CREATE
  - Rectangle



CADCAM Mastercam

STEP 1: CREATE A GEOMETRY



3

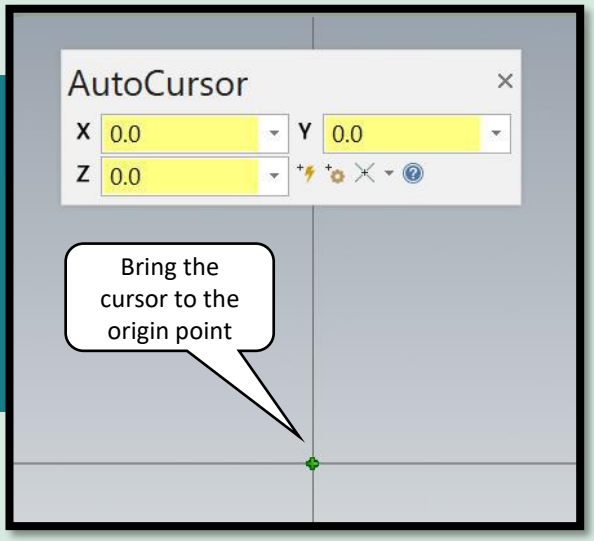
- Or you can use from SKETCHER feature
  - Create Rectangle

4

- Select position of corner
  - AutoCursor (0,0,0)

Or

- Bring the cursor to the origin point and click.

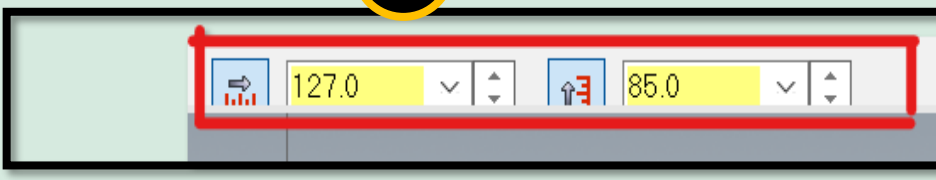


CADCAM Mastercam

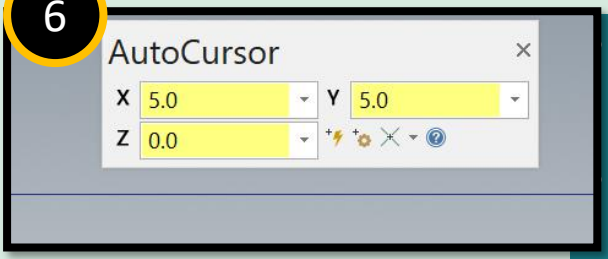
STEP 1: CREATE A GEOMETRY

- In a ribbon bar, setting the **width** and **height**
- Click ok

5



6



Again, select SKETCHER feature

- Create Rectangle

Select position of corner

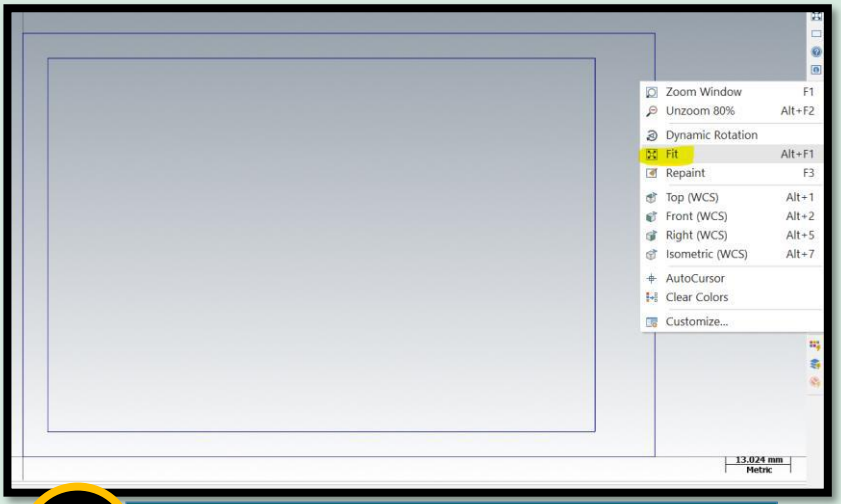
- AutoCursor (5,5,0)

CADCAM Mastercam

STEP 1: CREATE A GEOMETRY

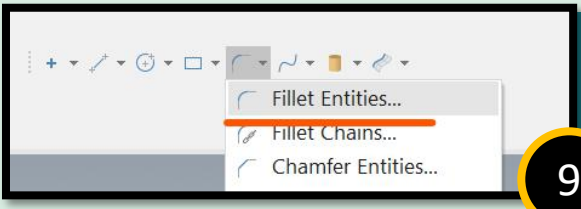
- In a ribbon bar, setting the **width** and **height**
- Click ok 

7



8

- Click right mouse and choose "**FIT**"

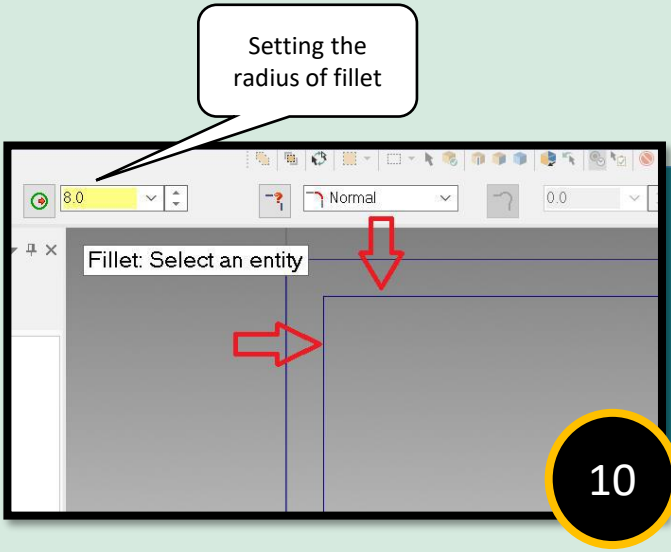


9

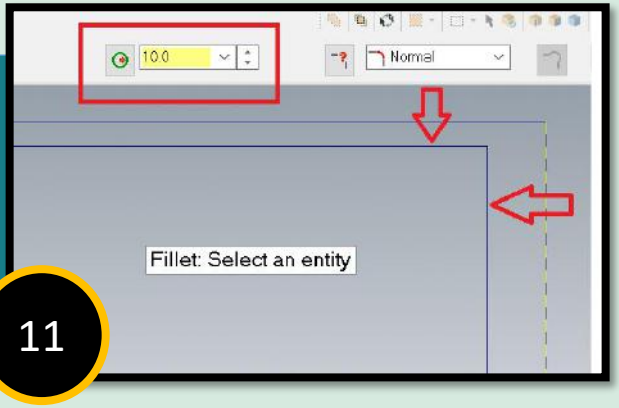
- Select SKETCHER feature
  - Fillet Entities

CADCAM
Mastercam

STEP 1: CREATE A GEOMETRY

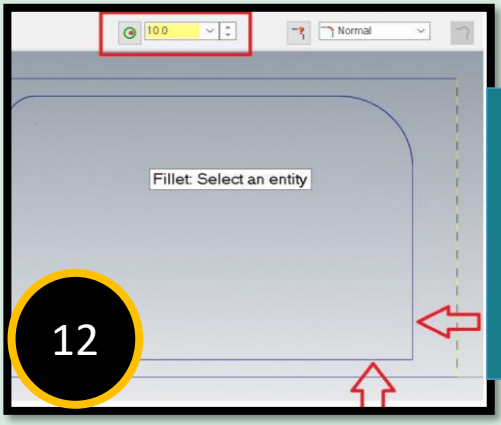


- Setting a radius of fillet
- Select both entities as shown in the figure.
- Click ok



- Setting a radius of fillet
- Select both entities as shown in the figure.
- Click ok

STEP 1: CREATE A GEOMETRY



- Setting a radius of fillet
- Select both entities as shown in the figure.
- Click ok

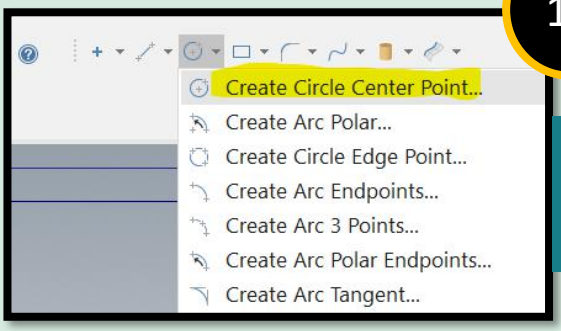
13

- Click right mouse and choose "FIT"

CADCAM Mastercam

STEP 1: CREATE A GEOMETRY

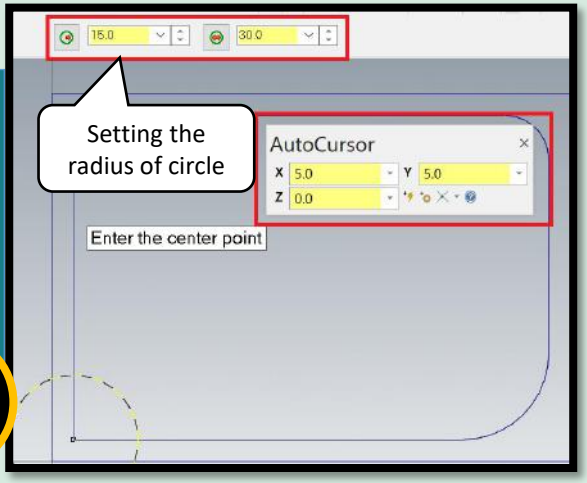
14



- Select SKETCHER feature
  - Circle

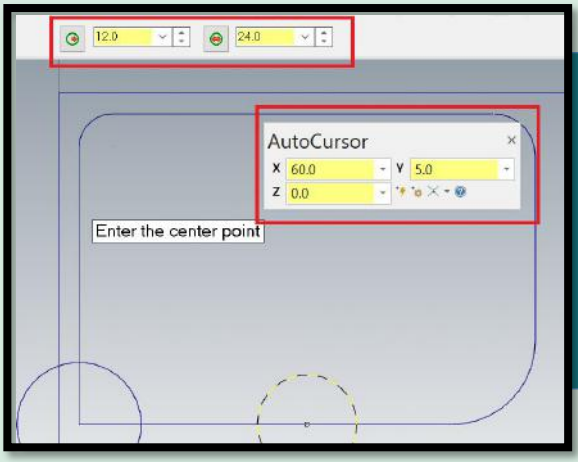
- Setting a coordinate of center point of circle
  - AutoCursor (5,5,0)
- Setting a radius of circle
- Click ok

15



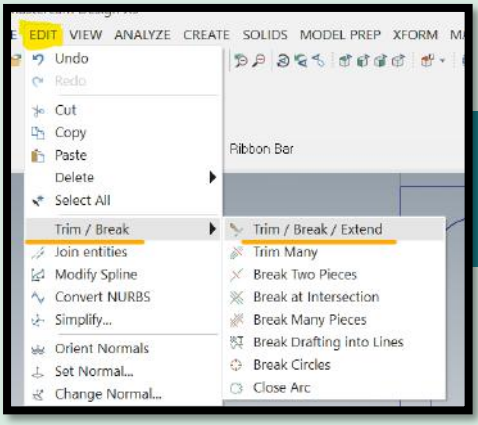
CADCAM Mastercam

STEP 1: CREATE A GEOMETRY



- Setting a coordinate of center point of circle
  - AutoCursor (60,5,0)
- Setting a radius of circle
- Click ok

16



17

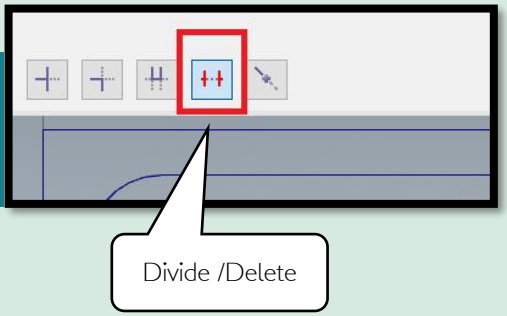
- Select **EDIT**
  - Trim/Break
  - Trim/Break/Extend

CADCAM Mastercam

STEP 1: CREATE A GEOMETRY

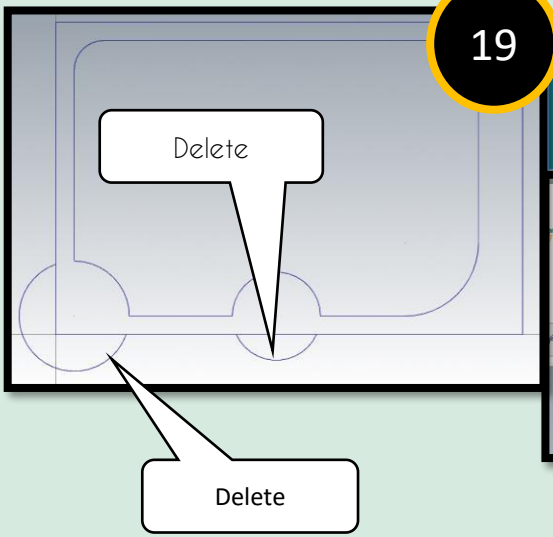
18

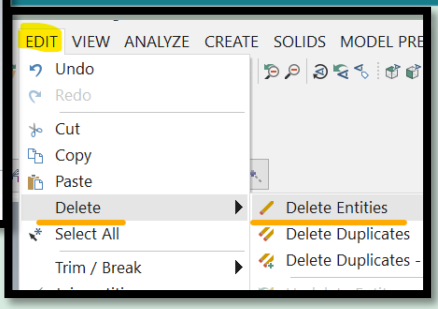
- Choose **Divide/Delete**
  - Select the entities you want to delete.



19

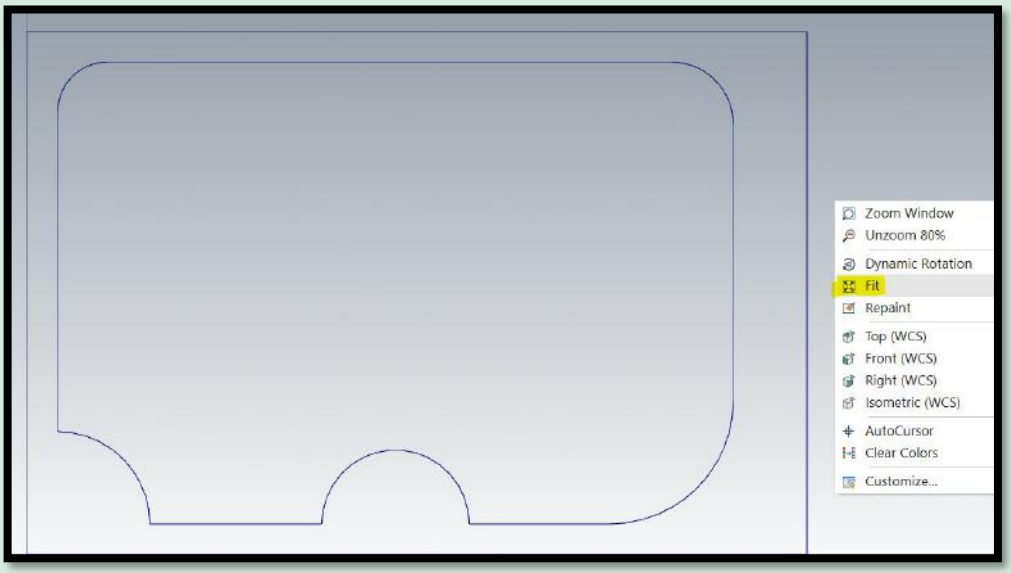
- After trim, select **Edit**
  - Delete
  - Delete entities





CADCAM Mastercam

STEP 1: CREATE A GEOMETRY



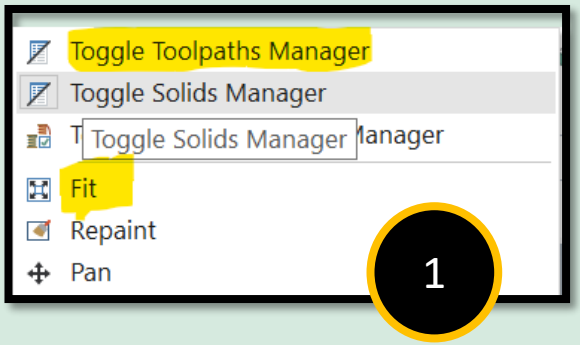
20

- Click right mouse and choose "FIT"
- Save the file : "Task_I"

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.

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

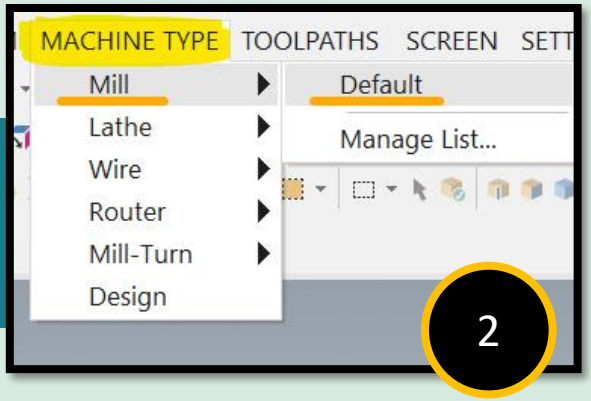


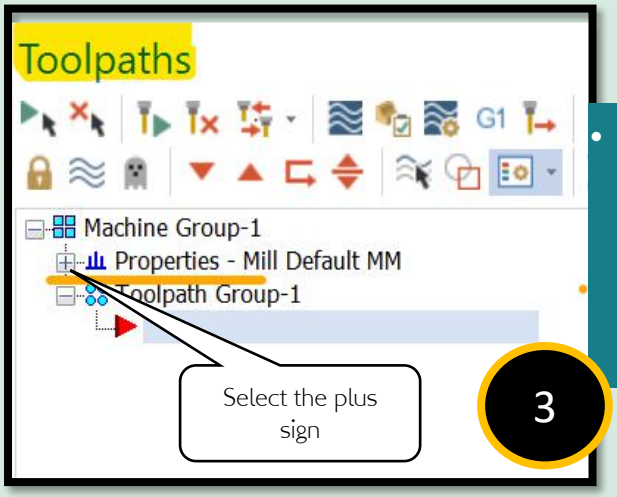
CADCAM Mastercam

STEP 2: SETUP THE MACHINE DEFINATION AND STOCK

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

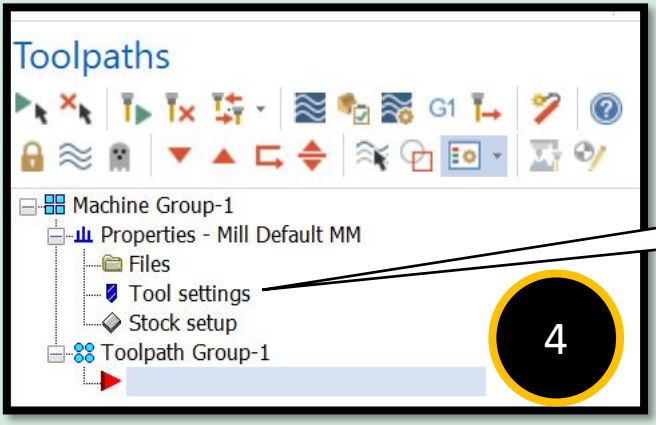
- From toolbar, click "Machine Type".
  - Mill
  - Default



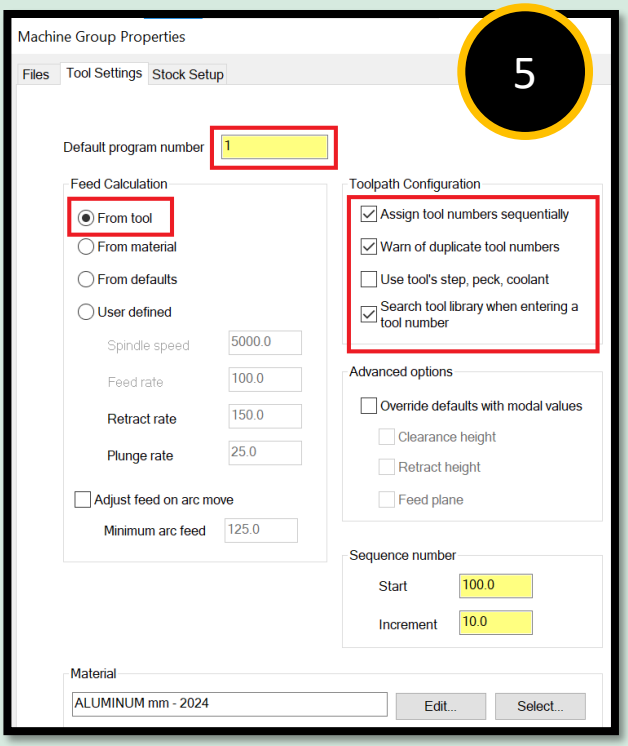


- Select the plus sign in front of **Properties** in the **Toolpaths Manager** to expend the **Toolpaths Group Properties** as shown in Figure

STEP 2: SETUP THE MACHINE DEFINATION AND STOCK



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



STEP 2: SETUP THE MACHINE DEFINATION AND STOCK

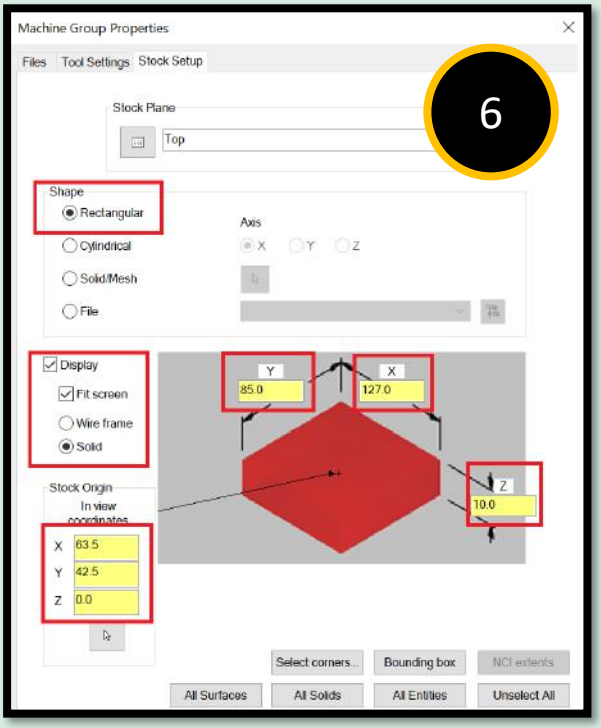
Program # is sequence number in series that used for any CNC machine consists the movements of cutting tool, to command the spindle speed, and feed rate and external M code in the command. The program is an order which the machine coded in a block programmed.

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).

.

- In **Machine Group Properties**, choose the **Stock Setup** tab.
- Set the shape of stock in **Rectangular. Please** enter the stock dimensions.

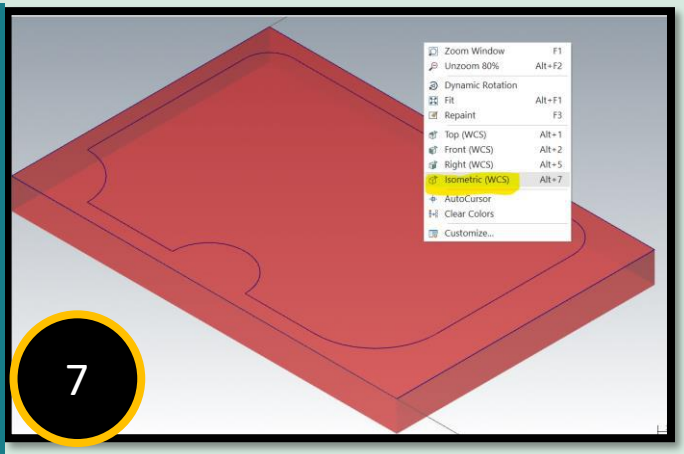


STEP 2: SETUP THE MACHINE DEFINATION AND STOCK

The **Stock Origin** values adjust the positioning of the stock, it is make sure that you have exactly amount of extra stock around the finished part.

Display options is an information for operation and machine operations item in the Toolpath Manager. You can set the view of stock as **wireframe, drive surfaces** or a **solid** and enable or disable the prompt.

- 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: MACHINING THE STOCK USING 2D CONTOUR TOOLPATH

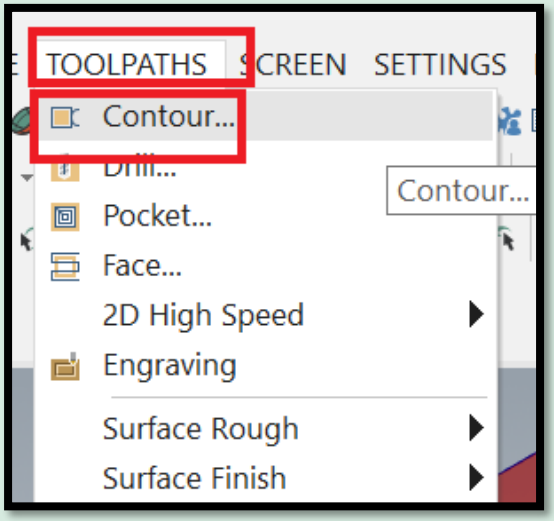
Contour toolpath is a 2D high Speed Dynamic milling toolpath. It is removes the material along direction of the tool axis. This path is defined by a loop or chain of the curves. Contouring is removed an enclosed area and follows a chain only.

1

From "**Toolbar**" Click "Toolpaths".

- Click "Contour Toolpath".

- Enter a name of NC programme.
- Select OK button.



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

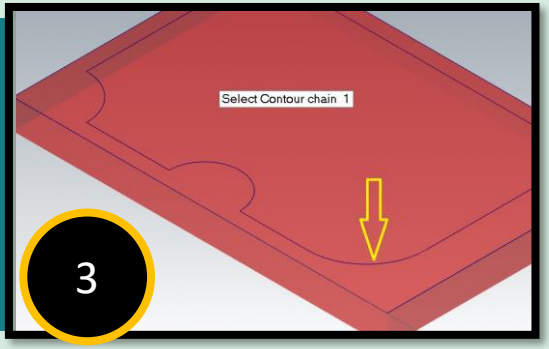
CADCAM Mastercam

STEP 3: MACHINING THE STOCK USING 2D CONTOUR TOOLPATH

- Select the Chain button like in figure. It is available to select only the outside contour.

2

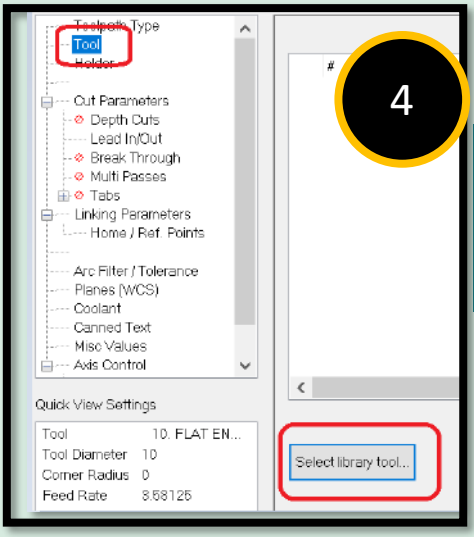
- Select the Contour chain.
- Click "OK".
- After that, the toolbar of "Contour (2D)" will come out.



3

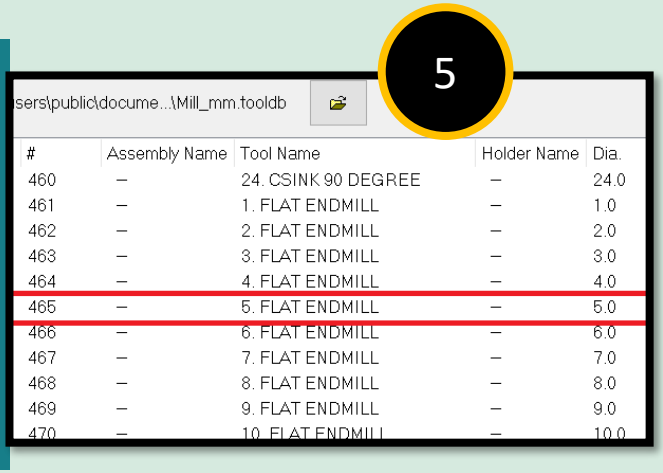
STEP 3: MACHINING THE STOCK USING 2D CONTOUR TOOLPATH

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



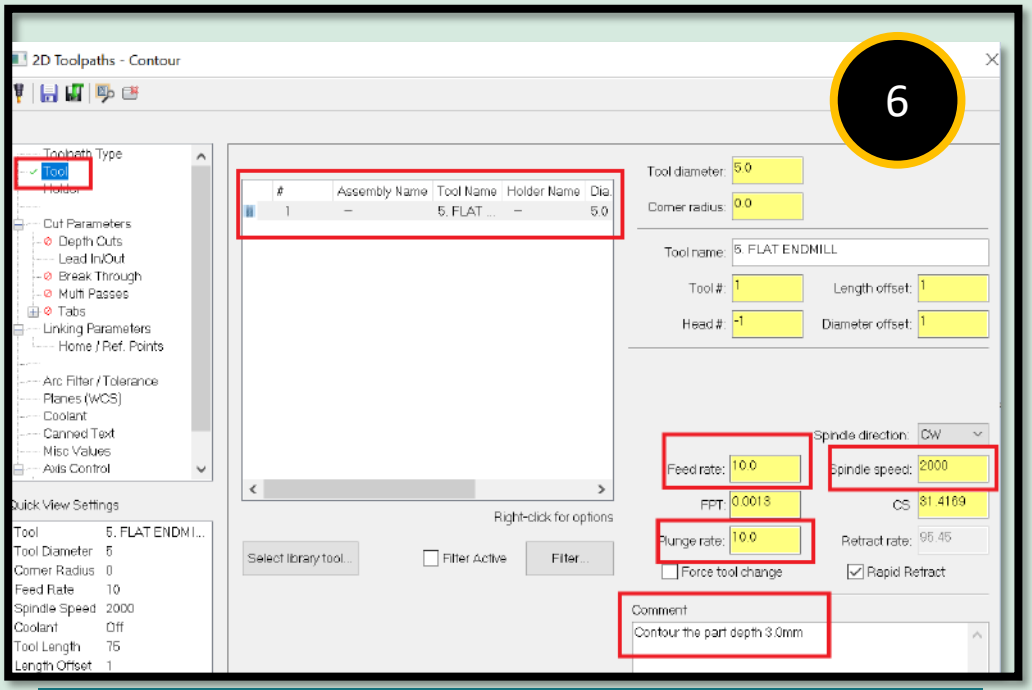
- From the Tree View List.
- select Tool
- Select the **Select Library Tool** button.

- Setting **Toolpath parameters:**
- Click **Select library tool** button to choose tool size.
- Choose tool size **Flat Endmill diameter 5.00 mm (#465)** as shown in Figure.



#	Assembly Name	Tool Name	Holder Name	Dia.
460	–	24. CSINK 90 DEGREE	–	24.0
461	–	1. FLAT ENDMILL	–	1.0
462	–	2. FLAT ENDMILL	–	2.0
463	–	3. FLAT ENDMILL	–	3.0
464	–	4. FLAT ENDMILL	–	4.0
465	–	5. FLAT ENDMILL	–	5.0
466	–	6. FLAT ENDMILL	–	6.0
467	–	7. FLAT ENDMILL	–	7.0
468	–	8. FLAT ENDMILL	–	8.0
469	–	9. FLAT ENDMILL	–	9.0
470	–	10. FLAT ENDMILL	–	10.0

STEP 3: MACHINING THE STOCK USING 2D CONTOUR TOOLPATH



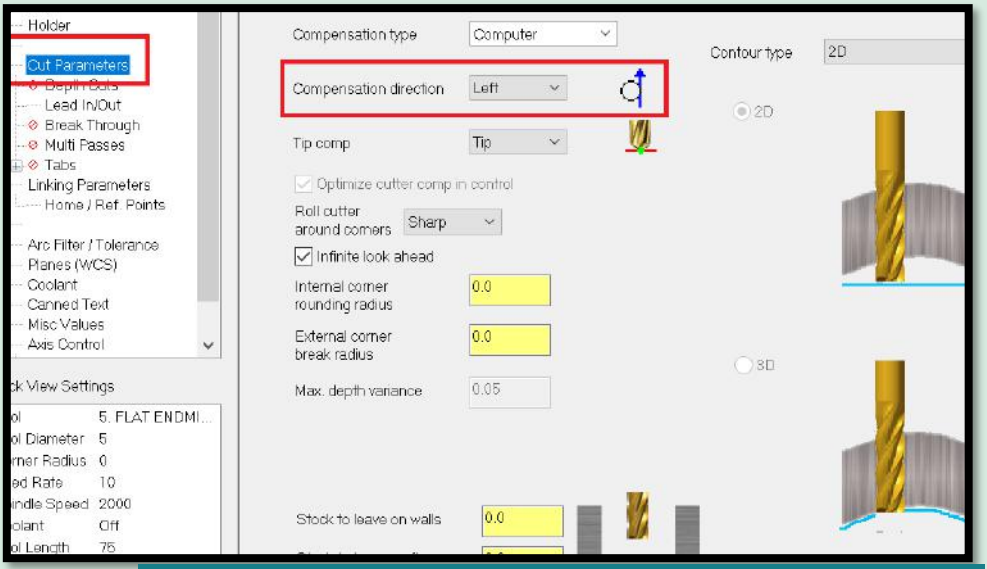
- Select the Tool Selection page
- Write the comment of the process.
- Choose all the necessary thing as shown in Figure.

Scan the QR codes. Or watch the video and listen carefully about cutting tool (**flat endmill**) in CNC milling.

STEP 3: MACHINING THE STOCK USING 2D CONTOUR TOOLPATH

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.



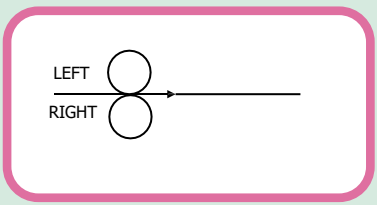
7

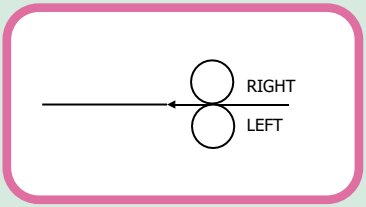
- From the Contour Parameter, select compensation director to set left or right depend the position of chaining direction.

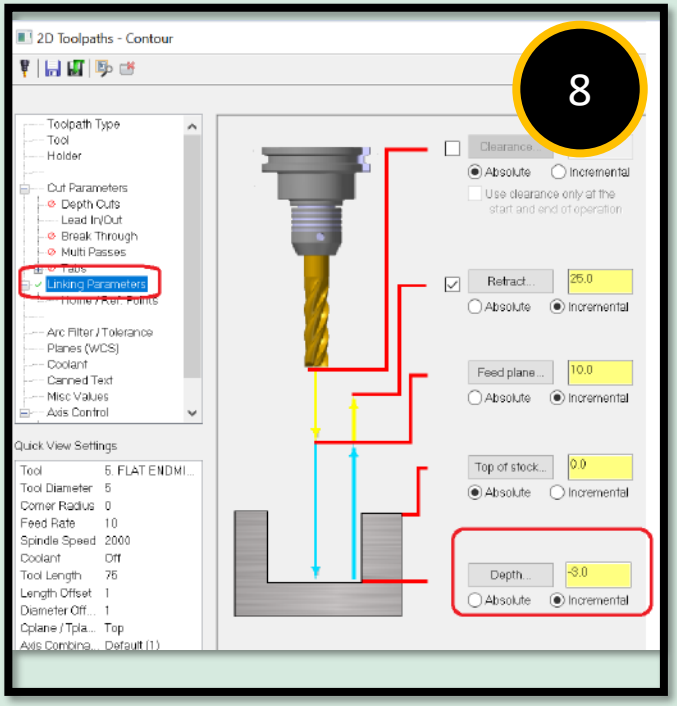
CADCAM Mastercam

STEP 3: MACHINING THE STOCK USING 2D CONTOUR TOOLPATH

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.



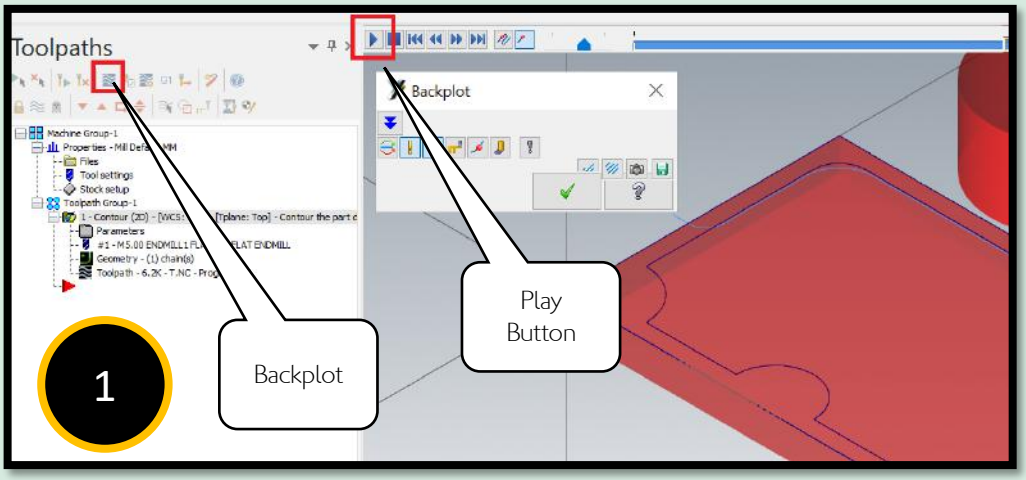




- Select the Linking Parameter
- Determine the parameters of depth.
- Click the OK for exit the command purposes.

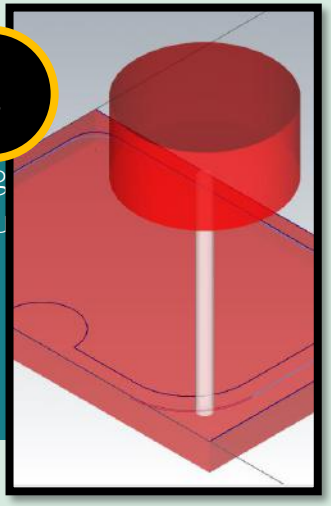
STEP 4: BACKPLOT THE TOOLPATHS

Backplotting shows the toolpath of cutting tool to cut the part. This display will lets 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.



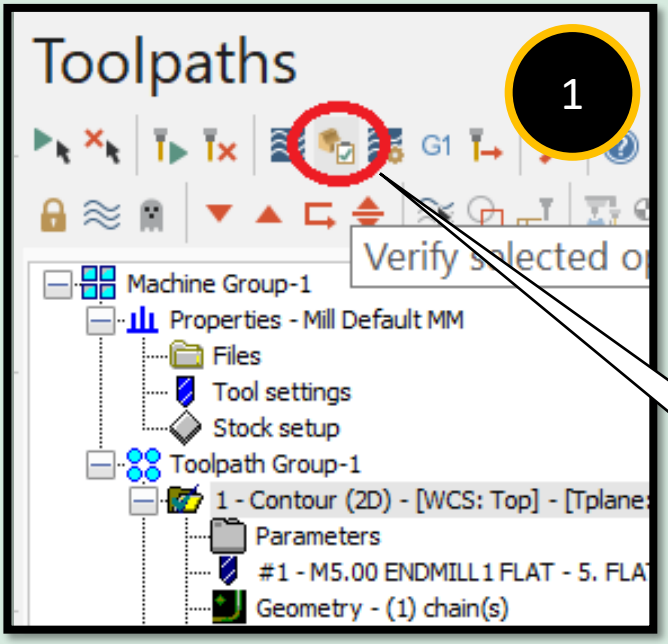
- On the operations button sele
  Backplot.
- Turn on all the buttons .(the cutting tool will appear pushed down). You can see the tool is running and follow the chain of toolpath.
- Adjust the speed of the backplot. Click a **Play** button.

2



STEP 5 : VERIFY THE TOOLPATHS

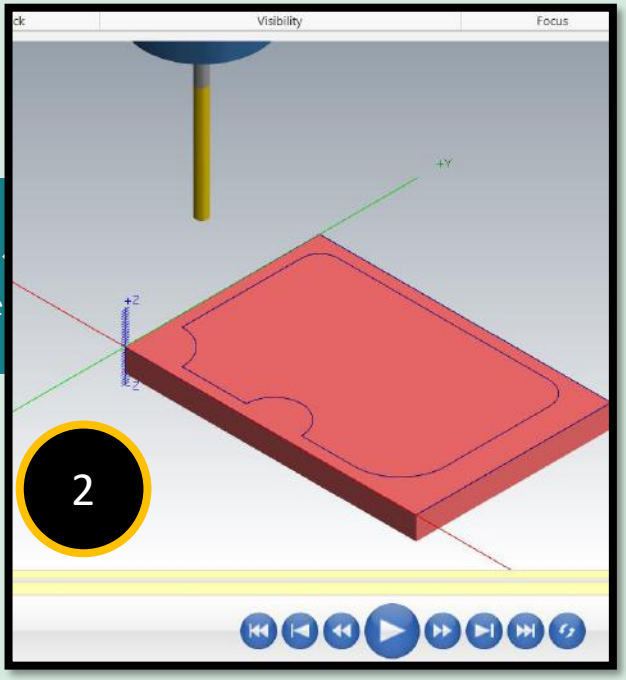
Verify allows you to simulate the machining of a part by using as solid model. It is simulation graphic of movement the cutting tool and the material removing pass by pass. It also can give a reminder of collisions between tool and stock if any happened.

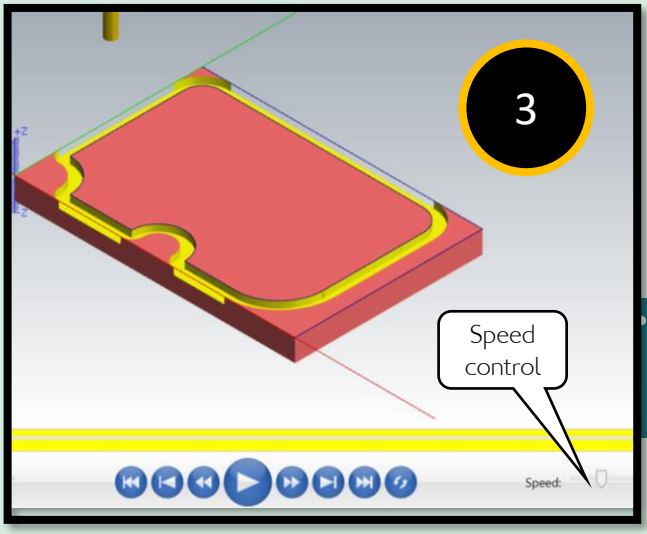


- From **Operation Manager**, choose **Verify selected operations** icon as shown in Figure.

STEP 5 : VERIFY THE TOOLPATHS

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

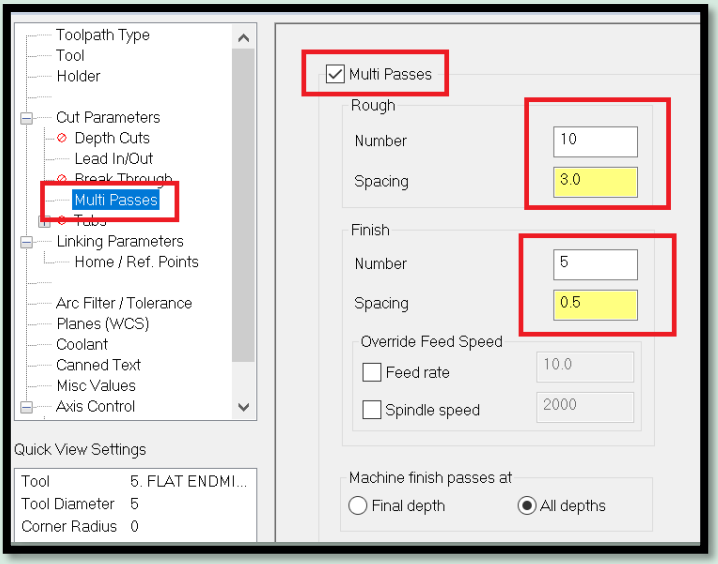




- The speed control can be adjusted as shown in the figure.

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.



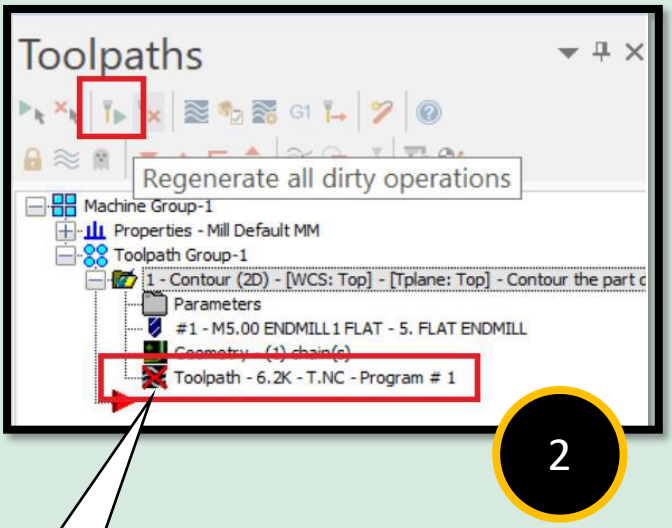
- Select **Multi Passes** from the **Tree view list**
- Select the necessary changes.
- Enter a value for the Number of passes.
- Define the **Spacing** distance.
- Set the machining depth for finish passes.
- Click OK to exit the **Multi Passes** parameters.

1

CADCAM Mastercam

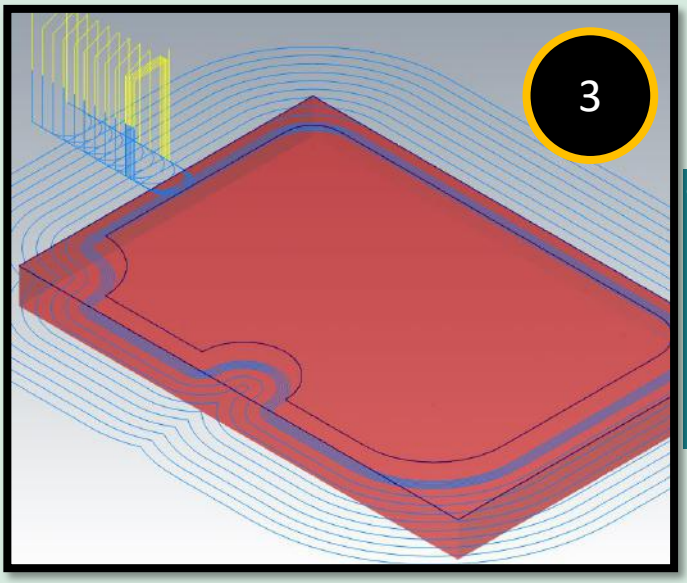
STEP 6: ADDING MULTI PASSES OF TOOLPATH

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

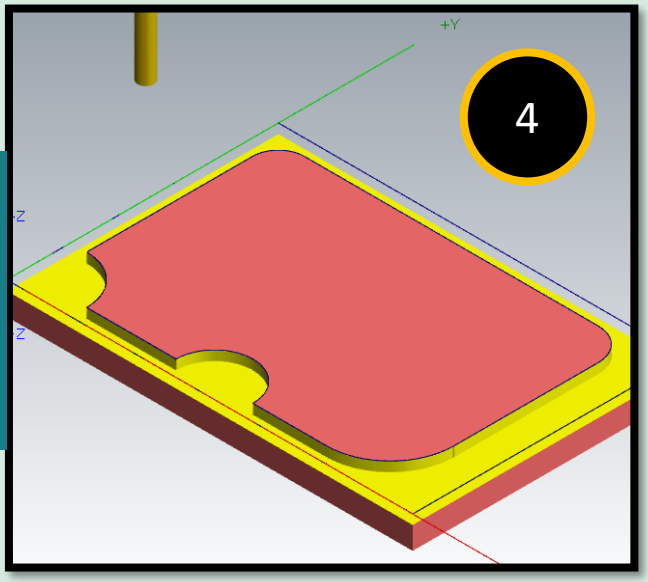


The cross at the toolpath showed the dirty operations have to be generated before simulation

STEP 6: ADDING MULTI PASSES OF TOOLPATH



- Once the operation had been regenerated and remove and select to review your toolpath in these procedures

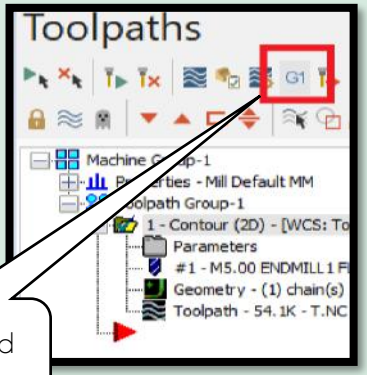


- To Backplot and Verify the toolpaths, please refer back the topic to review these procedures.

STEP 7: POST THE FILE

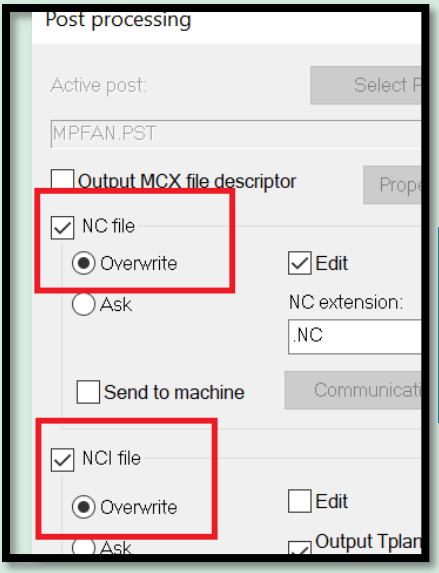
Post processing, or **posting a program**, it is refers to convert the process of toolpaths in your Mastercam part files to a format that can be understood by your CNC machine tool's control. For example, G-codes and M-codes.

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



1 Post selected operations

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



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

2

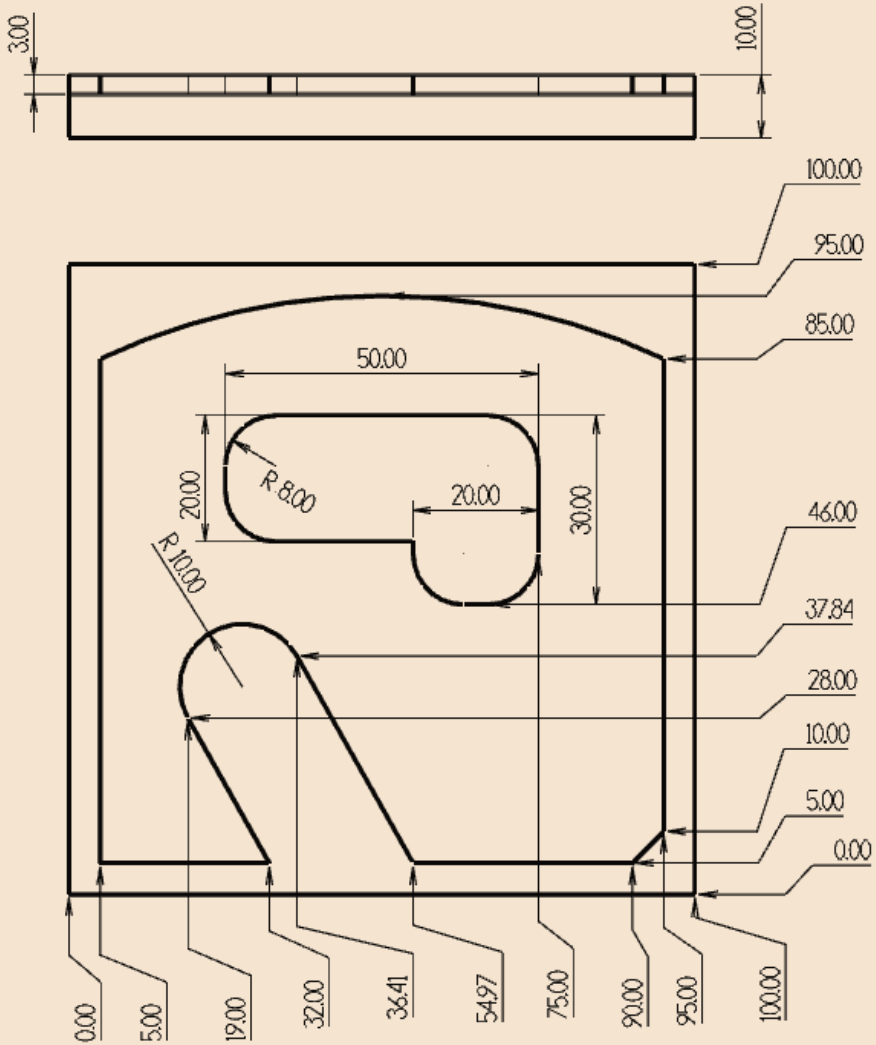
STEP 7: POST THE FILE

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.

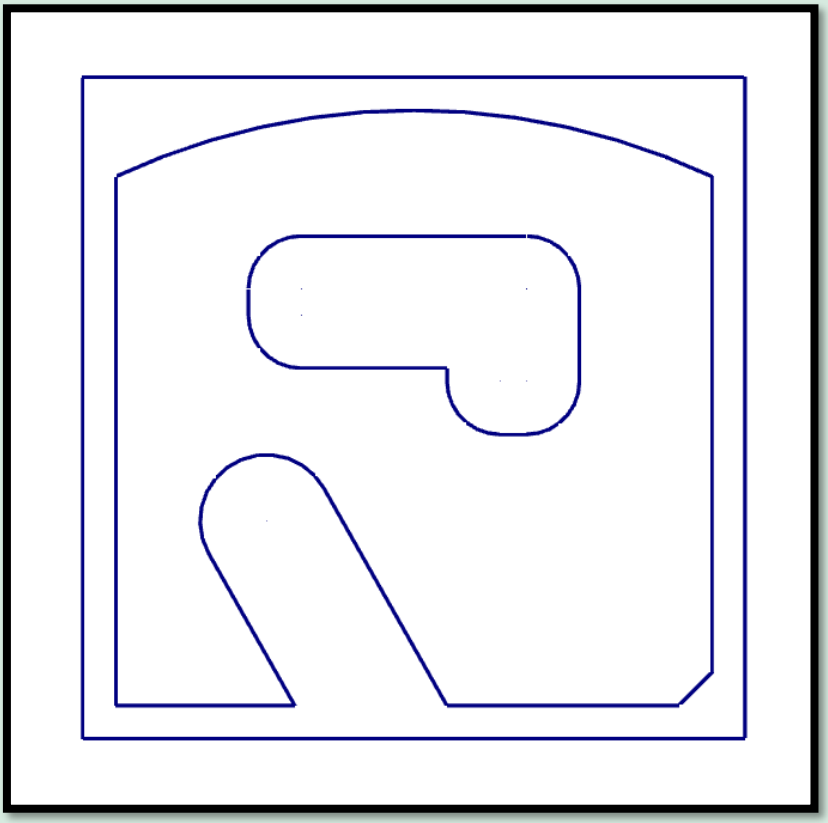
```
O0001(T)
(DATE=DD-MM-YY - 13-08-21 TIME=HH:MM -
(MCX FILE - C:\USERS\LENOVO\DOCUMENTS\
(NC FILE - C:\USERS\LENOVO\DOCUMENTS\M
(MATERIAL - ALUMINUM MM - 2024)
( T1 | 5. FLAT ENDMILL | H1 )
N100 G21
N110 G0 G17 G40 G49 G80 G90
( CONTOUR THE PART DEPTH 3.0MM )
N120 T1 M6
N130 G0 G90 G54 X-34. Y41. A0.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

PRACTICAL TASK 2



TITLE	PRACTICAL TASK 2	DEPT	MECHANICAL ENGINEERING
		COURSE	DJF41042
POLITEKNIK PORT DICKSON		LAB	CAD CAM
		UNITS	METRIC (MM)

STEP 1: CREATE A GEOMETRY

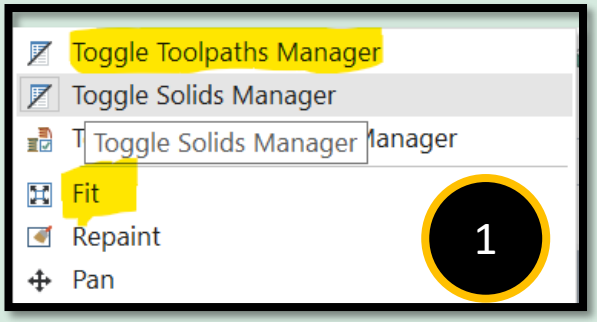


- Scan the QR code.
- Watch the tutorial video how to drafting the task in MasterCam X9 .
- Draw a picture that has been given.
- Save the file : "Task_2"

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.

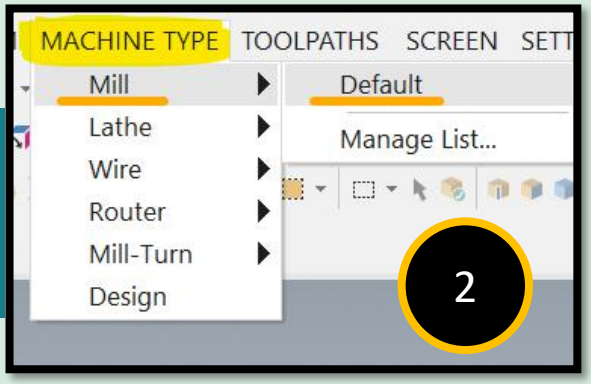
- Type **ALT+O** for display of **Operating Manager.** You can use **View Features** too.
- Select the **Fit** icon in **View Features** to zoom or fit the drawing to the screen



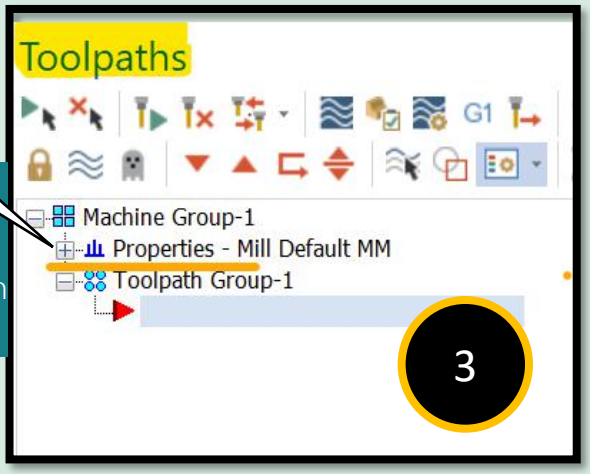
STEP 2: SETUP THE MACHINE DEFINATION AND STOCK

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

- From toolbar, click "**Machine Type**".
  - Mill
  - Default

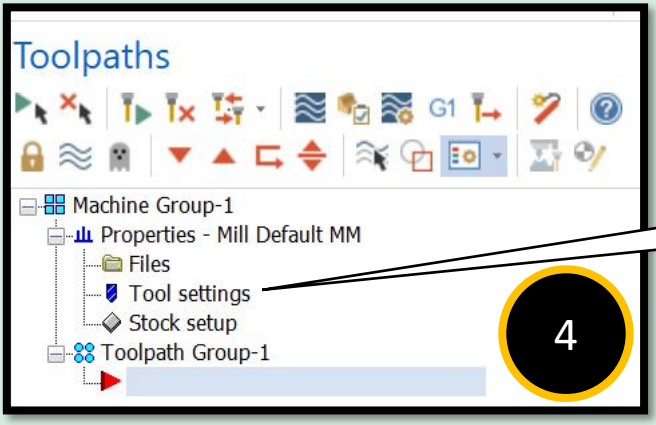


- Select plus sign-it **is** to expend the **Toolpaths Group Properties** as shown in Figure).

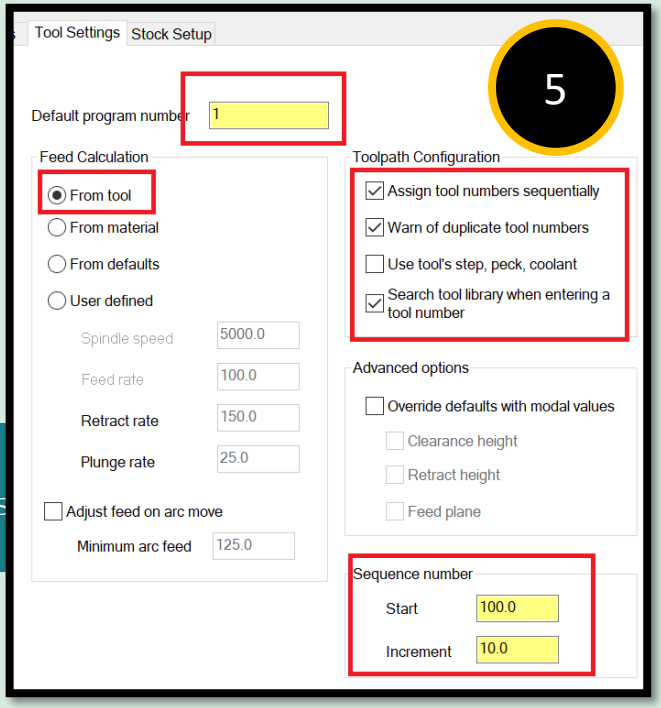


CADCAM Mastercam

STEP 2: SETUP THE MACHINE DEFINATION AND STOCK



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



STEP 2: SETUP THE MACHINE DEFINATION AND STOCK

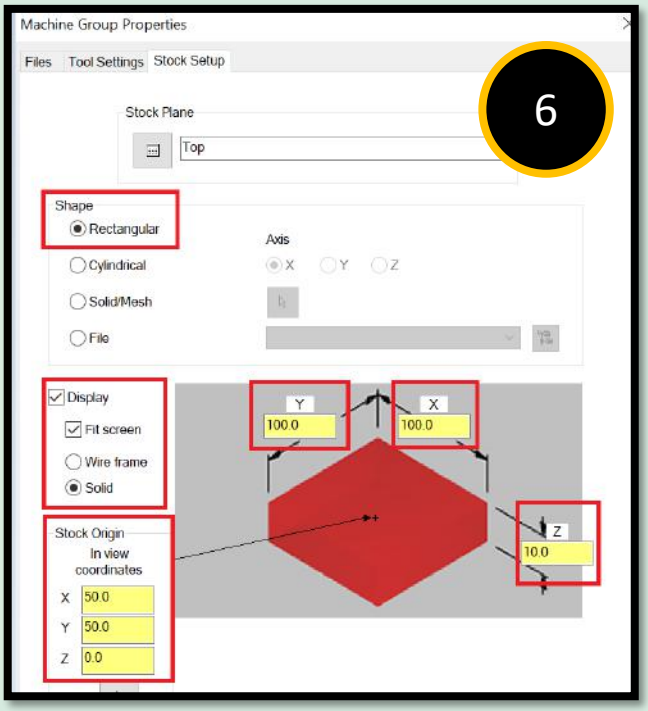
Program # is sequence number in series that used for any CNC machine consists the movements of cutting tool, to command the spindle speed, and feed rate and external M code in the command. The program is an order which the machine coded in a block programmed.

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 (when you enter the two tools with the same number, you will get a warning).

.

- In **Machine Group Properties**, choose the **Stock Setup** tab.
- Set the shape of stock in **Rectangular. Please** enter the stock dimensions.

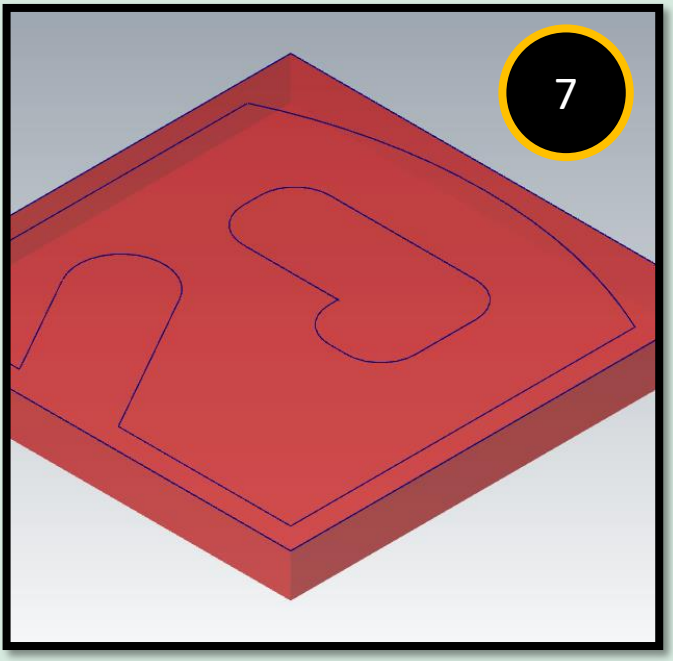


STEP 2: SETUP THE MACHINE DEFINATION AND STOCK

The **Stock Origin** values adjust the positioning of the stock, it is make sure that you have exactly amount of extra stock around the finished part.

Display options allow you to set the view of stock as **Wireframe** or a **Solid** and to fit the stock to the screen (Fit Screen)

- 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: MACHINING THE STOCK USING 2D CONTOUR TOOLPATH

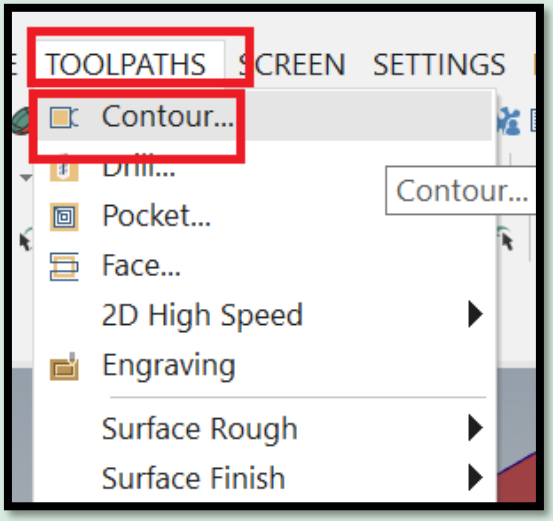
Contour toolpath is a 2D high Speed Dynamic milling toolpath. It is removes the material along direction of the tool axis. This path is defined by a loop or chain of the curves. Contouring is removed an enclosed area and follows a chain only.

1

From "**Toolbar**" Click "Toolpaths".

- Click "Contour Toolpath".

- Enter a name of NC programme.
- Select OK button.

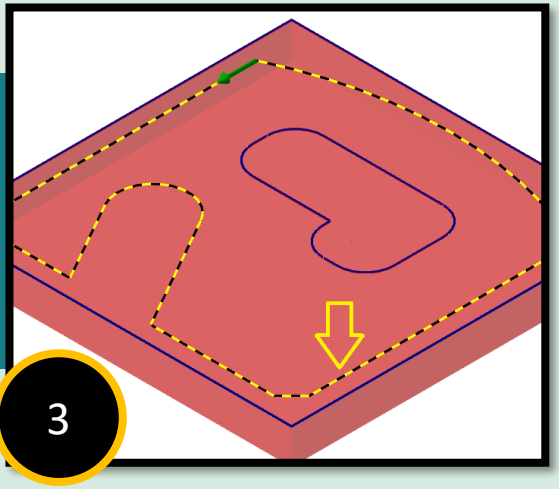


Note: A chain of entities which is are made up of one or more paths or entities. The length has to be same or less than the chaining tolerance 0.002mm (in between the endpoints of two consecutive entities or path). The entities must be together in direction and order

STEP 3: MACHINING THE STOCK USING 2D CONTOUR TOOLPATH

- Select the Chain button like in figure. It is available to choose only for the outside contour.

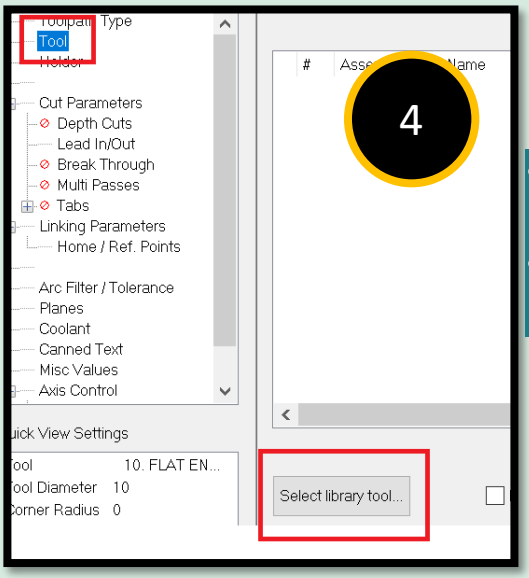
- Select the loop as shown in the Figure 1.9
- Click "**OK**" button to exit **Chaining.** After that, the toolbar of "Contour (2D)" will come out.



CADCAM Mastercam

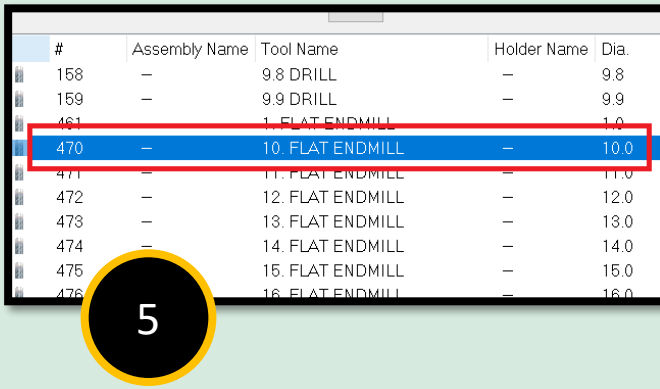
STEP 3: MACHINING THE STOCK USING 2D CONTOUR TOOLPATH

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

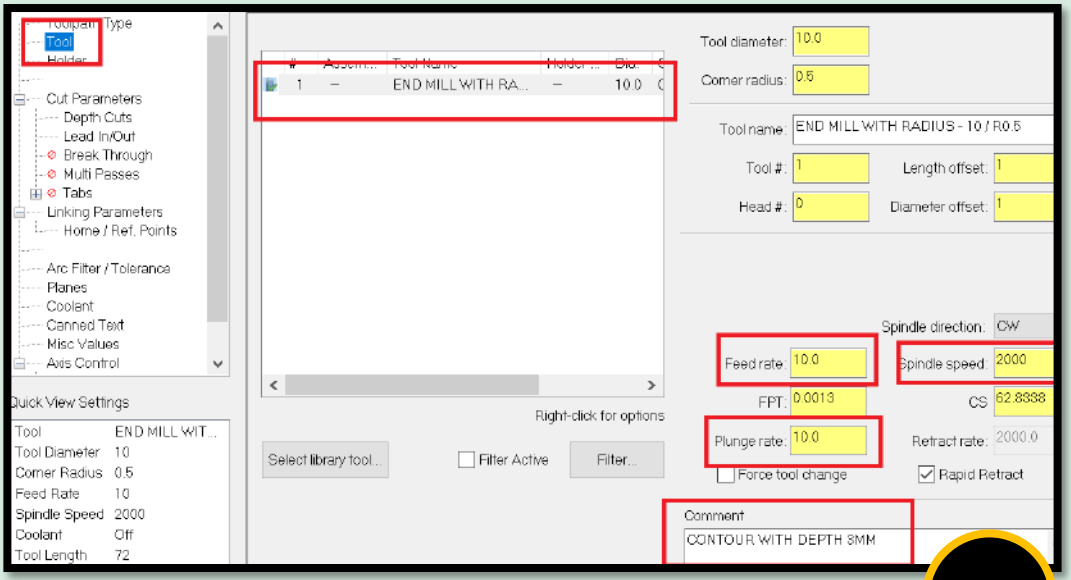


- From the Tree View List. select Tool
- Choose the **Select Library Tool** button.

- Setting **Toolpath parameters:**
- Click **Select library tool** button to choose tool size.
- Choose tool size **Flat Endmill diameter 10.00 mm (#470)** as shown in Figure.



STEP 3: MACHINING THE STOCK USING 2D CONTOUR TOOLPATH



6

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

The **Feed rate, Plunge rate, Retract rate** and **Spindle speed** calculation are based for material Aluminium and HSS tooling. You can change the values depend on your desired tools and material in the program.

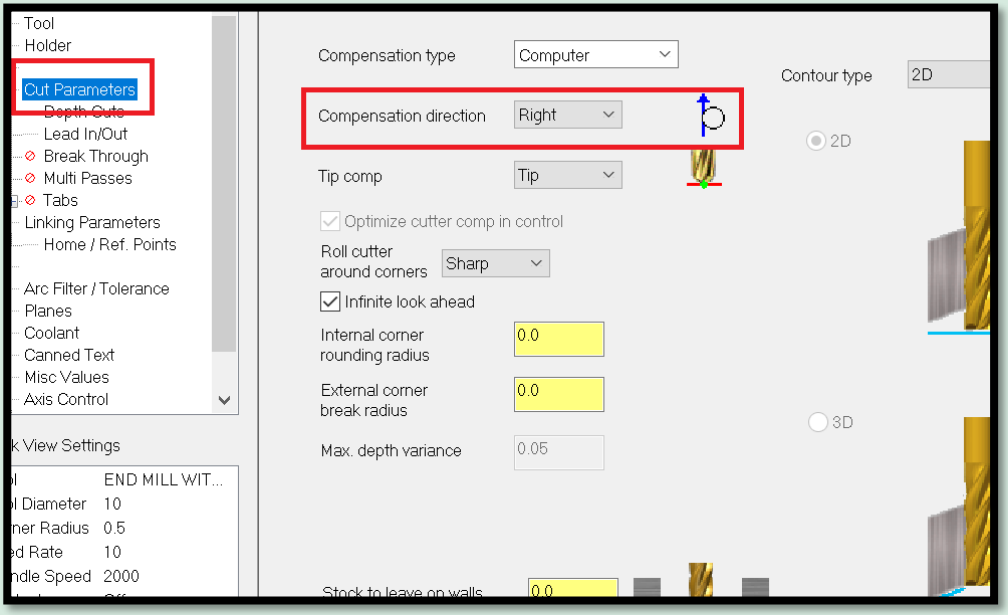
In the **Comment** field is to help you to identify the toolpath by enter a commenting **The Toolpaths/ Operation Manager.** Please refer the figure above.

CADCAM Mastercam

STEP 3: MACHINING THE STOCK USING 2D CONTOUR TOOLPATH

7

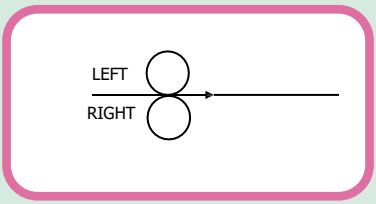
- From the Contour Parameter, select compensation director to set left or right depend the position of chaining direction.

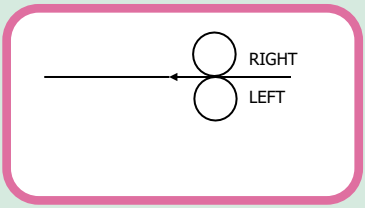


CADCAM
Mastercam

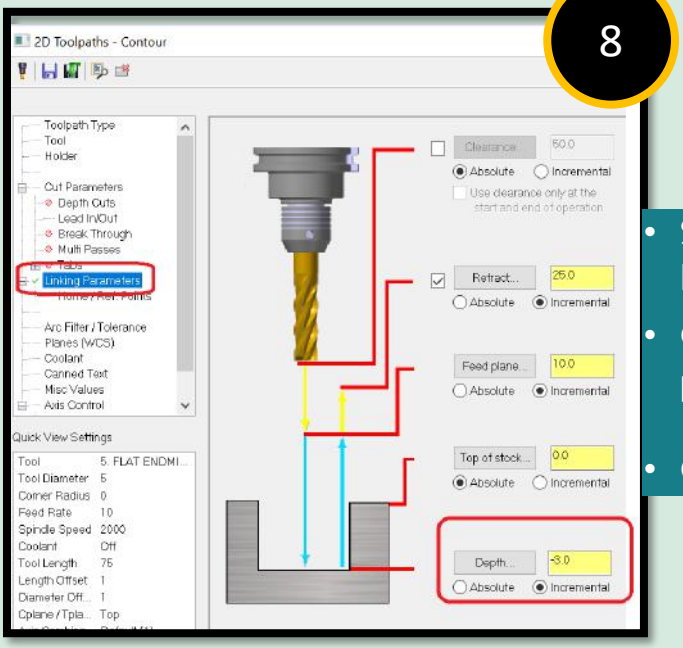
STEP 3: MACHINING THE STOCK USING 2D CONTOUR TOOLPATH

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.





8

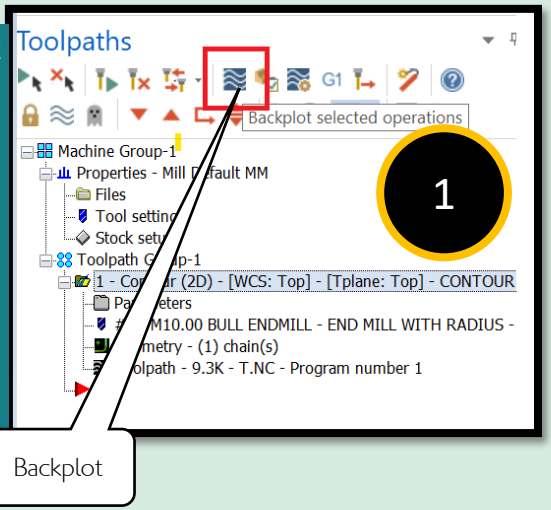


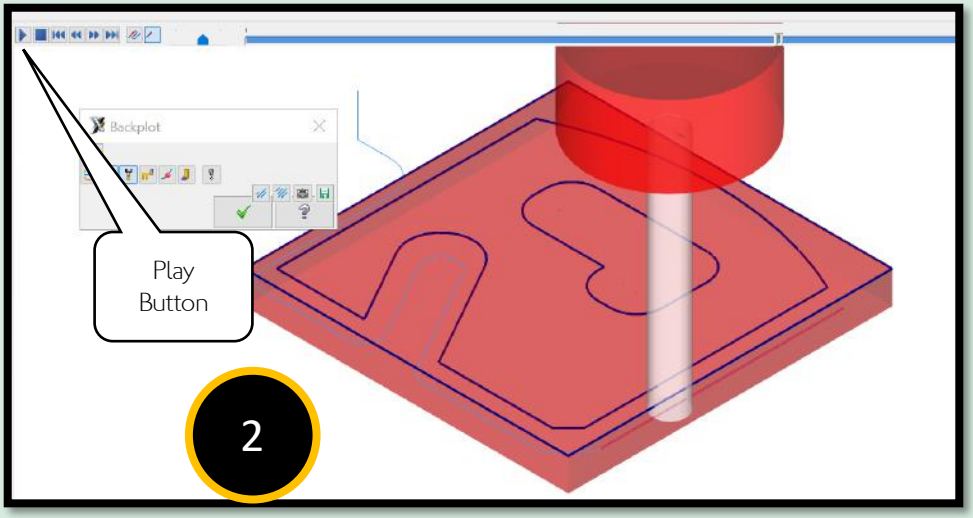
- Select the Linking Parameter
- Change the parameters of depth.
- Click the OK button.

STEP 4: BACKPLOT THE TOOLPATHS

Backplotting shows the toolpath of cutting tool to cut the part. This display will lets you to spot an errors. 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.

- 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.

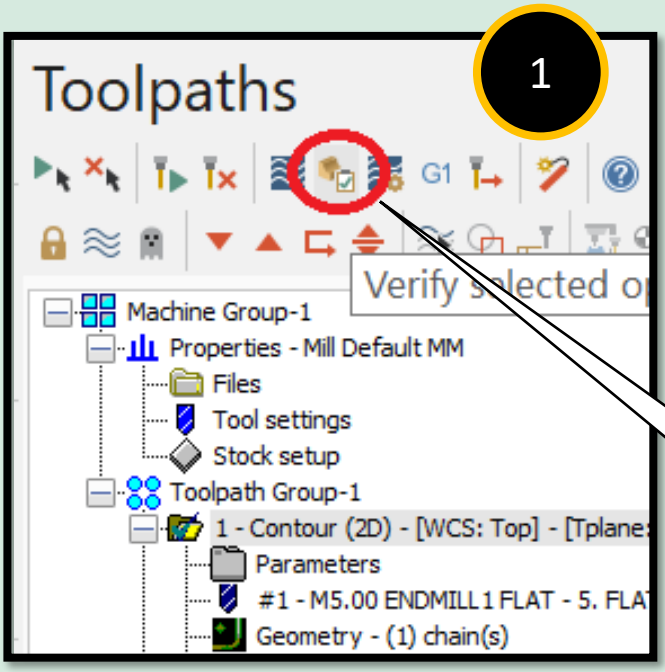




CADCAM Mastercam

STEP 5: VERIFY THE TOOLPATHS

Verify allows you to simulate the machining of a part by using as solid model. The solid model created by verification represents the surface roughing and finishing. It also shows collisions if any exist.

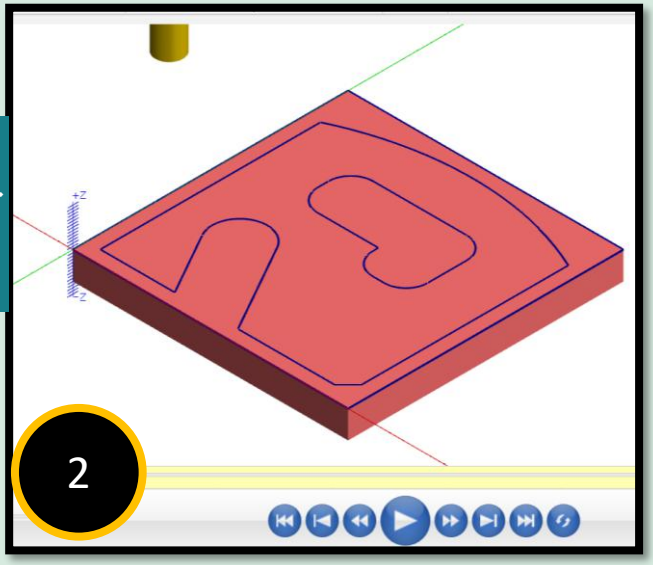


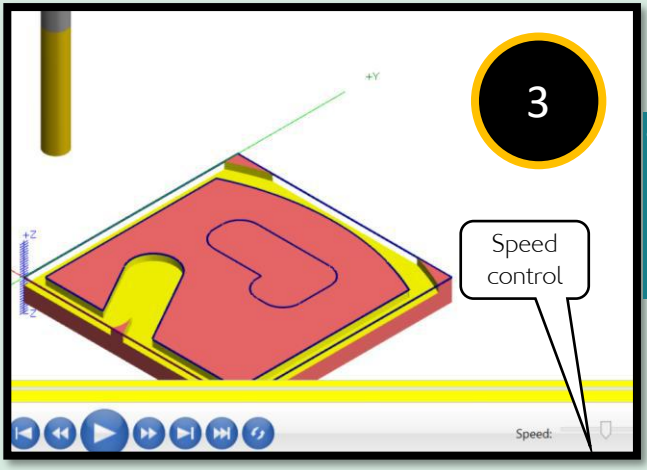
- From **Operation Manager**, choose **Verify selected operations** icon as shown in Figure.

CADCAM Mastercam

STEP 5: VERIFY THE TOOLPATHS

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



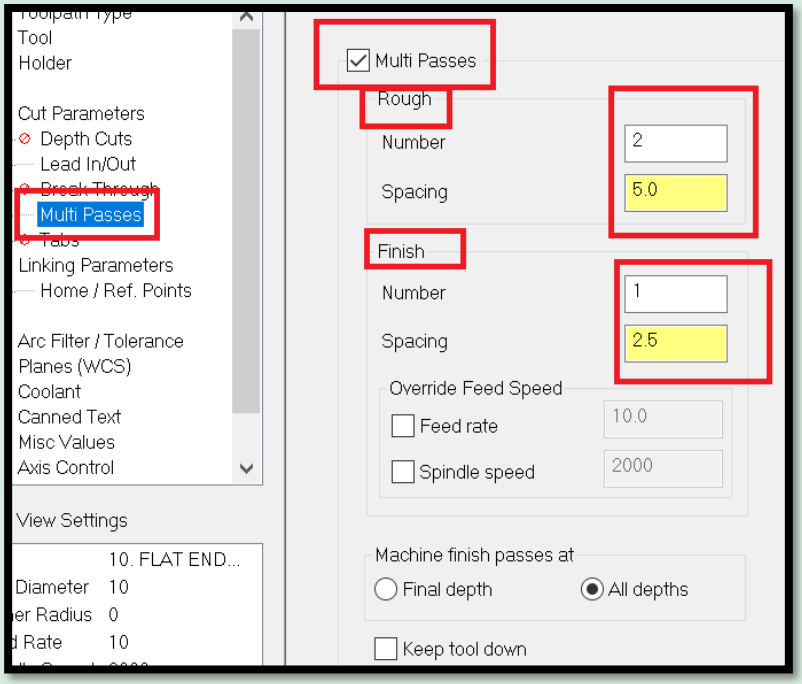


- Setting the speed of the **Verify** by moving the slider bar in the speed control bar.

CADCAM Mastercam

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.

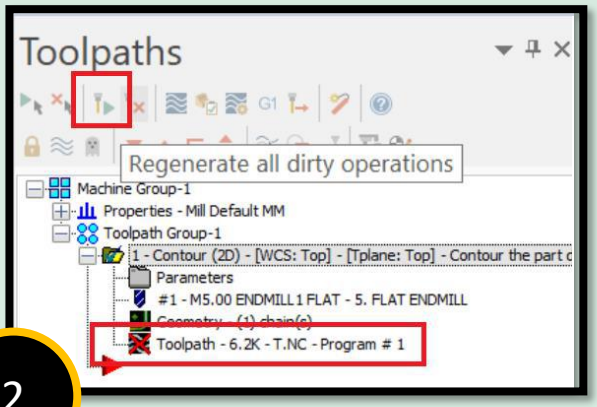


- Select **Multi Passes** from the **Tree view list**
- Make the changes number as shown in figure above.
- Enter a value for the Number of passes.
- Define the **Spacing** distance.
- Set the machining depth for finish passes.
- Click OK to exit the **Multi Passes** parameters.

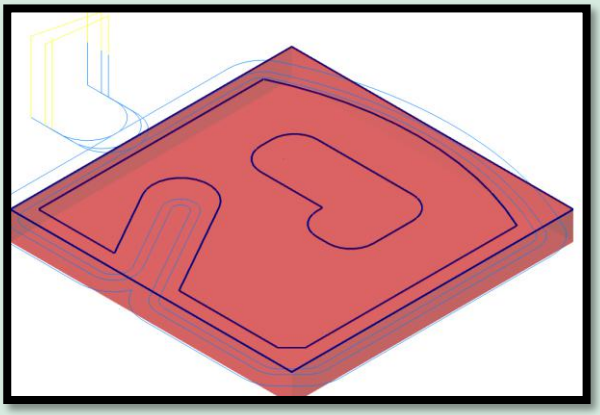
1

STEP 6: ADDING MULTI PASSES OF TOOLPATH

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



2



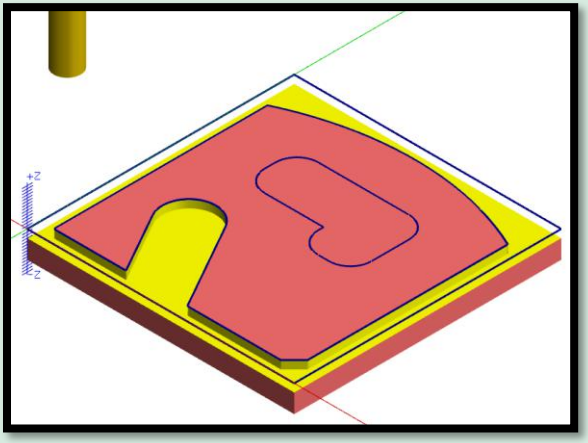
3

- Once the operation had been regenerated and remove and select to review your toolpath in these procedures

STEP 6: ADDING MULTI PASSES OF TOOLPATH

4

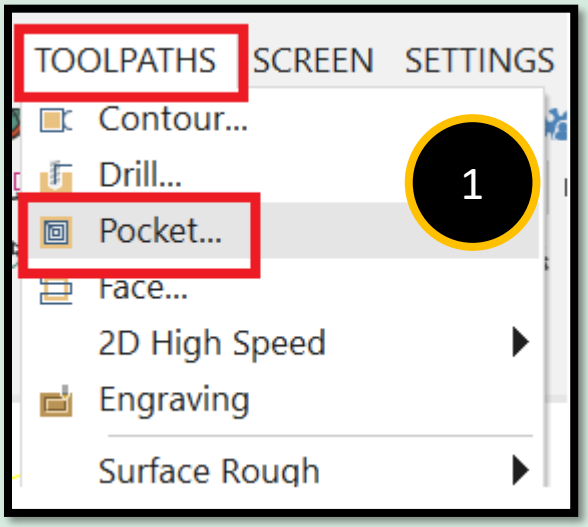
- To Backplot and Verify the toolpaths, please refer back to the topic to review these procedures.



STEP 7: MACHINING THE STOCK USING 2D POCKETING TOOLPATH

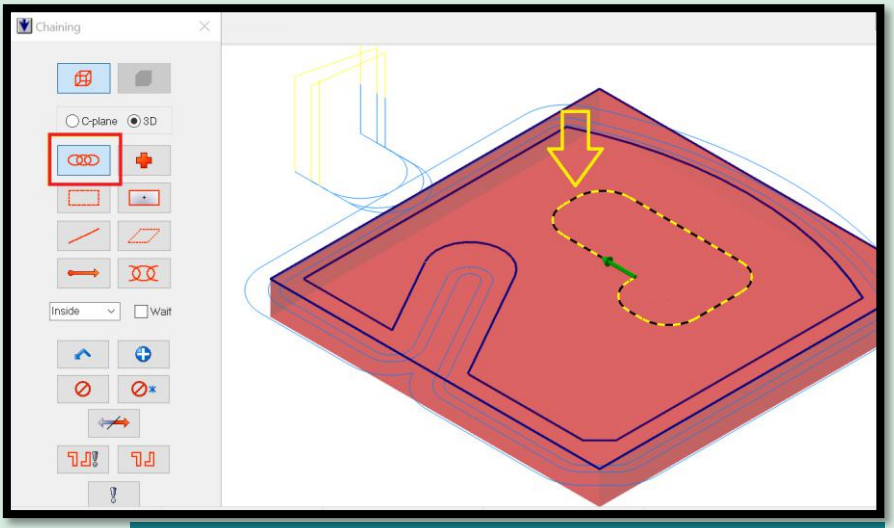
Note: Pocket toolpaths is removed the part of material from an enclosed boundary.

- From "**Toolbar**" Click "Toolpaths".
  - Click "Pocket Toolpath".
- Enter a name of NC programme.
- Select OK button.



STEP 7: MACHINING THE STOCK USING 2D POCKETING TOOLPATH

Note: Pocket toolpaths remove the material from an enclosed boundary.



2

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

STEP 7: MACHINING THE STOCK USING 2D POCKETING TOOLPATH

Note: Pocket toolpaths remove the material from an enclosed boundary.

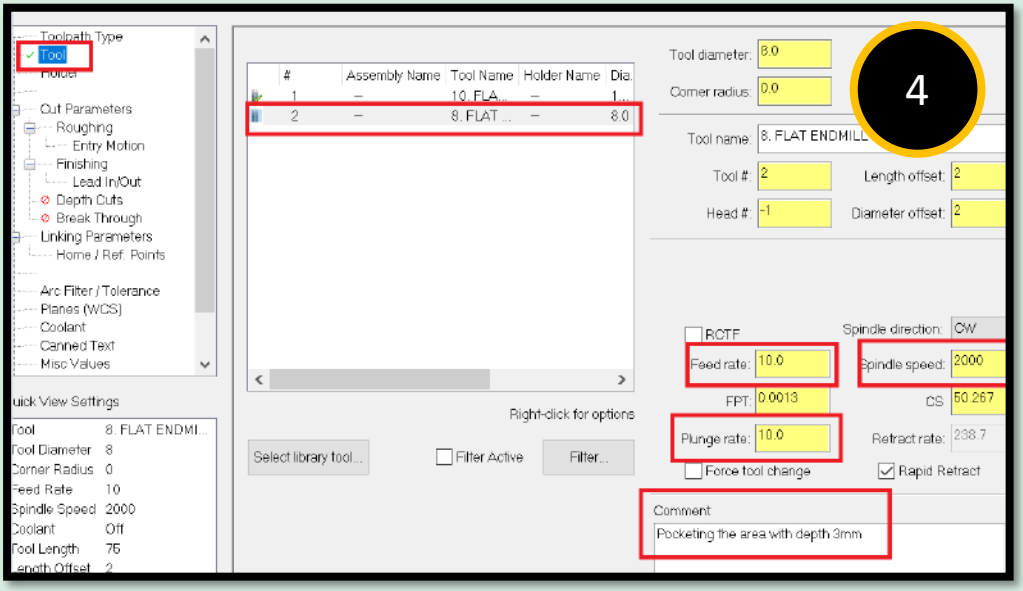
C:\users\public\docume...\Mill_mm.tooldb

#	Assembly Name	Tool Name	Holder Name	Dia.	Cor. rad.	Length	Type
464	–	4. FLAT ENDMILL	–	4.0	0.0	50.0	Endmill1 Flat
465	–	5. FLAT ENDMILL	–	5.0	0.0	50.0	Endmill1 Flat
466	–	6. FLAT ENDMILL	–	6.0	0.0	50.0	Endmill1 Flat
467	–	7. FLAT ENDMILL	–	7.0	0.0	50.0	Endmill1 Flat
468	–	8. FLAT ENDMILL	–	8.0	0.0	50.0	Endmill1 Flat
469	–	9. FLAT ENDMILL	–	9.0	0.0	50.0	Endmill1 Flat
486	–	1. BALL ENDMILL	–	1.0	0.5	50.0	Endmill2 Sph...
495	–	10. BALL ENDMILL	–	10.0	5.0	50.0	Endmill2 Sph...
496	–	11. BALL ENDMILL	–	11.0	5.5	50.0	Endmill2 Sph...
497	–	12. BALL ENDMILL	–	12.0	6.0	50.0	Endmill2 Sph...
498	–	13. BALL ENDMILL	–	13.0	6.5	50.0	Endmill2 Sph...
499	–	14. BALL ENDMILL	–	14.0	7.0	50.0	Endmill2 Sph...
500	–	15. BALL ENDMILL	–	15.0	7.5	50.0	Endmill2 Sph...
501	–	16. BALL ENDMILL	–	16.0	8.0	50.0	Endmill2 Sph...
502	–	17. BALL ENDMILL	–	17.0	8.5	50.0	Endmill2 Sph...
503	–	18. BALL ENDMILL	–	18.0	9.0	50.0	Endmill2 Sph...

3

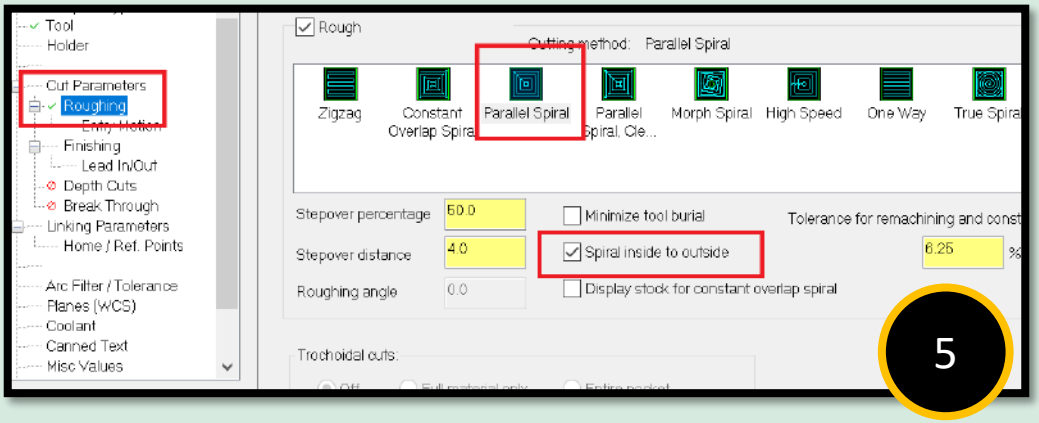
- Setting **Toolpath parameters:**
- Click **Select library tool** button to choose tool size
- Disable **Filter Active t**o able to see all the tools from the library.
- Select tool size **Flat Endmill diameter 8.00 mm (#468)** as shown Figure.

STEP 7: MACHINING THE STOCK USING 2D POCKETING TOOLPATH



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

STEP 7: MACHINING THE STOCK USING 2D POCKETING TOOLPATH



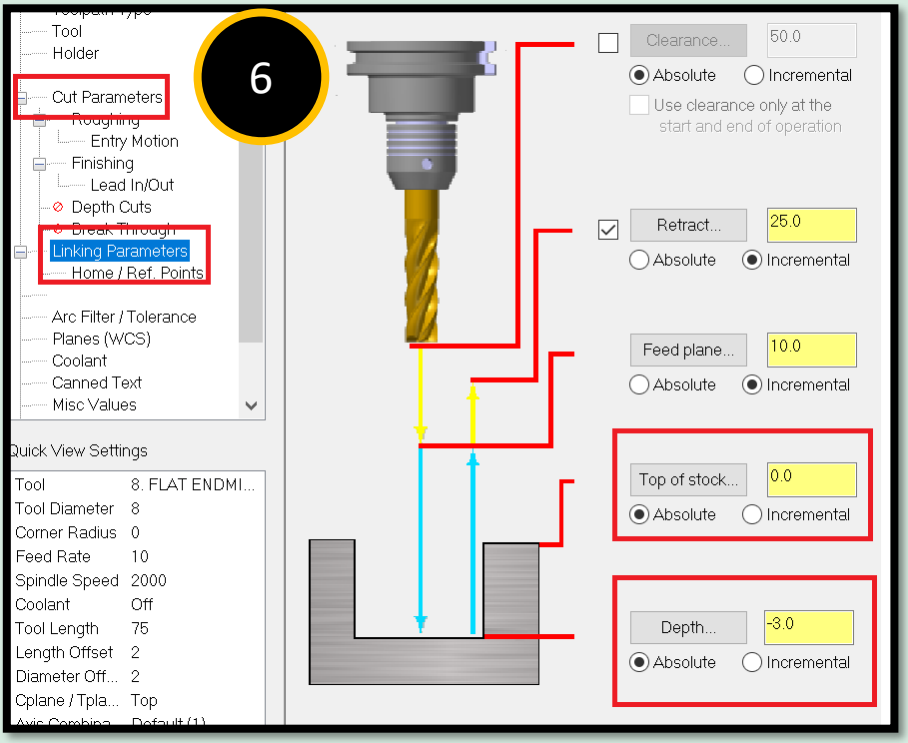
- From the **Tree view list**, select **Roughing Parameters.** Choose the option **Parallel Spiral** and change the necessary settings as shown in Figure.

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.

CADCAM Mastercam

STEP 7: MACHINING THE STOCK USING 2D POCKETING TOOLPATH

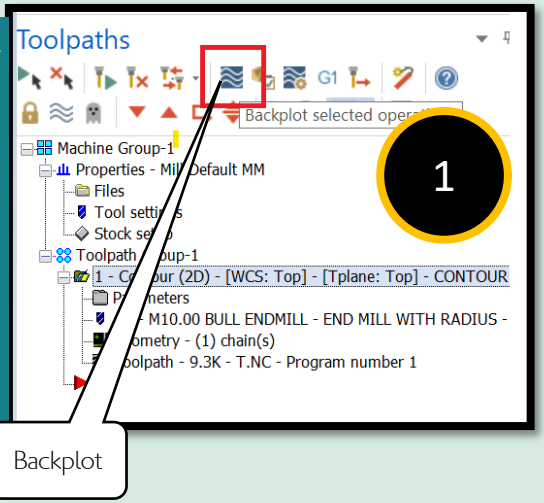


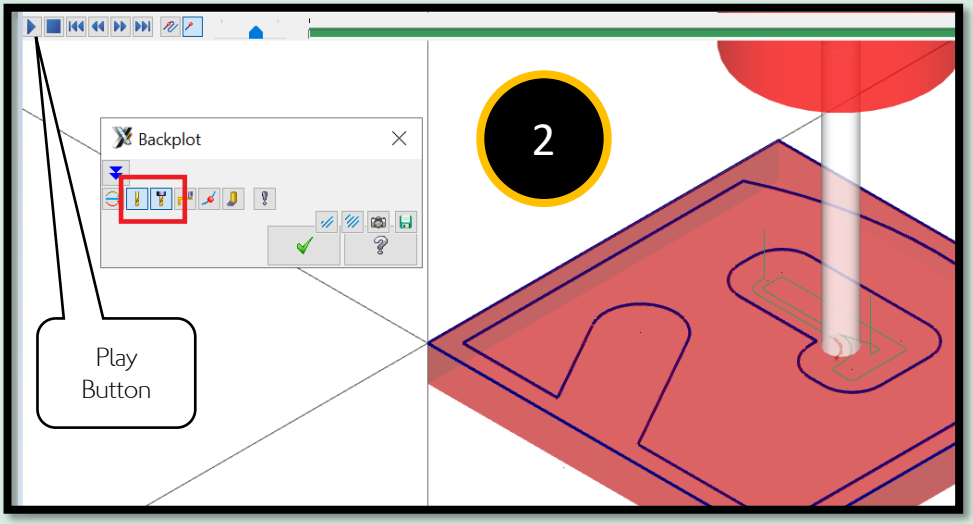
- Select **Linking Parameters** from the **Tree View list.**
- Change the **Top of stock** to **zero** and set the depth to **-3.0.**
- Ensure all the values are set to Absolute.
- Select the **OK** button.

STEP 8 : BACKPLOT THE TOOLPATHS

Backplotting shows the toolpath of cutting tool to cut the part. This display will lets you to spot an errors. 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.

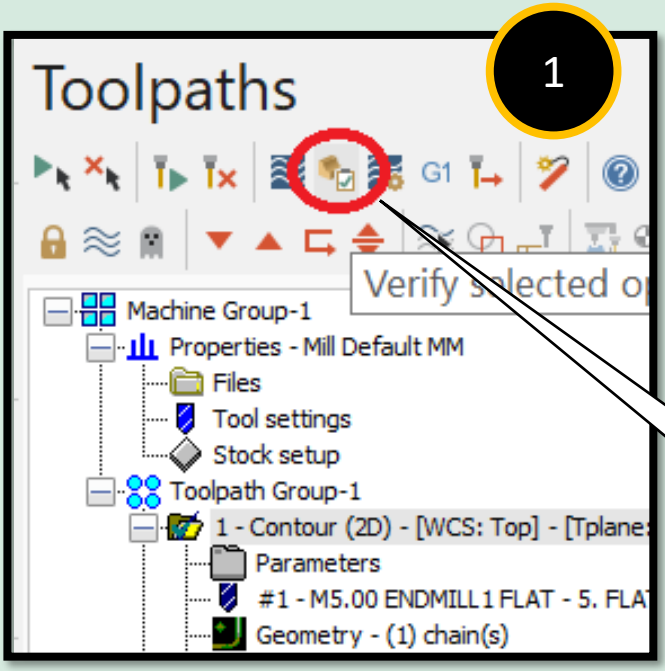
- On the operations button select the **Backplot.**
- Turn on all the buttons .(the cutting tool will appear pushed down). You can see the tool is running and follow the chain of toolpath.
- Adjust the speed of the backplot. Click a **Play** button.





STEP 9 : VERIFY THE TOOLPATHS

Verify allows you to simulate the machining of a part by using as solid model. The solid model created by verification represents the surface roughing and finishing. It also shows collisions if any exist.

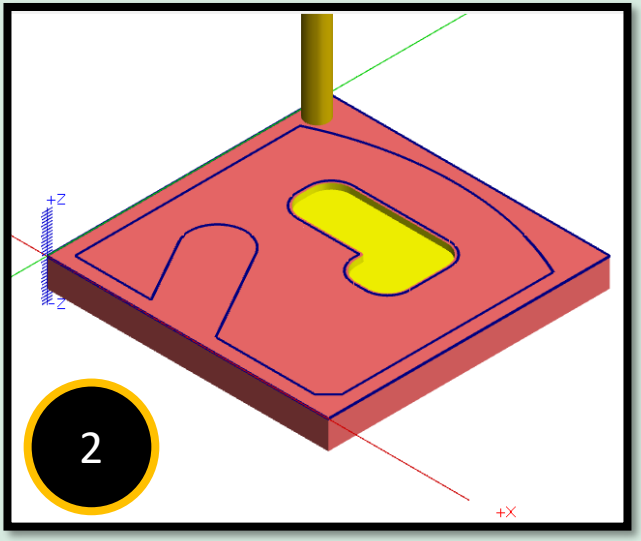


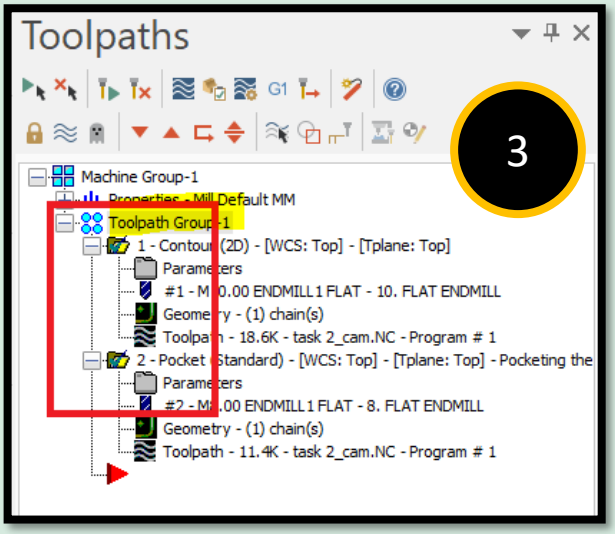
- From **Operation Manager**, choose **Verify selected operations** icon as shown in Figure.

CADCAM Mastercam

STEP 9 : VERIFY THE TOOLPATHS

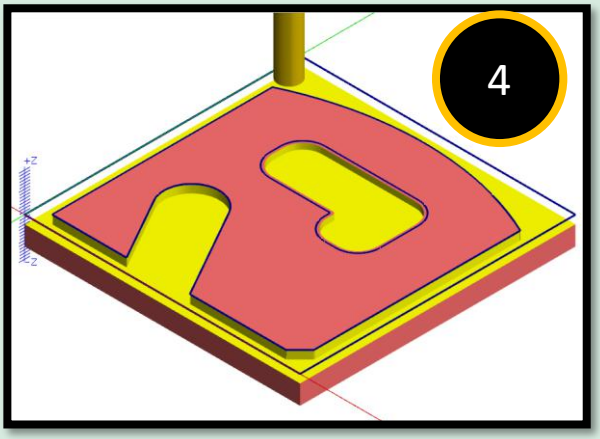
- To start simulation select the Play button.
- The simulation will be shown in window.





- Select the **Toolpath Group I** to pick all the folder for verify process.
- Make sure the folder are pick for playing the simulation.

STEP 9 : VERIFY THE TOOLPATHS

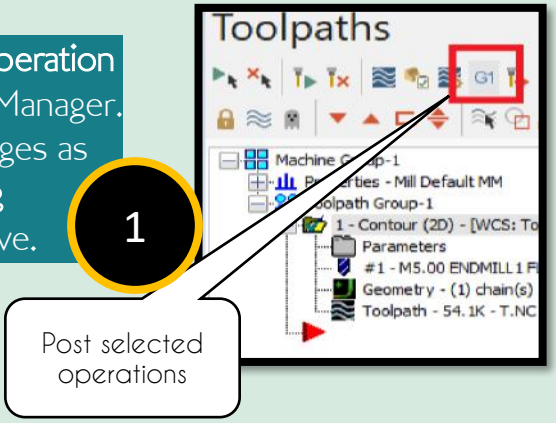


- After play the Verify, the simulation of contouring and pocketing will be shown in the figure.

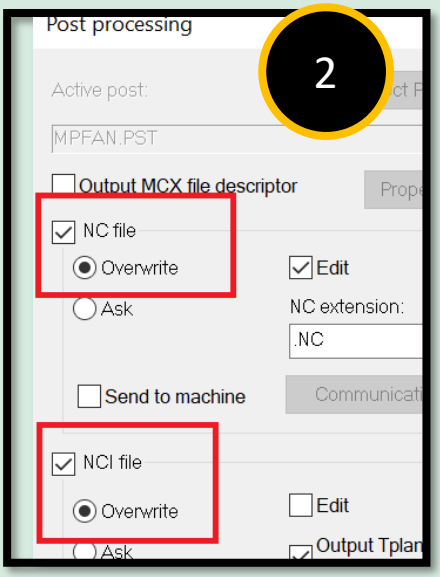
STEP 10 : POST THE FILE

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.

- 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**.



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

CADCAM
Mastercam

STEP 10 : POST THE FILE

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.

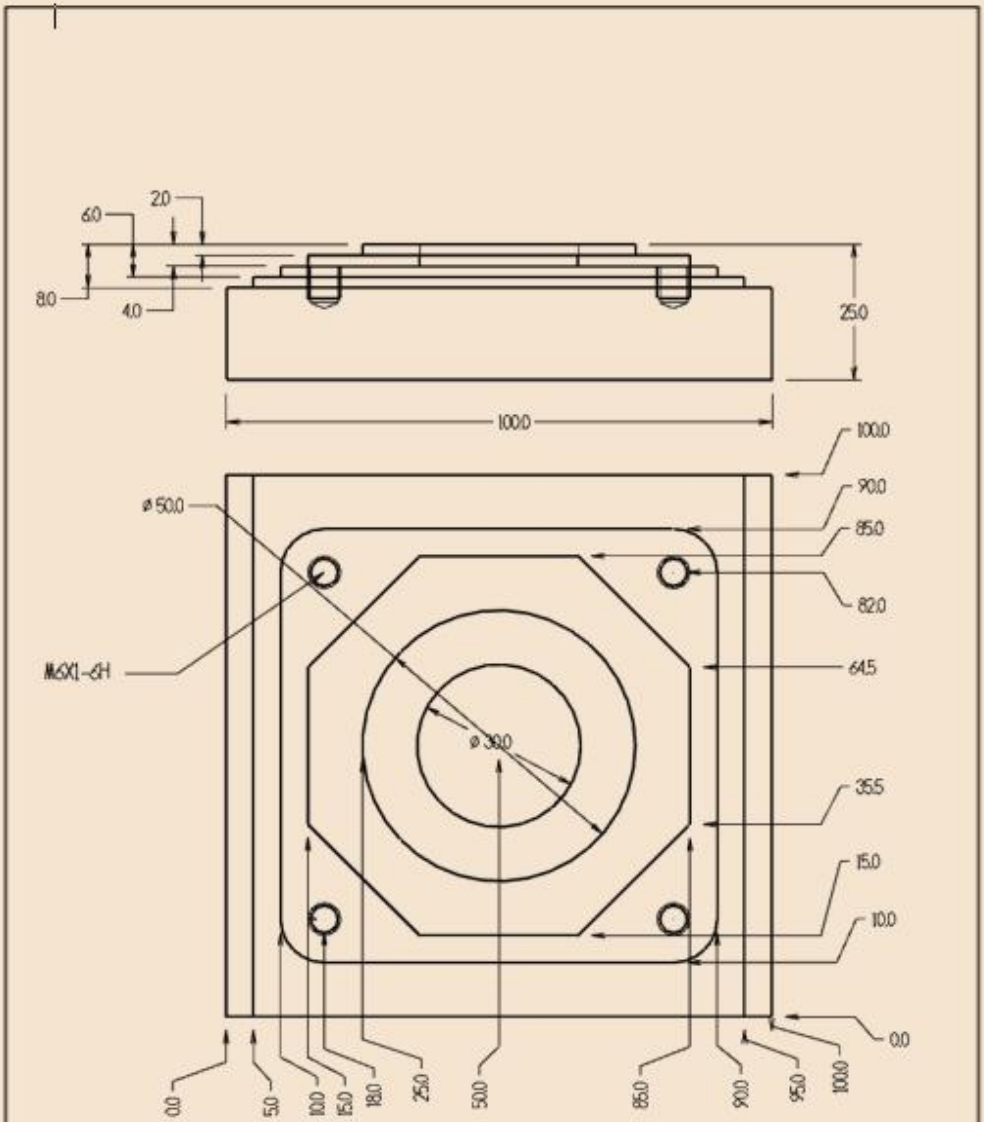
```
O0001(T)
(DATE=DD-MM-YY - 13-08-21 TIME=HH:MM -
(MCX FILE - C:\USERS\LENOVO\DOCUMENTS\
(NC FILE - C:\USERS\LENOVO\DOCUMENTS\M
(MATERIAL - ALUMINUM MM - 2024)
( T1 | 5. FLAT ENDMILL | H1 )
N100 G21
N110 G0 G17 G40 G49 G80 G90
( CONTOUR THE PART DEPTH 3.0MM )
N120 T1 M6
N130 G0 G90 G54 X-34. Y41. A0.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

TUTORIAL

TUTORIAL 1

Instruction :

1. Draw a drafting tasks by using MasterCam software.
2. Scan the **QR codes** to watch step by step how to draft the task.
3. Setup the roughing, semi finishing and finishing process include parameter and tool selection.
4. Generate the NC codes from the task.

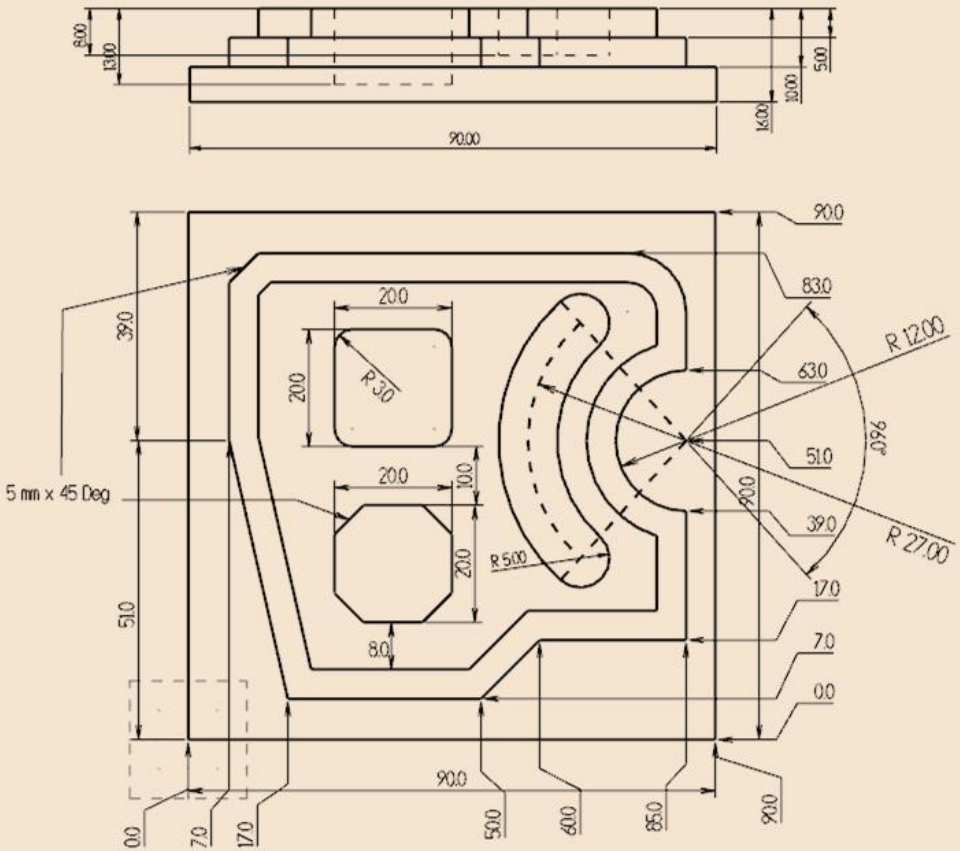


TITLE	TUTORIAL 1	DEPT	MECHANICAL ENGINEERING
		COURSE	DJF41042
POLITEKNIK PORT DICKSON		LAB	CAD CAM
		UNITS	METRIC (MM)

TUTORIAL 2

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	TUTORIAL 2	DEPT	MECHANICAL ENGINEERING
		COURSE	DJF41042
POLITEKNIK PORT DICKSON		LAB	CAD CAM
		UNITS	METRIC (MM)

QUESTIONS AND ANSWER

CADCAM
Mastercam

QUESTION AND ANSWER

Click the link below to answer all the question in LiveWorkSheet website. Good luck!!

https://www.liveworksheets.com/sn2527271kh

REFERENCE

REFERANCE

Main reference supporting the course

Alavala, C. R. (2015). *CAD/CAM: Concepts And Applications*. Delhi: PHI Learning Private Limited.

Additional references supporting the course

Rao, P.N.(2014) *CAD/CAM: Principles and Applications*. New Delhi: McGraw Hill Education (India) Private Limited.

Zeid, I. A., Zeid, I., &Sivasubramanian, R. (2010). *CAD/CAM: Theory and Practice*. New Delhi: Tata McGraw Hill Education Private.

Lee, K. (1999). *Principles of CAD/ CAM/ CAE systems*. Reading, MA: Addison-Wesley.

e ISBN 978-967-2897-41-5