



Competency Based Learning Material (CBLM)

CNC Machining Centre Operation with CAD CAM Level-4

Module: Applying CAM Software for Machining

Code: CBLM-OU-LE-CNCCDM-05-L4-V1



**National Skills Development Authority
Prime Minister's Office
Government of the People's Republic of Bangladesh**

Copyright

National Skills Development Authority
Prime Minister's Office
Level: 10-11, Biniyog Bhaban,
E-6 / B, Agargaon, Sher-E-Bangla Nagar Dhaka-1207, Bangladesh.
Email: ec@nsda.gov.bd
Website: www.nstda.gov.bd.
National Skills Portal: <http://skillsportal.gov.bd>

This Competency Based Learning Materials (CBLM) on “Applying CAM Software for Machining” under the CNC Machining Centre Operation with CAD CAM, Level-4” qualification is developed based on the national competency standard approved by National Skills Development Authority (NSDA)

This document is to be used as a key reference point by the competency-based learning materials developers, teachers/trainers/assessors as a base on which to build instructional activities.

National Skills Development Authority (NSDA) is the owner of this document. Other interested parties must obtain written permission from NSDA for reproduction of information in any manner, in whole or in part, of this Competency Standard, in English or other language.

This Competency Based Learning Materials is a document for the development of curricula, teaching and learning materials, and assessment tools. It also serves as the document for providing training consistent with the requirements of industry in order to meet the qualification of individuals who graduated through the established standard via competency-based assessment for a relevant job.

This document has been developed by NSDA in association with industry representatives, academia, related specialist, trainer, and related employee.

Public and private institutions may use the information contained in this CBLM for activities benefitting Bangladesh.

Approved by the Authority meeting held on.....

How to use this Competency Based Learning Material (CBLM)

The module, Applying CAM Software for Machining contains training materials and activities for you to complete. These activities may be completed as part of structured classroom activities or you may be required to work at your own pace. These activities will ask you to complete associated learning and practice activities in order to gain knowledge and skills you need to achieve the learning outcomes.

1. Review the **Learning Activity** page to understand the sequence of learning activities you will undergo. This page will serve as your road map towards the achievement of competence.
2. Read the **Information Sheets**. This will give you an understanding of the jobs or tasks you are going to learn how to do. Once you have finished reading the **Information Sheets** complete the questions in the **Self-Check**.
3. **Self-Checks** are found after each **Information Sheet**. **Self-Checks** are designed to help you know how you are progressing. If you are unable to answer the questions in the **Self-Check** you will need to re-read the relevant **Information Sheet**. Once you have completed all the questions check your answers by reading the relevant **Answer Keys** found at the end of this module.
4. Next move on to the **Job Sheets**. **Job Sheets** provide detailed information about *how to do the job* you are being trained in. Some **Job Sheets** will also have a series of **Activity Sheets**. These sheets have been designed to introduce you to the job step by step. This is where you will apply the new knowledge you gained by reading the Information Sheets. This is your opportunity to practise the job. You may need to practise the job or activity several times before you become competent.
5. Specification **sheets**, specifying the details of the job to be performed will be provided where appropriate.
6. A review of competency is provided on the last page to help remind if all the required assessment criteria have been met. This record is for your own information and guidance and is not an official record of competency

When working through this Module always be aware of your safety and the safety of others in the training room. Should you require assistance or clarification please consult your trainer or facilitator.

When you have satisfactorily completed all the Jobs and/or Activities outlined in this module, an assessment event will be scheduled to assess if you have achieved competency in the specified learning outcomes. You will then be ready to move onto the next Unit of Competency or Module

Table of Contents

Copyright	ii
How to use this Competency Based Learning Material (CBLM)	vi
Module Content.....	5
Learning Outcomel: Prepare for Computer-Aided Machining Operation	6
Learning Experience 1: Prepare for Computer-Aided Machining Operation	8
Information Sheet 1: Prepare for Computer-Aided Machining Operation	9
Self-Check 1: Prepare for Computer-Aided Machining Operation.....	17
Answer Sheet 1: Prepare for Computer-Aided Machining Operation	18
Job Sheet 1.1: Set Basic Parameters of MASTER CAM Software	19
Specification Sheet 1.1: Set Basic Parameters of MASTER CAM Software.....	20
Job Sheet 1.2: Identify The Menu Function in Master CAM.....	21
Specification Sheet 1.2: Identify The Menu Function in Master CAM	22
Learning Outcome -2: Identify the Sequence of Tool Path and Machining Strategy	23
Learning Experience 2: Identify The Sequence of Tool Path and Machining Strategy	25
Information Sheet 2: Identify The Sequence of Tool Path and Machining Strategy	26
Self-Check Sheet 2: Identify the Sequence of Tool Path and Machining Strategy	32
Answer Sheet 2 Identify the Sequence of Tool Path and Machining Strategy	33
Job Sheet 2.1: Select Pre-Machining Operations as Required.....	35
Specification Sheet 2.1: Select Pre-Machining Operations as Required	36
Job Sheet 2.2: Prepare Work Holding and Clamping to Avoid Tool Clash	37
Specification Sheet 2.2: Job Name: Prepare Work Holding and Clamping to Avoid Tool Clash.....	38
Learning Outcome -3: Create 2D Tool Paths	39
Learning Experience 3: Create 2D Tool Paths	42
Information Sheet 3: Create 2D Tool Paths.....	43
Job Sheet 3: Create A 2D Drill Toolpath that Drills All Four Holes of the Part.....	57
Activity 3.1: Configure Machine and Setting Up Stock for Drilling	58
Activity 3.2: Create a Drill Toolpath That Drills All Four Holes of The Part	60
Activity 3.3: Creating a Dynamic Mill Toolpath	62
Specification Sheet 3.2: Create A 2D Drill Toolpath That Drills All Four Holes of The Part	65
Learning Outcome -4: Create 3D Tool Paths	66
Learning Experience 4: Create 3D Tool Paths	75
Information Sheet 4: Create 3D Tool Paths.....	76
Self-Check 4 Create 3D Tool	82
Answer Sheet 4 Create 3D Tool	83
Job Sheet 4: Perform 3D Machining in Mastercam as Per Selected Mechanical 3D Model	84
Activity 4.1: Assigning a Machine Definition and Setting Stock for Machining	85
Activity 4.2: Perform program for Drilling Holes.....	86
Activity 4.3: Perform Program for Roughing the Pocket.....	88
Activity 4.4: Perform CAM Programing (Roughing the Slot)	90
Activity 4.5: Rest Roughing the Interior	92
Activity 4.6 Perform Program for Finishing Pocket Floor.....	94

Activity4.7 Perform Program for Finishing the Slot	95
Activity 4.8 Perform Program for Roughing the Exterior.....	97
Activity 4.9 Perform Program for Finishing the Exterior	99
Specification Sheet 4: Perform 3D Machining in Mastercam as Per Selected Mechanical 3D Model.....	101
Review Of Competency.....	102

Module Content

Unit of Competency	Apply CAM Software for Machining
Unit Code	OU-LE-CNCCDM-05-L4-V1
Module Title	Applying CAM Software for Machining
Module Descriptor	<p>This module covers the skills, knowledge and attitudes required to apply CAM software machining.</p> <p>It specifically includes preparing for computer aided machining operation, identifying the sequence of tool path and machining strategy, creating 2D tool paths, and creating 3D tool paths.</p>
Nominal Hours	50 Hours
Learning Outcome	<p>After completing the practice of the module, the trainees will be able to perform the following jobs:</p> <ol style="list-style-type: none"> 1. Prepare for computer-aided machining operation 2. Identify the sequence of tool path and machining strategy 3. Create 2D tool paths 4. Create 3D tool paths

Assessment Criteria

1. Computer-aided machining software is installed and verify the system requirement
2. Basic parameter is set of the CAM software
3. Menu functions are identified
4. Appropriate tools and equipment are used to produce a drawing as per job requirements
5. CAD files are imported as required
6. CAD models are created as required
7. The workpiece is verified and required machines are identified
8. Appropriate tool path sequence is selected for a model-specific machining operation
9. Appropriate tool diameter and length is selected to avoid chattering
10. Pre-machining operation is selected if required
11. Work holding & clamping is prepared to avoid tool clash
12. Appropriate cutting parameter is selected
13. Planes are identified
14. Machine definition and post-processor is selected
15. Origin & stock setup is defined
16. 2D tool paths are executed
17. Tapping and thread milling is executed
18. Tool paths are verified
19. G&M codes are generated
20. 3D tool path is prepared
21. level as per matching strategy is set
22. Surface is created as per requirement
23. Silhouette command is executed
24. Edge curve command is executed
25. Drive surface, containment region, check surface is selected
26. 3D rough & finishing tool path is executed
27. Stock model & rest milling is executed
28. 3D tool path is verified
29. G&M code is generated

Learning Outcome1: Prepare for Computer-Aided Machining Operation

Learning Outcome -1: Prepare for computer-aided machining operation	
Assessment Criteria	<ol style="list-style-type: none"> 1. Computer-aided machining software is installed and verify the system requirement 2. Basic parameter is set of the CAM software 3. Menu functions are identified 4. Appropriate tools and equipment are used to produce a drawing as per job requirements 5. CAD files are imported as required 6. CAD models are created as required
Conditions and Resources	<ol style="list-style-type: none"> 1. Workplace or Simulated Workplace 2. CBLM 3. Handout 4. Taptop 5. Multimedia Projector 6. Paper, Pen, Pencil, 7. Internet Facilities 8. White Board and 9. Audio Video Devices 10. Necessary tools, equipment and accessories 11. Necessary PPE
Contents	<ol style="list-style-type: none"> 1. Computer-aided machining software <ul style="list-style-type: none"> ▪ Master Cam/Solid Cam/Siemens NX/Fusion 360 2. Basic parameters of CAM software <ul style="list-style-type: none"> ▪ Inch ▪ Metric ▪ Decimal 3. Menu functions <ul style="list-style-type: none"> ▪ File ▪ Edit ▪ View ▪ Analyze ▪ Create ▪ Solid ▪ Transform ▪ Class/type of machine ▪ Tool path ▪ Setting ▪ wireframe ▪ Help 4. CAD File Import Requirements 5. Requirements for creating CAD models

Job/Task/Activity	<ol style="list-style-type: none"> 1. Install computer-aided machining software and verify system requirements 2. Set basic parameters of CAM software 3. Identify the menu function 4. Import the CAD file as needed 5. Create CAD model as required
Training Method	<ol style="list-style-type: none"> 1. Discussion 2. Presentation 3. Demonstration 4. Guided Practice 5. Individual Practice 6. Project Work 7. Problem Solving 8. Brainstorming 9. Role Play
Assessment Method	<ol style="list-style-type: none"> 1. Written Test 2. Demonstration 3. Oral questioning 4. Portfolio

Learning Experience 1: Prepare for Computer-Aided Machining Operation

In order to achieve the objectives stated in this learning guide, you must perform the learning steps below. Beside each step are the resources or special instructions you will use to accomplish the corresponding activity.

Learning Activities	Recourses/Special Instructions
1. Trainee will ask the instructor about the learning materials	1. Instructor will provide the learning materials 'Prepare for computer-aided machining operation
2. Read the Information sheet and complete the Self-Checks & Check answer sheets on “Prepare for computer-aided machining operation”	2. Read Information sheet 1: Prepare for computer-aided machining operation 3. Self-check 1: Prepare for computer-aided machining operation 4. Check your answer with Answer key 1 Prepare for computer-aided machining operation
3. Read the Job/Task Sheet and Specification Sheet and perform job/Task	5. Job/Task Sheet and Specification Sheet Job Sheet 1.1: Set basic parameters of MASTER CAM software Specification sheet 1.1: Set basic parameters of MASTER CAM software Job Sheet 1.2: Identify the menu function in Master CAM Specification sheet 1.2: Identify the menu function in Master CAM

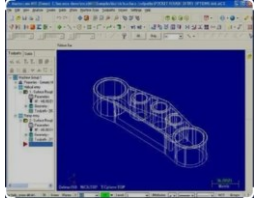
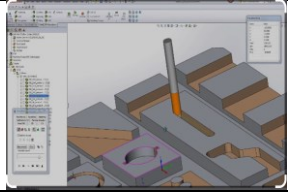
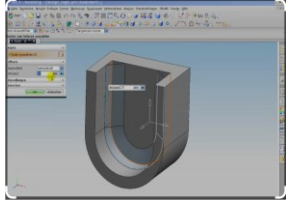
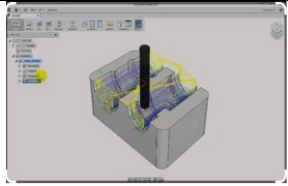
Information Sheet 1: Prepare for Computer-Aided Machining Operation

Learning Objective: After completion of this information sheet, the learners will be able to explain, define and interpret the following contents:

- 1.1 Computer-aided machining software
- 1.2 Basic parameters of CAM software
- 1.3 Menu functions
- 1.4 CAD File Import Requirements
- 1.5 Requirements for creating CAD models

1.1 Computer-Aided Machining Software

Computer-aided machining (CAM) software is used to generate toolpaths and instructions for CNC (Computer Numerical Control) machines to manufacture parts from various materials. These programs play a crucial role in automating the manufacturing process and optimizing efficiency. CAM software takes into account factors such as tool geometry, material properties, cutting parameters, and machine capabilities to generate precise machining instructions. It allows users to simulate and visualize the machining process before actual production, reducing the risk of errors and optimizing machining strategies for cost-effectiveness and quality. CAM software often integrates with CAD (Computer-Aided Design) software to streamline the transition from design to manufacturing.

<p>Master CAM Mastercam: Mastercam is a popular CAM software used primarily in the manufacturing industry for tasks like CNC (Computer Numerical Control) machining, milling, turning, and wire EDM (Electrical Discharge Machining). It provides a wide range of features for creating toolpaths and controlling machining operations.</p>	
<p>Solid CAM Solid CAM: Solid CAM is another CAM software that integrates with SolidWorks, providing users with seamless integration between CAD (Computer-Aided Design) and CAM. It offers a range of CNC machining capabilities, including milling, turning, and wire EDM.</p>	
<p>Siemens NX Siemens NX: Siemens NX is a comprehensive CAD/CAM/CAE (Computer-Aided Engineering) software suite developed by Siemens. It's widely used in various industries for product design, engineering analysis, and manufacturing. The CAM module of Siemens NX offers advanced machining capabilities for milling, turning, and multi-axis machining.</p>	
<p>Fusion 360 Fusion 360: Fusion 360 is a cloud-based CAD/CAM software developed by Autodesk. It offers integrated CAD, CAM, and CAE tools, making it suitable for product development and manufacturing workflows. Fusion 360 provides a wide range of machining strategies for CNC milling, turning, and additive manufacturing.</p>	

1.2 Basic Parameters of CAM Software

The basic parameters you have listed seem to relate to units of measurement and formatting options within CAM software. Here is a brief explanation of each:

a. **Inch**

This parameter refers to the unit of measurement commonly used in countries like the United States, Liberia, and Myanmar. In CAM software, selecting "inch" as the unit of measurement means that dimensions, tool sizes, cutting parameters, and other measurements will be displayed and calculated in inches.

b. **Metric**

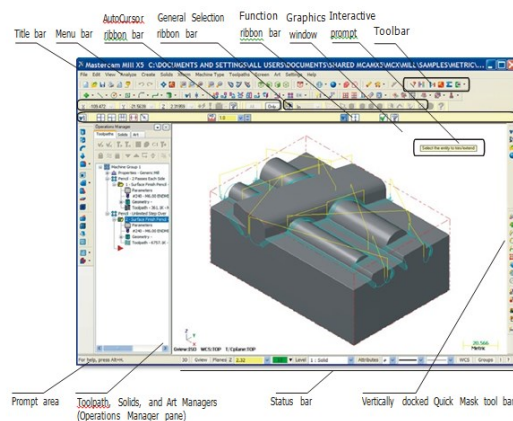
Metric units are the standard units of measurement in most parts of the world, including Europe, Asia, and Oceania. When selecting "metric" in CAM software, all dimensions, tool sizes, cutting parameters, and other measurements will be displayed and calculated in metric units such as millimeters (mm) or centimeters (cm).

c. **Decimal**

Decimal refers to a numerical system based on the powers of 10, where numbers are represented using digits 0 through 9. In the context of CAM software, selecting "decimal" typically means that dimensions and measurements will be displayed and entered as decimal numbers, rather than fractions or other formats.

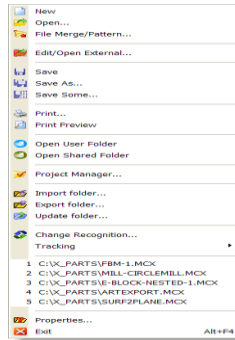
These basic parameters allow users to customize the units of measurement and formatting preferences within CAM software to align with their specific requirements and regional conventions.

1.3 Menu Functions



a. **File**

This menu likely includes options for creating, opening, saving, and managing files. Users can also find commands for setting up and configuring project properties, such as units, templates, and file formats.

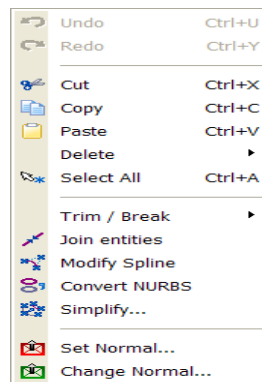


Use File menu functions to open, edit, print, save, track, and compare files. Mastercam provides seamless integration with most popular CAD/CAM file formats. Conversion occurs automatically when opening and saving files. When saving files,

you can save all or only some of the entities to a specified format, and include descriptive text and a thumbnail image of the geometry with the file data. You can also import and export files to and from specified directories, and merge data into the current file from a pattern file (for example, a nut or bolt you use repeatedly in Mastercam files).

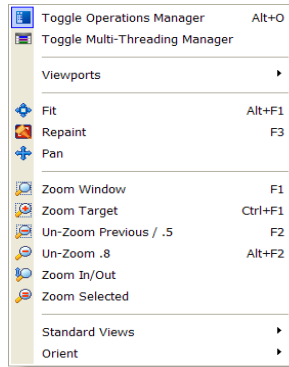
b. Edit

The Edit menu in Mastercam probably contains commands for modifying geometry, adjusting parameters, and managing elements within the CAM environment. This could include options for editing toolpaths, modifying features, copying, and pasting, and undoing or redoing actions. This menu provides access to functions you use to edit geometry, such as the Join entities, Modify Spline, Convert NURBS, and Simplify functions, and the Trim/Break submenu functions. Other Edit menu functions allow you to cut, copy, paste, delete, or select all entities in the graphics window.



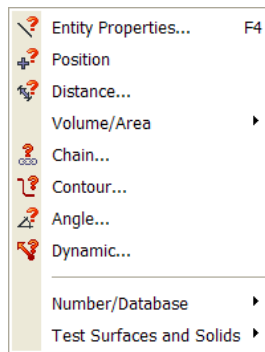
c. View:

The View menu likely offers options for customizing the display and visibility of elements within the Mastercam interface. Users can find commands for zooming, panning, rotating the view, and controlling the display of components or toolpaths. The View menu helps you manage the appearance and orientation of the Mastercam graphics window.



d. Analyze

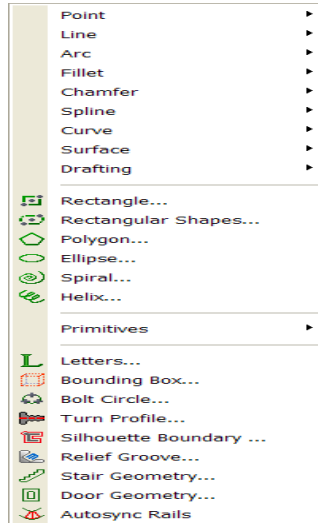
The Analyze menu in Mastercam may contain tools for inspecting and evaluating aspects of the design or machining process. This could include commands for measuring distances, angles, or dimensions, as well as tools for checking for interference, collisions, or material thickness. Use Analyze menu functions to view and edit entity properties.



Some analyse functions provide a report function that lets you save the information to a file. You can also use analyze functions to modify the color, line style and width, or point style attributes of a single entity, or apply the same attributes to all the entities you select in the graphics window.

e. Create

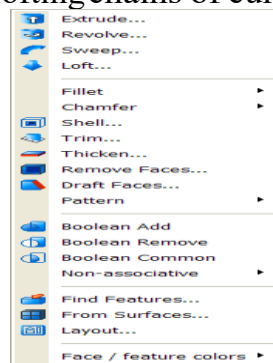
The Create menu likely provides commands for generating new geometry, features, or toolpaths within the Mastercam environment. Users can find options for creating sketches, solid models, surfaces, and defining machining operations like milling, turning, or drilling. This extensive menu inc



ludes all Sketcher (Create Geometry), Curve, Surfaces, and Drafting functions. Use these functions to create points, lines, arcs, splines, curves, fillets, chamfers, surfaces, drafting entities, and basic geometry, including rectangles, rectangular shapes, polygons, ellipses, spirals, and helixes.

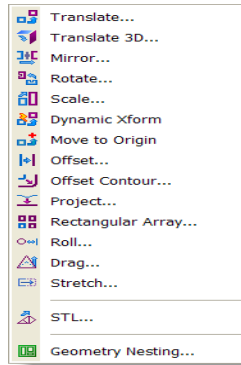
f. Solid

The Solid menu probably contains commands specific to solid modeling operations within Mastercam. This could include tools for creating, modifying, and analyzing solid geometry, such as extruding, revolving, filleting, and performing Boolean operations. Functions in this menu are available only if your Mastercam installation includes Mastercam ds. Use these functions to define a solid by extruding, revolving, sweeping, or lofting chains of curves.



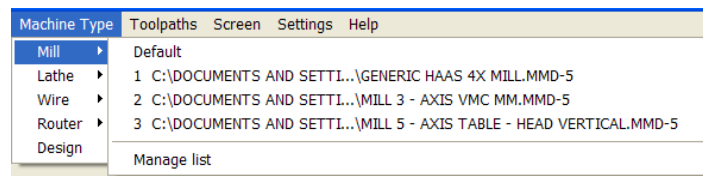
g. Transform:

The Transform menu in Mastercam likely provides commands for manipulating and transforming geometry within the CAM environment. Users can find options for scaling, rotating, mirroring, translating, or offsetting objects or toolpaths. Use Xform (transform) functions to move or copy selected entities by mirroring, rotating, scaling, offsetting, translating, stretching, and rolling them. With some functions, you can join the copied entities to the originals. When you perform a transform function, Mastercam creates a temporary group from the original entities and a result from the transformed entities.



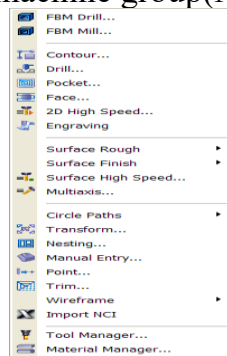
h. Class/type of machine

This menu may offer options for selecting or configuring the class or type of CNC machine for which toolpaths are being generated. Users can specify the machine's capabilities, such as its axis configuration, maximum speeds, and tool change capabilities. Use this menu to select a machine type by choosing a machine definition. Your access to the different types of machine definitions in the menu is based on the Mastercam products you have installed.



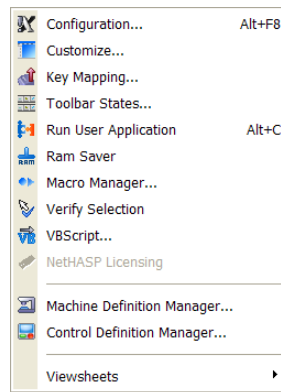
i. Tool path

The Tool Path menu probably contains commands and settings related to generating and editing toolpaths for machining operations in Mastercam. Users can select machining strategies, specify cutting parameters, optimize toolpaths, and simulate the machining process. Functions in this menu allow you to create and edit the toolpaths required to cut the part using the specified machine definition. Machine type associated with the active machine group (Mill, Lathe, Router, or Wire)



j. Setting

The setting menu in Mastercam likely provides access to various software preferences, options, and configurations. Users can customize settings related to units, display preferences, keyboard shortcuts, file management, and other aspects of the software environment.

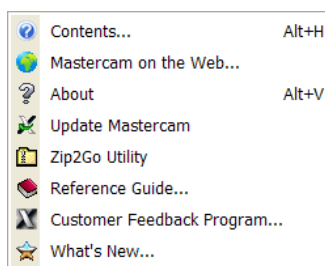


k. Wireframe

This term doesn't seem to relate directly to Mastercam. If it's a specific feature or function within Mastercam, its purpose would need to be clarified.

l. Help

The Help menu in Mastercam typically offers access to documentation, tutorials, support resources, and information about the software. Users can find assistance with using the software, troubleshooting issues, and accessing online resources for further learning and support



1.4 Cad File Import Requirements

When importing CAD files into Mastercam for machining purposes, there are several requirements and considerations to ensure a smooth and accurate conversion process. Here are some common requirements:

- a. Supported file formats:** Mastercam typically supports various CAD file formats for import, including STEP, IGES, SAT (ACIS), Parasolid, DXF, and DWG. Ensure that your CAD file is saved in one of these formats for compatibility.
- b. Clean geometry:** Before importing, ensure that the CAD model has clean geometry without errors such as gaps, overlaps, or self-intersections. Use CAD software tools to repair any issues before exporting.
- c. Unified units:** Make sure that the CAD model and Mastercam project are using the same units of measurement (e.g., inches or millimeters). Consistency in units prevents scaling issues during import.
- d. Layer organization:** Organize the CAD model into logical layers or groups based on part features, machining operations, or toolpath strategies. This facilitates efficient

selection and management within Mastercam.

- e. **Simplified geometry:** Complex CAD models may contain unnecessary details or features that are not relevant for machining. Simplify the geometry by removing unnecessary elements or detailing to improve processing speed and efficiency.
- f. **Bounding box:** Define a bounding box or stock geometry that encompasses the entire part or machining area. This helps establish the workpiece dimensions and orientation within Mastercam.
- g. **Part orientation:** Set the correct orientation and coordinate system for the CAD model to align with the desired machining setup in Mastercam. Ensure that the part is positioned correctly relative to the machine axes.
- h. **Surface normal direction:** Verify that the surface normal of the CAD model are oriented correctly, especially for parts with complex surfaces or features. Inverted normal can affect toolpath generation and machining simulation.
- i. **Cad model scale:** Check the scale of the CAD model to ensure that it matches the intended dimensions of the physical part. Adjust the scale if necessary, to achieve the desired size in Mastercam.
- j. **Cad software compatibility:** Ensure that the CAD software used to create the model is compatible with Mastercam for accurate file conversion. Some CAD programs may have specific settings or export options for better compatibility with CAM software.

1.5 Requirements For Creating CAD Models

When creating CAD models specifically for use in Mastercam for machining purposes, there are several requirements and best practices to consider to ensure compatibility and optimal performance. Here's a comprehensive list:

- a. **Choose compatible cad software:** Use CAD software that is compatible with Mastercam, such as SolidWorks, Autodesk Inventor, Solid Edge, or PTC Creo. These programs typically offer better integration and support for exporting files in formats that Mastercam can import seamlessly.
- b. **Select appropriate units:** Set the units of measurement (e.g., inches or millimeters) consistently across the CAD model and Mastercam project to avoid scaling issues during import and machining.
- c. **Model accuracy and detail:** Create CAD models with accurate dimensions and sufficient detail for the machining requirements. Include all necessary features, surfaces, holes, fillets, and other geometries required for the final part.
- d. **Clean geometry:** Ensure that the CAD model has clean geometry without errors such as gaps, overlaps, or self-intersections. Use CAD software tools to check and repair any issues before exporting.

Self-Check 1: Prepare for Computer-Aided Machining Operation

1. What is CAM?

Answer:

2. What is Master CAM?

Answer:

3. What is Edit Menu in Master CAM?

Answer:

4. What is Analyze in Master CAM?

Answer:

5. What is Solid menu in Master CAM?

Answer:

6. What is Tool Path in Master CAM?

Answer:

7. What is Supported File Formats in Master CAM?

Answer:

Answer Sheet 1: Prepare for Computer-Aided Machining Operation

1. What is CAM?

Answer: Computer-aided machining (CAM) software is used to generate toolpaths and instructions for CNC (Computer Numerical Control) machines to manufacture parts from various materials.

2. What is Master CAM?

Answer: Mastercam: Mastercam is a popular CAM software used primarily in the manufacturing industry for tasks like CNC (Computer Numerical Control) machining, milling, turning, and wire EDM (Electrical Discharge Machining). It provides a wide range of features for creating toolpaths and controlling machining operations.

3. What is Edit Menu in Master CAM?

Answer: The Edit menu in Mastercam probably contains commands for modifying geometry, adjusting parameters, and managing elements within the CAM environment. This could include options for editing toolpaths, modifying features, copying and pasting, and undoing or redoing actions.

4. What is Analyze in Master CAM?

Answer: The Analyze menu in Mastercam may contain tools for inspecting and evaluating aspects of the design or machining process. This could include commands for measuring distances, angles, or dimensions, as well as tools for checking for interference, collisions, or material thickness.

5. What is Solid Menu in Master CAM?

Answer: The Solid menu probably contains commands specific to solid modeling operations within Mastercam. This could include tools for creating, modifying, and analyzing solid geometry, such as extruding, revolving, filleting, and performing Boolean operations.

6. What is Tool Path in Master CAM?

Answer: The Tool Path menu probably contains commands and settings related to generating and editing toolpaths for machining operations in Mastercam. Users can select machining strategies, specify cutting parameters, optimize toolpaths, and simulate the machining process.

7. What is Supported File Formats in Master CAM?

Answer: Mastercam typically supports various CAD file formats for import, including STEP, IGES, SAT (ACIS), Parasolid, DXF, and DWG. Ensure that your CAD file is saved in one of these formats for compatibility.

Job Sheet 1.1: Set Basic Parameters of MASTER CAM Software

Name of Job: Set basic parameters of MASTER CAM software

Procedure:

1. Follow OSH
2. Read job sheet and specification sheet.
3. Install Mastercam software on your computer to follow the installation instructions provided with the software.
4. Launch Mastercam and activate your license if required to follow the prompts to enter your license information and activate the software.
5. Open Mastercam and navigate to the Configuration Manager and set up various parameters for your machining environment.
6. Create or select a machine definition that matches the CNC machine you'll be using specifying the machine type, axis configuration, and other relevant parameters.
7. Define the control type for your CNC machine.
8. Set up specifying details such as the post processor, units (inches or millimeters), and other control-specific settings.
9. Set up stock material and dimensions.
10. Set up tool library by adding or importing tool definitions such as tool type, dimensions, and cutting parameters.
11. Set up material properties in the material library such as material type, cutting speeds, and feeds.
12. Set up System (WCS): Define the work coordinate system for your part. This establishes the origin and orientation of your machining operations relative to the part.
13. Configure toolpath parameters such as cutting speeds, feeds, stepovers, and tolerances.
14. Configure post processor settings to generate G-code compatible with your CNC machine.
15. Configure simulation settings to visualize toolpaths and verify machining operations before sending programs to the CNC machine.
16. set up all the parameters, save your configuration settings for future use.

Specification Sheet 1.1: Set Basic Parameters of MASTER CAM Software

Name of Job: Set basic parameters of MASTER CAM software

Conditions for the job: You must practice safe operation procedures at all times during the assessment. You will be required to demonstrate OSH competencies Set basic parameters of MASTER CAM software



Required Personal Protective Equipment (PPE): Pre/person

Sl.	Name of the PPE	Specification	Unit	Quantity	Remarks
1	Hand Gloves	Cotton	pair	01	
2	Mask	N 95	Nos	01	
3	Safety shoe	As required	pair	01	
4	Safety goggles	Standard	Nos	01	
5	Apron	Standard	Nos	01	

Required Tools and Equipment

Sl.	Name of tools and equipment	Specification	Unit	Quantity	Remarks
1	Computer	Core i5 Ram 8GB	No	01	

Required Materials

Sl.	Name Material	Specification	Unit	Quantity	Remarks
2	Master cam software		No	01	

Job Sheet 1.2: Identify The Menu Function in Master CAM

Name of Job: Identify the menu function in Master CAM

Procedure:

1. Follow OSH
2. Read job sheet and specification sheet.
3. Machine Type Selection:
4. Navigate to the "Machine" menu.
5. Choose "Machine Type" such as milling, turning, or routing.
6. Press Tool Manager and Go to the "Toolpaths" menu.
7. Configure "Tool Manager" or a similar option to access the tool library and define or edit tool parameters.
8. Identify Material Library for options like "Material Library" or "Material Properties" to define materials and their cutting parameters.
9. Identify located within the "Toolpaths" or "Machine" menu.
10. Identify "Work Coordinate System" or "Set WCS" to define the origin and orientation of machining operations.
11. Identify "Post Processor" or "Configure Post Processor" to select or customize the post processor for generating G-code.
12. Identify "Simulation Control" or "Verification Settings" to configure simulation parameters.
13. Save Configuration:

Specification Sheet 1.2: Identify The Menu Function in Master CAM

Name of Job: Set basic parameters of MASTER CAM software

Conditions for the job: You must practice safe operation procedures at all times during the assessment. You will be required to demonstrate OSH competencies **Identify the menu function in Master cam software.**



Required Personal Protective Equipment (PPE): Pre/person

Sl.	Name of the PPE	Specification	Unit	Quantity	Remarks
6	Hand Gloves	Cotton	pair	01	
7	Mask	N 95	Nos	01	
8	Safety shoe	As required	pair	01	
9	Safety goggles	Standard	Nos	01	
10	Apron	Standard	Nos	01	

Required Tools and Equipment

Sl.	Name of tools and equipment	Specification	Unit	Quantity	Remarks
3	Computer	Core i5 Ram 8GB	No	01	

Required Materials

Sl.	Name Material	Specification	Unit	Quantity	Remarks
4	Master cam software		No	01	

Learning Outcome -2: Identify the Sequence of Tool Path and Machining Strategy

Learning Outcome -2: Identify the sequence of tool path and machining strategy	
Assessment Criteria	<ol style="list-style-type: none"> 1. The workpiece is verified and required machines are identified 2. Appropriate tool path sequence is selected for a model-specific machining operation 3. Appropriate tool diameter and length is selected to avoid chattering 4. Pre-machining operation is selected if required 5. Work holding & clamping is prepared to avoid tool clash 6. Appropriate cutting parameter is selected
Conditions and Resources	<ol style="list-style-type: none"> 1. Workplace or Simulated Workplace 2. CBLM 3. Handout 4. Laptop 5. Multimedia Projector 6. Paper, Pen, Pencil, 7. Internet Facilities 8. White Board and 9. Audio Video Devices 10. Necessary Tools and Equipment 11. Necessary Materials 12. Necessary PPE

<p>Contents</p>	<ol style="list-style-type: none"> 1. Necessary machines 2. Model-Specific Machining 3. Tool path sequence/continuity 4. Suitable tool diameter and length 5. Pre-machining operation <ul style="list-style-type: none"> ▪ Face Milling ▪ Side milling 6. Work holding and clamping <ul style="list-style-type: none"> ▪ Vice ▪ Chalk ▪ Clamping kit set ▪ Parelle bar ▪ Angle plate 7. Cutting parameters <ul style="list-style-type: none"> ▪ Feed ▪ RPM ▪ Deft of Cut ▪ Plunge ▪ Cutting speed
<p>Job/Task/Activity</p>	<ol style="list-style-type: none"> 1. Identify the required machine by inspecting the workpiece 2. Select the appropriate tool path sequence for a model-specific machining operation 3. Select pre-machining operations as required 4. Prepare work holding and clamping to avoid tool clash 5. Select the appropriate cutting parameters
<p>Training Method</p>	<ol style="list-style-type: none"> 1. Discussion 2. Presentation 3. Demonstration 4. Guided Practice 5. Individual Practice 6. Project Work 7. Problem Solving 8. Brainstorming
<p>Assessment Method</p>	<ol style="list-style-type: none"> 1. Written Test 2. Demonstration 3. Oral questioning 4. Portfolio

Learning Experience 2: Identify The Sequence of Tool Path and Machining Strategy

In order to achieve the objectives stated in this learning guide, you must perform the learning steps below. Beside each step are the resources or special instructions you will use to accomplish the corresponding activity.

Learning Activities	Recourses/Special Instructions
1. Trainee will ask the instructor about the learning materials	1. Instructor will provide the learning materials 'Apply CAM Software for Machining
2. Read the Information sheet and complete the Self Checks & Check answer sheets on " Identify the sequence of tool path and machining strategy "	2. Read Information sheet 1 Identify the sequence of tool path and machining strategy 3. Answer Self-check 1: Identify the sequence of tool path and machining strategy 4. Check your answer with Answer key 1 Identify the sequence of tool path and machining strategy
3. Read the Job/Task Sheet and Specification Sheet and perform job/Task	5. Job/Task Sheet and Specification Sheet Job Sheet 2.1: Select pre-machining operations as required Specification sheet 2.1: Select pre-machining operations as required Job Sheet 2.2: Prepare work holding and clamping to avoid tool clash Specification sheet 2.2: Prepare work holding and clamping to avoid tool clash



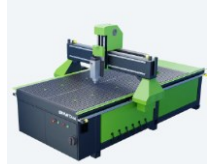

Information Sheet 2: Identify The Sequence of Tool Path and Machining Strategy



Learning Objective: After completion of this information sheet, the learners will be able to explain, define and interpret the following contents:

- 2.1 Necessary machines
- 2.2 Model-Specific Machining
- 2.3 Tool path sequence/continuity
- 2.4 Suitable tool diameter and length
- 2.5 Pre-machining operation
- 2.6 Work holding and clamping
- 2.7 Cutting parameters
- 2.8 Cutting speed

2.1 Necessary Machines

In Mastercam, a CAM (Computer-Aided Manufacturing) software, the software itself is used to generate toolpaths and instructions for various types of CNC (Computer Numerical Control) machines. The choice of CNC machine depends on the specific machining operations required for the given project. Here are some common types of CNC machines that are compatible with Mastercam:

<p>CNC milling machine: CNC milling machines are used to remove material from a workpiece by rotating a cutting tool against it. Mastercam can generate toolpaths for various milling operations, including 2D milling, 3D milling, pocketing, contouring, and engraving. Different configurations of CNC milling machines, such as vertical mills, horizontal mills, and multi-axis mills, can be used depending on the complexity and size of the parts being machined.</p>	
<p>CNC turning machine (lathe): CNC turning machines, also known as lathes, are used to create cylindrical parts by rotating the workpiece against a stationary cutting tool. Mastercam can generate toolpaths for turning operations such as facing, roughing, finishing, threading, and grooving. CNC lathes come in various configurations, including horizontal lathes, vertical lathes, and multi-axis lathes.</p>	
<p>CNC router: CNC routers are used for cutting and shaping various materials, including wood, plastic, foam, and composites. Mastercam can generate toolpaths for routing operations such as profile cutting, pocketing, drilling, and carving. CNC routers are commonly used in woodworking, signage, and prototyping industries.</p>	
<p>CNC plasma cutter: CNC plasma cutters are used to cut metal sheets and plates using a high-velocity jet of ionized gas (plasma). Mastercam can generate toolpaths for plasma cutting operations, including straight cuts, bevel cuts, and hole piercing. CNC plasma</p>	

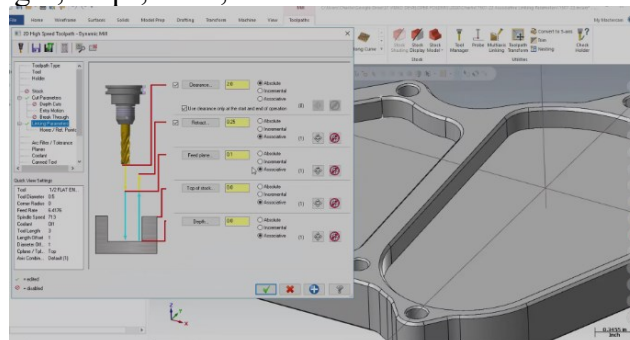
cutters are commonly used in metal fabrication, automotive, and aerospace industries.	
CNC waterjet cutter: CNC waterjet cutters use a high-pressure jet of water mixed with abrasive particles to cut a wide range of materials, including metal, stone, glass, and composites. Mastercam can generate toolpaths for waterjet cutting operations, allowing for precise and intricate cutting of complex shapes.	
CNC wire edm machine: CNC wire EDM (Electrical Discharge Machining) machines use electrical discharges to remove material from a workpiece. Mastercam can generate toolpaths for wire EDM operations, including contouring, pocketing, and threading. CNC wire EDM machines are commonly used for producing precision parts with tight tolerances in industries such as aerospace, medical, and toolmaking.	

2.2 Model-Specific Machining

Model-specific machining in Mastercam typically involves creating toolpaths and machining strategies tailored to the specific geometry and requirements of the model you're working with.

Importing the Model: Start by importing your 3D model into Mastercam. This could be a CAD file in formats like IGES, STEP, or directly from CAD software like SolidWorks or AutoCAD.

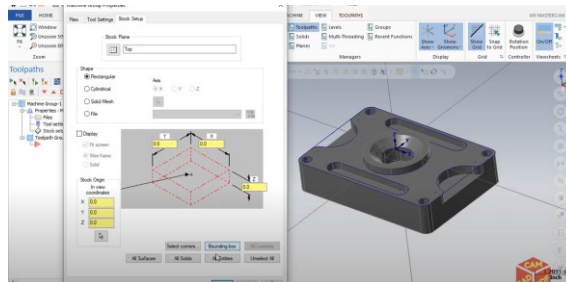
- a. **Creating toolpaths:** Mastercam offers various toolpath strategies for different machining operations, such as milling, turning, routing, and wire EDM. Choose the appropriate toolpath type based on the machining operation you need to perform.
- b. **Geometry selection:** Once you've selected the type of toolpath you want to create, you'll need to select the specific geometry on your model where you want the toolpath to be applied. Mastercam provides tools for selecting geometry easily, such as surface edges, loops, faces, or entire models.



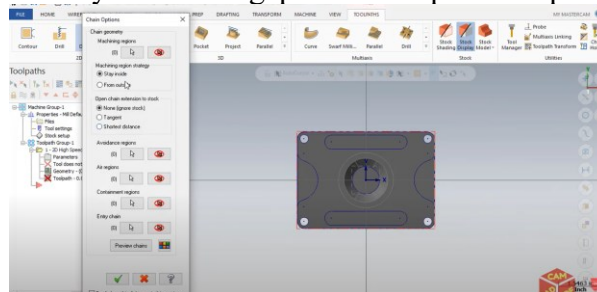
2.3 Tool Path Sequence/Continuity

In Mastercam, toolpath sequence and continuity are crucial aspects of machining operations to ensure efficient and accurate production. Here's how you can manage toolpath sequence and continuity in Mastercam:

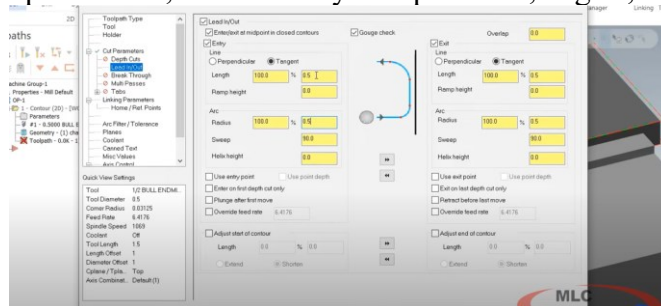
- a. **Order of operations:** Mastercam allows you to define the sequence of machining operations for your part. You can specify the order in which different toolpaths are generated and executed. Typically, roughing operations come before finishing operations to remove material quickly before the final pass.



- b. **Chaining and geometry:** Chaining refers to selecting the geometry on which the toolpath will be applied. Mastercam provides tools to efficiently chain geometry, such as selecting entire surfaces, loops, edges, or specific regions. Ensuring proper chaining helps maintain continuity and avoid gaps or overlaps in toolpaths.



- c. **Lead-in/Lead-out:** Lead-in and lead-out moves are essential for smooth transitions between toolpath segments and to prevent sudden changes in cutting direction that could lead to tool marks or material damage. Mastercam offers options to customize lead-in/lead-out parameters, such as entry/exit positions, angles, and lengths.



- d. **Transitions:** Smooth transitions between toolpath segments are critical for maintaining surface finish quality and reducing machining time. Mastercam provides options for controlling transitions, such as blending between toolpath segments with arcs or linear moves.

2.4 Suitable Tool Diameter and Length

Selecting suitable tool diameter and length in Mastercam depends on various factors such as the geometry of the part, material being machined, machining operation, available machinery, and desired surface finish. Here's how you can determine the appropriate tool diameter and length:

a. Tool diameter

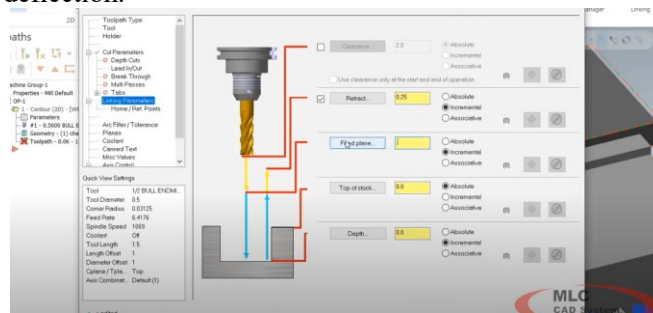
Feature Size: Choose a tool diameter that is suitable for the features you need to machine. For roughing operations, larger tool diameters can remove material quickly, while smaller diameters are suitable for finishing and detail work.

Cutter Engagement: Consider the cutter engagement during machining. Smaller diameter tools allow for more precise control and are often preferred for intricate details, while larger diameter tools are better suited for heavy material removal.

b. Tool Length:

Reach: Choose a tool length that provides sufficient reach to access all necessary features of the part. Consider the depth of cavities, pockets, or holes that need to be machined.

Stability: Longer tools may exhibit more deflection during machining, especially when performing deep cuts or in materials with poor rigidity. Use shorter tools or appropriate tool holders to maintain stability and minimize deflection.



2.5 Pre-Machining Operation

Pre-machining operations such as facing and side milling are essential in Mastercam for several reasons:

Face Milling






- This is a milling operation where a milling cutter (typically with multiple cutting edges) is used to remove material from the face of a workpiece. It's commonly used to create flat surfaces.

Side Milling

- Side milling involves cutting with the side of a milling cutter to remove material from the side of a workpiece. This operation can be used to create slots, pockets, and other features.

2.6 Work Holding and Clamping

In CAM operations, work holding and clamping are crucial for securing the workpiece in place during machining. Different methods and devices can be used depending on the size, shape, and material of the workpiece, as well as the specific machining operation being performed. Here are some common work holding and clamping devices used in CAM operations:

<p>Vice</p> <p>A vice is a mechanical device used to hold and secure workpieces during machining operations. It typically consists of two jaws that can be tightened together to grip the workpiece firmly in place. Vices are versatile and widely used for holding various types of workpieces in milling, drilling, and other machining processes.</p>	
<p>Chalk</p> <p>Chalk is a temporary adhesive substance that can be used to secure workpieces to the machine table or fixture. It provides a non-permanent holding method that allows for easy removal and repositioning of the workpiece between machining operations. Chalk is often used in conjunction with other clamping methods for added stability.</p>	
<p>Clamping kit set</p> <p>A clamping kit set consists of various clamping components such as T-nuts, studs, bolts, clamps, and straps that are used to secure workpieces to the machine table or fixture. Clamping kits offer flexibility and adaptability, allowing machinists to create custom clamping setups tailored to the specific requirements of their workpieces.</p>	
<p>Parallel bar</p> <p>Parallel bars are precision machined bars with flat, parallel surfaces that are used to support and align workpieces during machining operations. They help ensure that the workpiece remains level and parallel to the machine table, minimizing the risk of distortion or misalignment during machining.</p>	
<p>Angle plate</p> <p>An angle plate is a versatile work holding device with one or more precision machined surfaces at a fixed angle (typically 90 degrees). It is used to support and position workpieces at specific angles relative to the machine table or cutting tool. Angle plates are commonly used for machining operations that require angular or inclined surfaces.</p>	
<p>These works holding and clamping devices are essential components of CAM operations, providing stability, precision, and flexibility in securing workpieces for machining. Machinists must carefully select the appropriate clamping method and devices based on the specific requirements of their machining tasks to ensure safe and efficient operation.:</p>	

2.7 Cutting Parameters

Cutting parameters are essential factors that determine the efficiency, quality, and safety of machining operations. Here are the listed cutting parameters and their explanations:

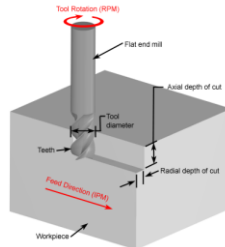
A. Feed

Feed refers to the rate at which the cutting tool advances along the workpiece surface during machining. It is usually expressed in units of distance per revolution (e.g., inches per revolution or millimeters per revolution). A higher feed rate results in faster material removal but may require more power and produce higher cutting forces.

B. RPM (Revolutions Per Minute)

RPM is the rotational speed of the cutting tool, typically measured in revolutions per minute. It determines how fast the cutting edge of the tool moves past the workpiece surface. The RPM is calculated based on the desired cutting speed, tool diameter, and material being machined. Proper RPM selection is crucial for achieving the desired cutting performance and tool life.

C. Depth of cut

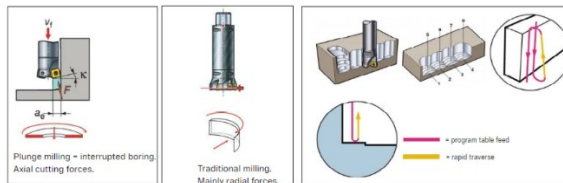
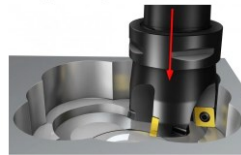


Depth of cut refers to the distance that the cutting tool penetrates into the workpiece during each pass. It is typically measured perpendicular to the workpiece surface and is expressed in units of length (e.g., inches or millimeters). The depth of cut affects material removal rates, cutting forces, and chip formation. It is important to select an appropriate depth of cut based on the material properties, tool geometry, and machining conditions.

D. Plunge

Plunge refers to the vertical movement of the cutting tool into the workpiece at the beginning of a machining operation. It is commonly used for drilling, boring, and other operations where the tool enters the workpiece directly without a horizontal feed motion. Plunge feed rates and depths are carefully controlled to prevent tool breakage, minimize chatter, and ensure accurate hole geometry.

Plunge milling



E. Cutting process

Plunge milling operations vary considerably from traditional milling. It uses the end of the tool to cut instead of the periphery, which beneficially changes the direction of the cutting forces from predominantly radial to axial. It can be compared to a boring operation with interrupted cuts.

Power consumption and noise are low.

Self-Check Sheet 2: Identify the Sequence of Tool Path and Machining Strategy

1. What is CNC Milling Machine?

Answer:

2. What is CNC Router?

Answer:

3. What is CNC Waterjet Cutter?

Answer:

4. What is Tool and Toolpath Parameters of master cam?

Answer:

5. What is Simulation of master cam?

Answer:

6. What is post-processing of master cam?

Answer:

7. What is Order of Operations of master cam?

Answer:

8. Why need Chaining and Geometry?

Answer:

9. What is Lead-in/Lead-out?

Answer:

10. Why need Simulation and Verification before Code Generate?

Answer:

Answer Sheet 2 Identify the Sequence of Tool Path and Machining Strategy

1. What is CNC Milling Machine?

Answer: CNC milling machines are used to remove material from a workpiece by rotating a cutting tool against it. Mastercam can generate toolpaths for various milling operations, including 2D milling, 3D milling, pocketing, contouring, and engraving. Different configurations of CNC milling machines, such as vertical mills, horizontal mills, and multi-axis mills, can be used depending on the complexity and size of the parts being machined.

2. What is CNC Router?

Answer: CNC routers are used for cutting and shaping various materials, including wood, plastic, foam, and composites. Mastercam can generate toolpaths for routing operations such as profile cutting, pocketing, drilling, and carving.

3. What is CNC Waterjet Cutter?

Answer: CNC waterjet cutters use a high-pressure jet of water mixed with abrasive particles to cut a wide range of materials, including metal, stone, glass, and composites.

4. What is Tool and Toolpath Parameters of master cam?

Answer: Configure the tool and toolpath parameters according to your machining requirements. This includes specifying the cutting tool, feeds, and speeds, stepovers, step downs, and other parameters that affect how the tool interacts with the material.

5. What is Simulation of master cam?

Answer: Before executing the machining operation on an actual machine, it's essential to simulate the toolpath to ensure that it behaves as expected and doesn't result in collisions or other issues.

6. What is post-processing of master cam?

Answer: Once you are satisfied with your toolpaths, you'll need to generate G-code using a post-processor that's specific to your CNC machine. Mastercam includes a library of post-processors for various machine types, but you may need to customize or create a new post-processor if none are available for your machine.

7. What is Order of Operations of master cam?

Answer: Mastercam allows you to define the sequence of machining operations for your part. You can specify the order in which different toolpaths are generated and executed. Typically, roughing operations come before finishing operations to remove material quickly before the final pass.

8. Why need Chaining and Geometry?

Answer: Chaining refers to selecting the geometry on which the toolpath will be applied. Mastercam provides tools to efficiently chain geometry, such as selecting entire surfaces, loops, edges, or specific regions. Ensuring proper chaining helps maintain continuity and avoid gaps or overlaps in toolpaths.

9. What is Lead-in/Lead-out?

Answer: Lead-in and lead-out moves are essential for smooth transitions between toolpath segments and to prevent sudden changes in cutting direction that could lead to tool marks or material damage. Mastercam offers options to customize lead-in/lead-out parameters, such as entry/exit positions, angles, and lengths.

10. Why need Simulation and Verification before Code Generate?

Answer: Before sending toolpaths to the CNC machine, it's essential to simulate and verify them using Mastercam's built-in simulation tools. This helps identify any issues with toolpath sequence, continuity, or collision avoidance before actual machining begins.

Job Sheet 2.1: Select Pre-Machining Operations as Required

Job Name: Select pre-machining operations as required

Procedure:

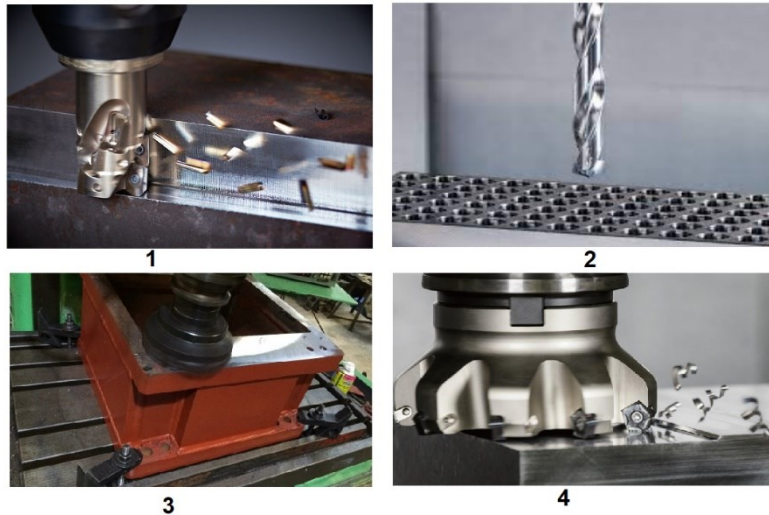
1. Collect and wear the PPE.
2. Read job sheet and specification sheet.
3. Choose the appropriate material based on the requirements of the final product and its properties.
4. Inspect the material for any defects, such as cracks, inclusions, or surface imperfections.
5. Cut the material into the desired size and shape using processes.
6. Cut the workpiece into a rough shape, leaving excess material for subsequent machining operations.
7. Remove any burrs, sharp edges, or imperfections from the edges of the workpiece to ensure safety and improve the quality of the final product.
8. Clean the surface of the workpiece to remove any contaminants such as dirt, oil, or rust that could interfere with the machining process or affect the surface finish of the final product.
9. Select Machine a flat surface on the workpiece to provide a reference point for subsequent machining operations.
10. Ensure that the workpiece is properly centered and aligned on the machine tool to prevent errors and inaccuracies during machining.
11. Mark and select reference points, dimensions, and machining instructions on the workpiece to guide the machining process and ensure accuracy.
12. Prepare fixtures, jigs, or clamps to securely hold the workpiece in place during machining.
13. Select Drill pilot holes or clearance holes as needed to facilitate subsequent drilling or machining operations.

SI No	Name of pre-Operation
1.	Side Milling
2.	Point drilling
3.	Surface fishing
4.	Facing

Specification Sheet 2.1: Select Pre-Machining Operations as Required

Name of Job: Select pre-machining operations as required

Conditions for the job: You must always practice safe operation procedures during the assessment. You will be required to demonstrate OSH competencies to Select pre-machining operations as required



Required Personal Protective Equipment (PPE): Pre/Person

Sl.	Name of the PPE	Specification	Unit	Quantity	Remarks
11	Hand Gloves	Cotton	pair	01	
12	Mask	N 95	Nos	01	
13	Safety shoe	As required	pair	01	
14	Safety goggles	Standard	Nos	01	
15	Apron	Standard	Nos	01	

Required Tools and equipment

Sl.	Name of tools and equipment	Specification	Unit	Quantity	Remarks
5	CNC Milling Machine		No	01	

Job Sheet 2.2: Prepare Work Holding and Clamping to Avoid Tool Clash

Job Name: Prepare work holding and clamping to avoid tool clash

Procedure:

1. Open Mastercam software on your computer.
2. Import CAD Model it into Mastercam using the appropriate file format (e.g., STEP, IGES, or native CAD format).
3. Set up the workpiece in Mastercam by defining the stock material size and origin.
4. Use Mastercam's CAD/CAM tools to define the toolpaths required for machining the workpiece. This includes operations such as roughing, finishing, drilling, and profiling.
5. Perform simulation feature to visualize the toolpaths and simulate the machining process.
6. identify any potential collisions or tool clashes with the workpiece or clamping devices.
7. Choose the appropriate work holding devices within Mastercam's tool library or create custom fixtures as needed for securing the workpiece during machining.
8. Position the work holding devices in the virtual machining environment within Mastercam, ensuring they are aligned with the workpiece and provide secure clamping without interfering with the toolpaths.
9. Define clearance regions within Mastercam to specify areas where the tool and tool holder should avoid, such as clamps, fixtures, or other obstructions.
10. Use collision detection feature to check for collisions between the tool, tool holder, workpiece, and work holding devices. Adjust the setup as needed to avoid collisions and tool clashes.
11. Modify the toolpaths as necessary to optimize the machining process and avoid potential collisions or tool clashes, such as adjusting entry and exit points or adding additional clearance moves.
12. Conduct a final simulation of the modified toolpaths to verify that all collisions and tool clashes have been avoided.
13. setup is finalized and collisions are eliminated, generate the G-code for the CNC machine based on the optimized toolpaths.
14. By following these steps within Mastercam, you can effectively set up work holding

Specification Sheet 2.2: Job Name: Prepare Work Holding and Clamping to Avoid Tool Clash

Name of Job: Prepare work holding and clamping to avoid tool clash

Conditions for the job: You must practice safe operation procedures at all times during the assessment. You will be required to demonstrate OSH competencies to Job Name: Prepare work holding and clamping to avoid tool clash



Required Personal Protective Equipment (PPE): Pre/person

Sl.	Name of the PPE	Specification	Unit	Quantity	Remarks
16	Hand Gloves	Cotton	pair	01	
17	Mask	N 95	Nos	01	
18	Safety shoe	As required	pair	01	
19	Safety goggles	Standard	Nos	01	
20	Apron	Standard	Nos	01	

Required Tools and Equipment

Sl.	Name of tools and equipment	Specification	Unit	Quantity	Remarks
6	CNC Milling Machine		No	01	

Learning Outcome -3: Create 2D Tool Paths

Learning Outcome -3: Create 2D tool paths	
Assessment Criteria	<ol style="list-style-type: none"> 1. Planes are identified 2. Machine definition and post-processor is selected 3. Origin & stock setup is defined 4. 2D tool paths are executed 5. Tapping and thread milling is executed 6. Tool paths are verified 7. G&M codes are generated
Conditions and Resources	<ol style="list-style-type: none"> 1. Workplace or Simulated Workplace 2. CBLM 3. Handout 4. Laptop 5. Multimedia Projector 6. Paper, Pen, Pencil, 7. Internet Facilities 8. White Board and 9. Audio Video Devices 10. Necessary Tools and Equipment 11. Necessary Materials 12. Necessary PPE
Contents	<ol style="list-style-type: none"> 1. Plan <ul style="list-style-type: none"> ▪ Construction Plan ▪ WCS Plan ▪ Tool plan 2. Machine definition and post-processor 3. Original and stock setup 4. 2D tool path <ul style="list-style-type: none"> ▪ Facing ▪ Cutting style ▪ Tip compensation ▪ Max stepover

	<ul style="list-style-type: none"> ▪ Across overlap ▪ Along overlap ▪ Approach Distance ▪ Exit distance ▪ Climb ▪ Conventional ▪ Contour ▪ Pocket ▪ Machining direction ▪ Pocket type ▪ Tip compensation ▪ Stock to lib ▪ Lead in/out ▪ Cutting method ▪ Entry Motion ▪ Finishing ▪ Drilling ▪ Point/Arc Geometry ▪ Gross ▪ Tip compensation ▪ Spindle direction ▪ Feed speed ▪ Dynamic ▪ Toolpath type ▪ Cutting method ▪ Max stepover ▪ Stock to Live ▪ Lead in/out ▪ Machining region ▪ Avoidance region ▪ Air region ▪ Containment boundary <p>5. Tapping and thread milling</p> <ul style="list-style-type: none"> ▪ Pitch ▪ Thread angle ▪ Thread depth ▪ Major and minor diameter ▪ Internal and external threads ▪ Right and left-handed threads <p>6. Requirement to verify tool path</p>
Job/Task/Activity	<ol style="list-style-type: none"> 1. Identify the plan 2. Select Machine Definition and Post-Processor 3. Execute the 2D tool path

	<ol style="list-style-type: none"> 4. Execute tapping and thread milling 5. Verify the tool path
Training Method	<ol style="list-style-type: none"> 1. Discussion 2. Presentation 3. Demonstration 4. Guided Practice 5. Individual Practice 6. Project Work 7. Problem Solving 8. Brainstorming
Assessment Method	<ol style="list-style-type: none"> 1. Written Test 2. Demonstration 3. Oral questioning 4. Portfolio

Learning Experience 3: Create 2D Tool Paths

In order to achieve the objectives stated in this learning guide, you must perform the learning steps below. Beside each step are the resources or special instructions you will use to accomplish the corresponding activity.

Learning Activities	Recourses/Special Instructions
1. Trainee will ask the instructor about the learning materials	1. Instructor will provide the learning materials ‘Apply CAM Software for Machining
2. Read the Information sheet and complete the Self Checks & Check answer sheets on “ Create 2D tool paths ”	2. Read Information sheet 1: Create 2D tool paths 3. Answer Self-check 1: Create 2D tool paths 4. Check your answer with Answer key 1: Create 2D tool paths
3. Read the Job/Task Sheet and Specification Sheet and perform job/Task	5. Job/Task Sheet and Specification Sheet Job sheet 3: Create a 2D drill toolpath that drills all four holes of the part Activity 3.1: Configure Machine and Setting Up Stock for drilling Activity 3.2: Create a drill toolpath that drills all four holes of the part. Activity 3.3: Creating a Dynamic Mill Toolpath Specification sheet 3: Create a 2D drill toolpath that drills all four holes of the part

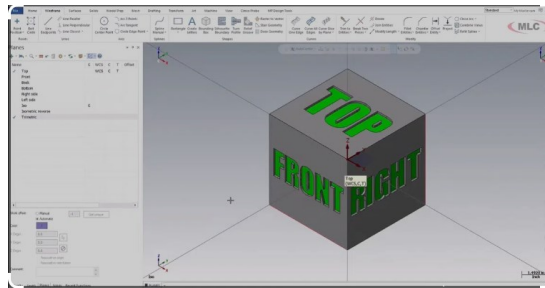
Information Sheet 3: Create 2D Tool Paths

Learning Objective: After completion of this information sheet, the learners will be able to explain, define and interpret the following contents:

- 3.1 Plan
- 3.2 Machine definition and post-processor
- 3.3 Original and stock setup
- 3.4 2D tool path
- 3.5 Tapping and thread milling
- 3.6 Requirement to verify tool path

3.1 Plan

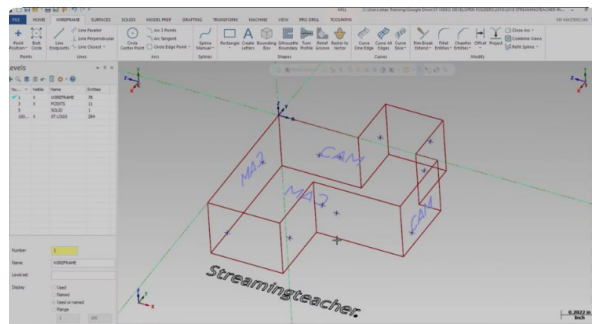
Mastercam, planning is a crucial step in the machining process that involves creating detailed plans and strategies for various aspects of the machining operation. Here are the listed plans and their explanations:



Plans in Mastercam:

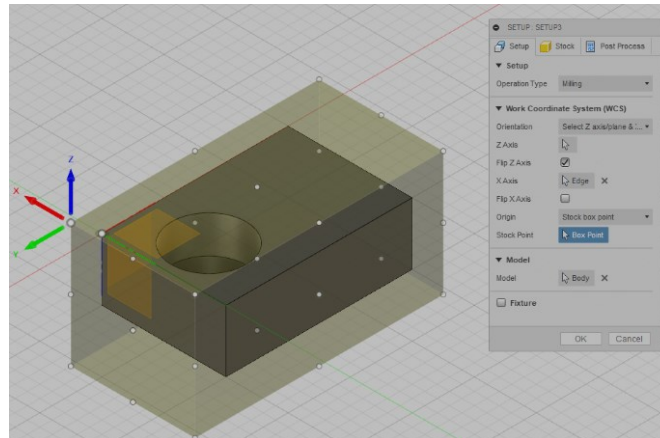
a. Construction plan

The Construction Plan in Mastercam involves creating a plan for the construction of the part or component to be machined. This includes defining the overall geometry of the part, creating sketches, defining dimensions, and specifying features such as holes, pockets, and surfaces. The Construction Plan serves as the foundation for creating toolpaths and generating G-code for machining the part.



b. WCS (work coordinate system) plan

The WCS Plan in Mastercam involves defining the work coordinate system that specifies the orientation and position of the part relative to the machine tool. This includes setting up the origin point, selecting reference planes, and defining the axis directions. The WCS Plan ensures that the machining operations are performed accurately and consistently based on the defined coordinate system.



c. Tool plan

The Tool Plan in Mastercam involves selecting and planning the tools to be used for machining the part. This includes choosing the appropriate cutting tools, specifying tool parameters such as diameter, length, and geometry, and defining toolpaths for various machining operations. The Tool Plan ensures that the right tools are selected and optimized for the specific requirements of the part, material, and machining process.

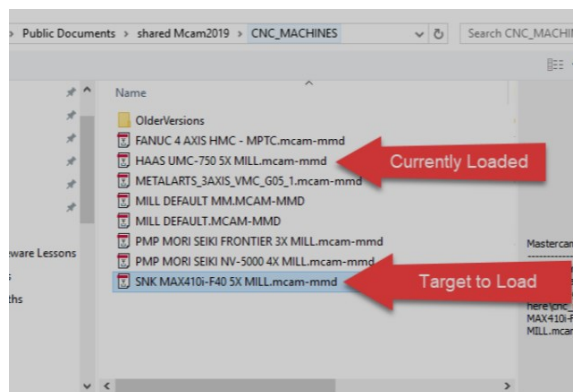
These plans in Mastercam are essential for effective planning and preparation of machining operations. They help ensure that the part is accurately defined, the workpiece is properly oriented and positioned, and the correct tools and strategies are selected for machining. By carefully creating and implementing these plans, machinists can optimize the machining process, improve efficiency, and achieve high-quality results.

3.2 Machine Definition and Post-Processor

A. Machine definition

Machine definition in Mastercam involves specifying the characteristics and capabilities of the CNC machine that will be used to manufacture the part. This includes:

- a. **Machine type:** Selecting the type of CNC machine, such as milling machine, lathe, router, or wire EDM machine.
- b. **Machine Configuration:** Specifying the machine's configuration, including the number of axes (e.g., 3-axis, 4-axis, 5-axis), spindle types (e.g., vertical or horizontal), and any additional features or accessories (e.g., rotary tables, probing systems).



- c. **Machine kinematics:** Defining the machine's kinematic configuration, including axis orientations, travel limits, and axis motion types (e.g., linear, rotary, simultaneous).

- d. **Tool Library:** Setting up a library of cutting tools with detailed information such as tool type, geometry, dimensions, and cutting parameters. This allows Mastercam to select the appropriate tools for machining operations based on the defined machine setup.
- e. **Work coordinate system (WCS):** Establishing the work coordinate system for the machine, which defines the origin point and orientation of the part relative to the machine's axes. This ensures that toolpaths are generated accurately and consistently.
- f. **Machine limits:** Defining the physical limits of the machine, including maximum feed rates, spindle speeds, and travel distances. This helps prevent toolpath errors and collisions during machining.

B. Post-Processor Configuration

Post-processor configuration in Mastercam involves selecting or customizing a post-processor that translates toolpaths generated in Mastercam into machine-specific G-code instructions that the CNC machine can understand. This includes:

- a. **Post-processor selection:** Choosing a pre-configured post-processor from Mastercam's library that matches the make and model of the CNC machine. Mastercam provides a wide range of post-processors for popular CNC machines, but custom post-processors may be required for specific machine configurations.
- b. **Post-processor customization:** Modifying and customizing the selected post-processor to accommodate unique machine configurations, preferences, or specific requirements. This may involve adjusting G-code formatting, adding custom commands, or fine-tuning parameters to optimize machine performance.
- c. **Simulation and verification:** Testing the post-processor with simulated toolpaths to ensure that the generated G-code is compatible with the CNC machine and produces the desired machining results. This helps identify and resolve any issues or discrepancies before running the program on the actual machine.

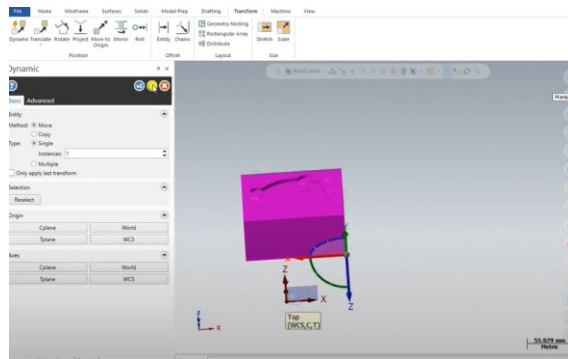
By accurately defining the machine characteristics and configuring the post-processor in Mastercam, machinists can ensure seamless communication between the CAM software and CNC machine, leading to efficient and accurate machining operations.

3.3 Origin and Stock Setup

In Mastercam, setting up the origin and stock is a fundamental step in preparing the CAM environment for generating toolpaths and machining operations. Here's an overview of how to set up the origin and stock in Mastercam:

A. Origin Setup:

- a. **Select machine coordinate system (MCS):** Determine the machine coordinate system (MCS) that defines the global reference frame for the machining operations. The MCS is typically aligned with the physical axes of the CNC machine.
- b. **Define work coordinate system (WCS):** Establish the work coordinate system (WCS) that specifies the origin and orientation of the part relative to the machine's axes. The WCS defines the zero point for machining operations and determines the part's position and orientation on the machine table.

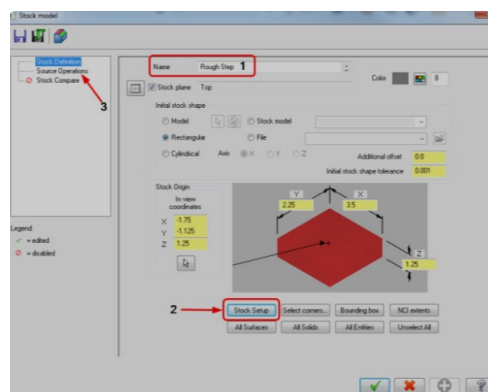


- c. **Set origin point:** Define the origin point within the WCS where the machining operations will begin. This can be set manually by specifying coordinates or by selecting a reference point on the part geometry.
- d. **Choose axis directions:** Determine the positive directions of the X, Y, and Z axes within the WCS. This ensures that toolpaths are generated correctly and that the tool moves in the desired direction during machining operations.
- e. **Verify setup:** Review and verify the origin setup to ensure that the WCS origin, axis directions, and MCS alignment are correct. This helps prevent errors and ensures that toolpaths are generated accurately.

B. Stock Setup

Define Stock Material: Specify the material type and dimensions of the raw stock from which the part will be machined. This includes parameters such as stock shape (e.g., block, cylinder), dimensions (length, width, height), and material properties (e.g., material type, density).

- a. **Select stock model:** Choose a stock model representation that accurately reflects the physical stock geometry. Mastercam provides options for simple geometric shapes (e.g., box, cylinder) as well as more complex stock models imported from CAD files.



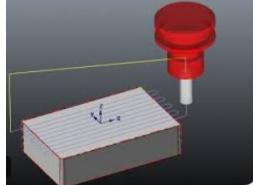
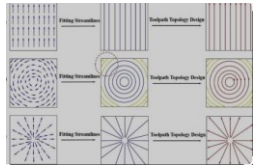
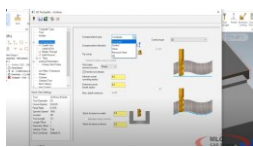
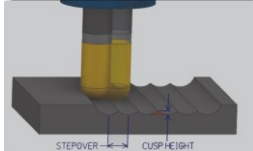
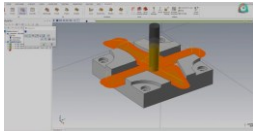
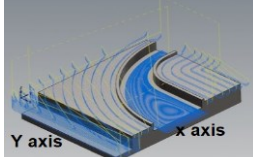
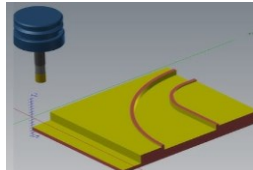
- b. **Position Stock Relative to Origin:** Position the stock model relative to the origin and WCS to align it with the desired machining setup. This ensures that toolpaths are generated correctly and that the tool engages with the stock material as intended.
- c. **Stock allowances:** Specify any additional material allowances or offsets to account for machining tolerances, finishing requirements, or stock material variations. This helps ensure that the final part dimensions meet the design specifications.

By accurately setting up the origin and stock in Mastercam, machinists can establish

a solid foundation for generating toolpaths and executing machining operations. This ensures that toolpaths are generated correctly, material is removed efficiently, and the final parts meet the desired dimensional accuracy and quality standards.

3.4 2D Tool Path

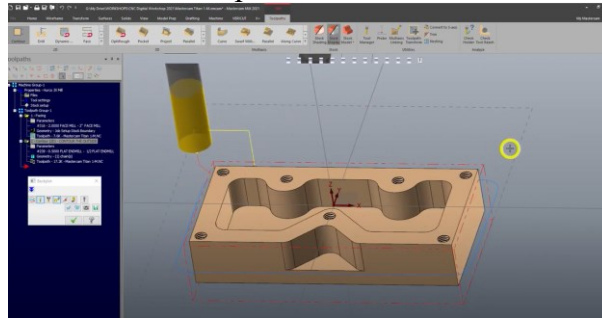
In Mastercam, 2D toolpaths are used to machine parts or features that are primarily defined in two dimensions, such as pockets, slots, contours, and profiles. Here is an explanation of the listed parameters commonly found in 2D toolpath operations

<p>Facing Facing is a 2D toolpath operation used to machine flat surfaces on the top of the workpiece to ensure they are smooth and perpendicular to the machine's spindle axis.</p>	
<p>Cutting style Cutting style refers to the direction in which the tool cuts material during machining. It can be set to either "Climb" or "Conventional" milling. In climb milling, the tool rotates in the same direction as the feed motion, resulting in reduced cutting forces and smoother cuts. In conventional milling, the tool rotates opposite to the feed motion, which can lead to higher cutting forces and more aggressive cutting action</p>	
<p>Tip compensation Tip compensation adjusts the toolpath to account for the diameter of the cutting tool. It ensures that the tool cuts along the intended path, taking into consideration the tool's geometry and avoiding deviations from the desired dimensions.</p>	
<p>Max stepover Max stepover defines the maximum distance the tool moves sideways between successive passes. It controls the width of cuts and helps prevent excessive tool wear, deflection, and chatter by limiting the amount of material removed in each pass.</p>	
<p>Across overlap Across overlap specifies the amount of overlap between adjacent toolpath passes in the X-axis direction. It ensures that adjacent passes blend smoothly and prevents scalloping or uneven surfaces caused by abrupt transitions between passes.</p>	
<p>Along overlap Along overlap specifies the amount of overlap between adjacent toolpath passes in the Y-axis direction. Similar to across overlap, along overlap ensures smooth transitions between passes and helps achieve a consistent surface finish</p>	
<p>Approach distance Approach distance determines the distance the tool travels before engaging with the workpiece at the beginning of the toolpath. It provides a lead-in move to gradually introduce the cutting tool to the workpiece, reducing the risk of tool marks or damage.</p>	
<p>Exit distance Exit distance specifies the distance the tool travels after completing the cutting motion</p>	

at the end of the toolpath. It provides a lead-out move to smoothly retract the tool from the workpiece, minimizing the risk of tool marks or burrs.

A. Contour

In Mastercam, both Contour and Pocket are commonly used 2D toolpath operations for machining features such as profiles, pockets, and slots. Here's an explanation of the parameters associated with each operation:



a. Machining direction

Specifies the direction in which the toolpath is generated along the contour geometry. It can be set to climb or conventional milling based on the desired cutting style.

b. Tip compensation

Adjusts the toolpath to account for the diameter of the cutting tool, ensuring that the tool follows the desired path accurately while taking into consideration the tool geometry.

c. Stock to leave (stock to lib)

Specifies the amount of material to leave on the part surface after machining. It allows for a final pass to remove any remaining stock material to achieve the desired part dimensions.

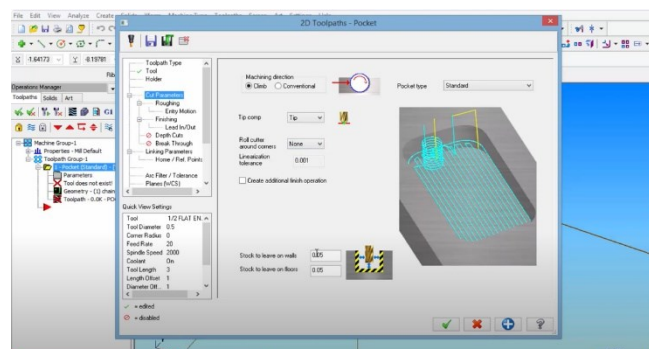
d. Lead in/out

Defines the approach and retract motions of the cutting tool at the beginning and end of the toolpath. Lead-in/out moves help minimize tool marks, reduce the risk of part damage, and ensure smooth transitions into and out of the machining area.

B. Pocket

a. Machining direction

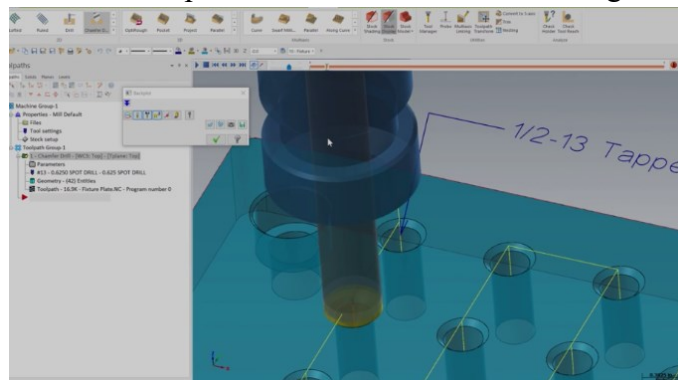
Specifies the direction in which the toolpath is generated along the contour geometry. It can be set to climb or conventional milling based on the desired cutting style.



- b. **Specifies the type of pocket geometry, such** as rectangular, circular, or custom-shaped. The selected pocket type determines the machining strategy and toolpath generation method.
- c. **Machining direction**
Determines the direction in which the toolpath is generated within the pocket geometry. It can be set to climb or conventional milling based on the desired cutting style.
- d. **Entry motion**
Defines the approach motion of the cutting tool as it enters the pocket area. Entry motions help minimize tool marks and ensure smooth transitions into the machining area.
- e. **Cutting method**
Specifies the cutting strategy to be used within the pocket, such as conventional pocketing, climb pocketing, or high-speed machining (HSM). Each cutting method has its advantages in terms of material removal rate, surface finish, and tool life.
- f. **Finishing**
Specifies whether the pocket operation includes a finishing pass to achieve the final surface quality and dimensional accuracy. Finishing passes are typically performed with smaller stepovers and reduced cutting depths to achieve smoother surfaces. These parameters allow machinists to customize contour and pocket toolpaths in Mastercam to meet specific machining requirements, including cutting style, toolpath generation method, stock material considerations, and surface finish requirements. Adjusting these parameters appropriately ensures efficient, accurate, and high-quality machining results.

C. Drilling

In Mastercam, the drilling operation is used to create holes or bores in the workpiece. Here's an explanation of the parameters associated with drilling:



- a. **Point/arc geometry**
Specifies the geometry of the hole, such as point or arc. Point geometry creates simple holes with a defined center point and diameter. Arc geometry allows for more complex hole shapes, including counterbores or countersinks, by defining additional parameters such as start and end angles and radii.
- b. **Tip compensation**
Tip compensation adjusts the toolpath to account for the diameter of the cutting tool, ensuring that the tool follows the desired path accurately while considering the tool geometry. It ensures that the hole is drilled with the correct diameter and position.
- c. **Spindle direction**

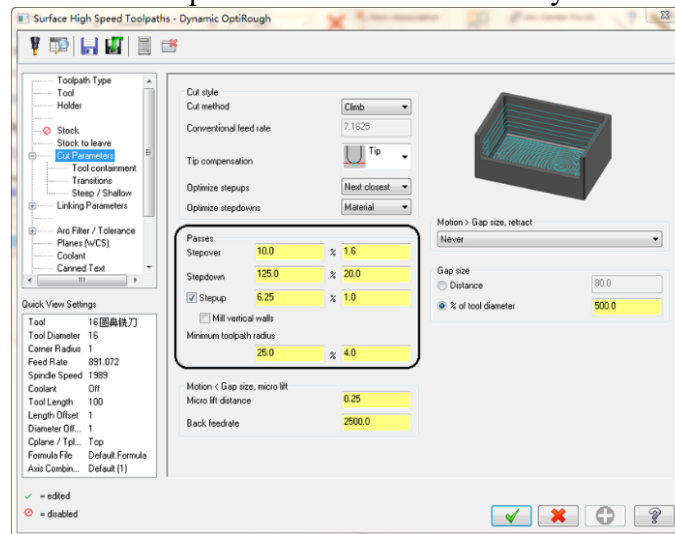
Specifies the direction of spindle rotation during the drilling operation. It can be set to either clockwise (CW) or counterclockwise (CCW) rotation, depending on the type of drill bit being used and the material being drilled.

d. Feed speed

Feed speed refers to the rate at which the cutting tool advances into the workpiece during drilling. It is typically specified in units of distance per revolution (e.g., inches per revolution or millimeters per revolution) and determines the speed at which material is removed from the workpiece.

D. Dynamic Toolpath

In Mastercam, dynamic toolpaths are advanced machining strategies that dynamically adjust cutting parameters based on the geometry of the part and the cutting conditions. Here's an explanation of the parameters associated with dynamic toolpaths:



a. Toolpath type

Specifies the type of dynamic toolpath strategy to be used, such as Dynamic Milling, Dynamic Contour, or Dynamic Opti Rough. Each toolpath type has its unique approach to optimizing tool engagement and material removal rates based on the part geometry and cutting conditions.

b. Cutting method

Defines the cutting strategy to be used within the dynamic toolpath, such as climb milling or conventional milling. The cutting method determines the direction in which the tool cuts material and can affect cutting forces, chip evacuation, and surface finish.

c. Max stepover

Specifies the maximum distance the tool moves sideways between successive passes during dynamic machining. It controls the width of cuts and helps prevent excessive tool wear, deflection, and chatter by limiting the amount of material removed in each pass.

d. Stock to leave (stock to live)

Determines the amount of material left on the part surface after dynamic machining. It allows for a final pass to remove any remaining stock material to achieve the desired part dimensions.

e. Lead In/Out

DEFINES the approach and retract motions of the cutting tool at the beginning and end of the dynamic toolpath. Lead-in/out moves help minimize tool marks, reduce the risk of part damage, and ensure smooth transitions into and out of the machining area.

f. Machining region

Specifies the region of the part where dynamic machining will be performed. It defines the boundaries within which the toolpaths are generated, ensuring that only the specified areas of the part are machined.

g. Avoidance region

Defines areas on the part where the cutting tool should avoid during machining. Avoidance regions can be used to protect certain features or areas of the part from being machined, such as holes, pockets, or critical surfaces.

h. Air region

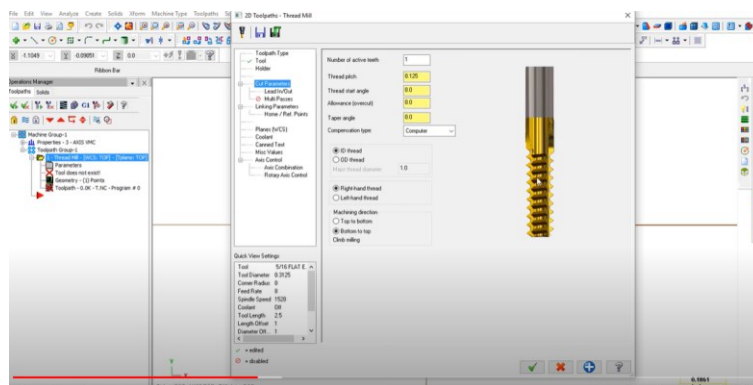
Specifies areas where the cutting tool is allowed to move without engaging with the workpiece. Air regions are typically used to optimize toolpath efficiency by minimizing unnecessary tool engagement and reducing machining time.

i. Containment boundary

Defines the outer boundary or perimeter of the machining area for dynamic toolpaths. It ensures that the toolpaths are contained within the specified region and do not extend beyond the desired machining limits.

3.5 Tapping And Thread Milling

In Mastercam, tapping and thread milling are machining operations used to create threaded features in a workpiece. Here are the parameters associated with each operation:



a. Pitch

Specifies the distance between adjacent threads in the threaded feature. It determines the thread density and is typically measured in units of length per thread (e.g., inches per thread or millimeters per thread).

b. Thread angle

Defines the angle of the threads relative to the axis of the threaded feature. The thread angle depends on the type of thread profile being used (e.g., V-thread, Acme thread) and is typically standardized for specific thread standards (e.g., ANSI, ISO).

c. Thread depth

Specifies the depth to which the threads are cut into the workpiece material. It determines the effective length of engagement between mating threaded components and is typically measured from the crest of the thread to the root.

d. Major and minor diameter

The major diameter represents the outer diameter of the threaded feature, measured at the peaks (crests) of the threads. The minor diameter represents the inner diameter of the threaded feature, measured at the valleys (roots) of the threads.

e. Internal and external threads

Internal threads are threads cut into a hole or recess in the workpiece, while external threads are threads cut onto the outer surface of a cylindrical or conical feature. Mastercam allows for the creation of both internal and external threads.

f. Right and left-handed threads

Right-handed threads are threads that follow a clockwise direction when viewed from the end of the threaded feature, while left-handed threads follow a counterclockwise direction. Mastercam supports the creation of both right and left-handed threads.

3.6 Requirements To Verify Toolpath

To verify toolpaths generated in Mastercam, machinists typically perform the following steps:

- a. Simulation:** Use Mastercam's simulation feature to visualize the toolpath and machining operations in a virtual environment. This allows machinists to identify any potential issues, such as collisions, gouges, or improper toolpath motions, before running the program on the actual machine.
- b. Collision detection:** Check for collisions between the cutting tool, tool holder, and the workpiece geometry. Mastercam's collision detection feature helps identify potential collisions and allows machinists to adjust toolpath parameters to avoid interference.
- c. Material removal simulation:** Verify the material removal process by simulating the cutting tool's engagement with the workpiece material. This helps ensure that the toolpath effectively removes material and produces the desired features with the correct dimensions and surface finish.
- d. Toolpath analysis:** Analyze the toolpath for efficiency, tool engagement, and cutting conditions. Mastercam provides tools for analyzing toolpath parameters such as feed rates, cutting speeds, stepovers, and toolpath optimization strategies to ensure optimal machining performance.

By verifying toolpaths in Mastercam, machinists can identify and address any potential issues or errors before machining the part, reducing the risk of scrap, rework, and machine downtime.

Self-Check Sheet 3.1 Create 2D Tool Paths

1. What is Construction Plan:

Answer:

2. What is WCS (Work Coordinate System) Plan?

Answer:

3. What is Tool Plan:

Answer:

4. What is Stock Material:

Answer:

5. What is Facing:

Answer:

6. What is Tip Compensation:

Answer:

7. What is Max Stepover:

Answer:

8. What is Across Overlap:

Answer:

9. What is Along Overlap:

Answer:

10. What is Approach Distance:

Answer:

11. What is Exit Distance:

Answer:

12. What is Pocket Entry Motion?

Answer:

13. What is Cutting Method for pocketing?

Answer:

14. What is Dynamic Toolpath:

Answer:

15. What is Lead In/Out?

Answer:

16. What is Pitch:

Answer:

17. What is Thread Angle:

Answer:

18. What is Thread Depth:

Answer:

Answer Sheet 3.1 Create 2D Tool Paths

1. What is Construction Plan:

Answer: The Construction Plan in Mastercam involves creating a plan for the construction of the part or component to be machined. This includes defining the overall geometry of the part, creating sketches, defining dimensions, and specifying features such as holes, pockets, and surfaces. The Construction Plan serves as the foundation for creating toolpaths and generating G-code for machining the part.

2. What is WCS (Work Coordinate System) Plan?

Answer: The WCS Plan in Mastercam involves defining the work coordinate system that specifies the orientation and position of the part relative to the machine tool. This includes setting up the origin point, selecting reference planes, and defining the axis directions. The WCS Plan ensures that the machining operations are performed accurately and consistently based on the defined coordinate system.

3. What is Tool Plan:

Answer: The Tool Plan in Mastercam involves selecting and planning the tools to be used for machining the part. This includes choosing the appropriate cutting tools, specifying tool parameters such as diameter, length, and geometry, and defining toolpaths for various machining operations. The Tool Plan ensures that the right tools are selected and optimized for the specific requirements of the part, material, and machining process.

4. What is Stock Material:

Answer: Specify the material type and dimensions of the raw stock from which the part will be machined. This includes parameters such as stock shape (e.g., block, cylinder), dimensions (length, width, height), and material properties (e.g., material type, density).

5. What is Facing:

Answer: Facing is a 2D toolpath operation used to machine flat surfaces on the top of the workpiece to ensure they are smooth and perpendicular to the machine's spindle axis.

6. What is Tip Compensation:

Answer: Tip compensation adjusts the toolpath to account for the diameter of the cutting tool. It ensures that the tool cuts along the intended path, taking into consideration the tool's geometry and avoiding deviations from the desired dimensions.

7. What is Max Stepover?

Answer: Max stepover defines the maximum distance the tool moves sideways between successive passes. It controls the width of cuts and helps prevent excessive tool wear, deflection, and chatter by limiting the amount of material removed in each pass.

8. What is Across Overlap:

Answer: Across overlap specifies the amount of overlap between adjacent toolpath passes in the X-axis direction. It ensures that adjacent passes blend smoothly and prevents scalloping or uneven surfaces caused by abrupt transitions between passes.

9. What is Along Overlap?

Answer: Along overlap specifies the amount of overlap between adjacent toolpath passes in the Y-axis direction. Similar to across overlap, along overlap ensures smooth transitions

between passes and helps achieve a consistent surface finish.

10. What is Approach Distance?

Answer: Approach distance determines the distance the tool travels before engaging with the workpiece at the beginning of the toolpath. It provides a lead-in move to gradually introduce the cutting tool to the workpiece, reducing the risk of tool marks or damage.

11. What is Exit Distance?

Answer: Exit distance specifies the distance the tool travels after completing the cutting motion at the end of the toolpath. It provides a lead-out move to smoothly retract the tool from the workpiece, minimizing the risk of tool marks or burrs.

12. What is Pocket Entry Motion?

Answer: Defines the approach motion of the cutting tool as it enters the pocket area. Entry motions help minimize tool marks and ensure smooth transitions into the machining area.

13. What is Cutting Method for pocketing?

Answer: Specifies the cutting strategy to be used within the pocket, such as conventional pocketing, climb pocketing, or high-speed machining (HSM). Each cutting method has its advantages in terms of material removal rate, surface finish, and tool life.

14. What is Dynamic Toolpath:

Answer: In Mastercam, dynamic toolpaths are advanced machining strategies that dynamically adjust cutting parameters based on the geometry of the part and the cutting conditions. Here's an explanation of the parameters associated with dynamic toolpaths.

15. What is Lead In/Out?

Answer: Defines the approach and retract motions of the cutting tool at the beginning and end of the dynamic toolpath. Lead-in/out moves help minimize tool marks, reduce the risk of part damage, and ensure smooth transitions into and out of the machining area.

16. What is Pitch?

Answer: Specifies the distance between adjacent threads in the threaded feature. It determines the thread density and is typically measured in units of length per thread (e.g., inches per thread or millimeters per thread).

17. What is Thread Angle?

Answer: Defines the angle of the threads relative to the axis of the threaded feature. The thread angle depends on the type of thread profile being used (e.g., V-thread, Acme thread) and is typically standardized for specific thread standards (e.g., ANSI, ISO).

18. What is Thread Depth?

Answer: Specifies the depth to which the threads are cut into the workpiece material. It determines the effective length of engagement between mating threaded components and is typically measured from the crest of the thread to the root.

Job Sheet 3: Create A 2D Drill Toolpath that Drills All Four Holes of the Part

Job Name: Create a 2D drill toolpath that drills all four holes of the part

Activity 3.1: Configure Machine and Setting Up Stock for drilling

Activity 3.2: Create a drill toolpath that drills all four holes of the part.

Activity 3.3: Creating a Dynamic Mill Toolpath

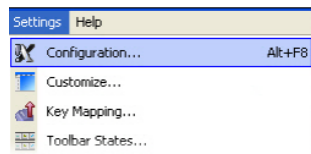
Activity 3.1: Configure Machine and Setting Up Stock for Drilling

Procedure:

1. Collect and wear the PPE.
2. Read job sheet and specification sheet.
3. Start Mastercam using your preferred method Double-click Mastercam's desktop icon.
4. Select the default metric configuration file. Select Settings, Configuration from Mastercam's menu.

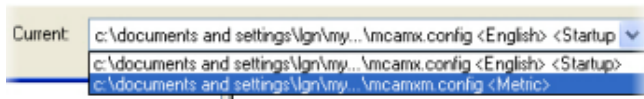


3



4

5. Choose ...\mcamxm.config from the Current drop-down list and Click OK.



6. Start Mastercam from the Mastercam menu bar, choose File, Open.

7. Open BASIC_2D_MACHINING_START.MCX-6,

8. Click OK if prompted to switch to a metric configuration



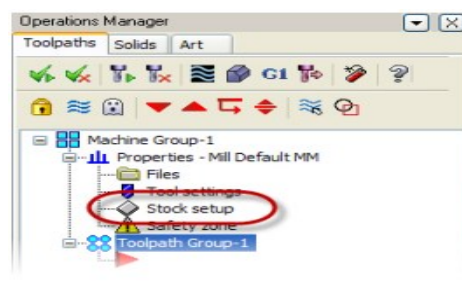
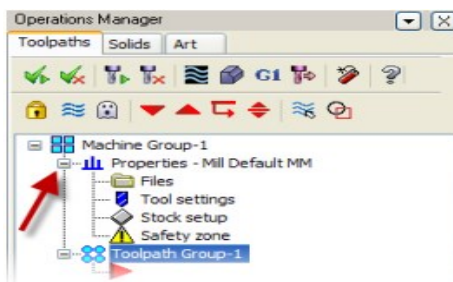
9. Press [Alt +S] to shade the part for easier viewing.

10. From the Mastercam menu, choose Machine Type, Mill, Default to open the default Mill machine definition.

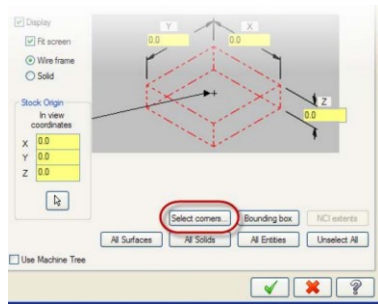
11. Choose File, Save As and save the part under a different file name than the original file

12. In the Operations Manager, click on the plus sign (+) next to Properties - Mill Default MM.

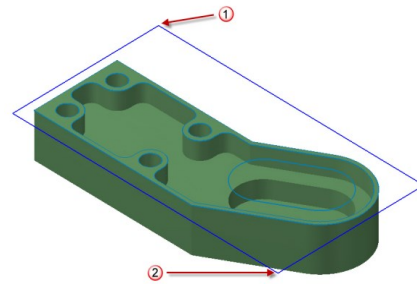
13. Click the Stock Setup icon to display the Stock Setup tab in the Machine Group Properties dialog box



14. Click the Select corners button near the bottom of the dialog box. Mastercam brings you back to the graphics window to select the two opposite stock corners.
15. Click the two opposite corners as shown in the following picture. The Stock Setup tab displays again with X and Y coordinate values from the selected corners.
16. Enter 25 for the Z coordinate in the stock diagram to provide some depth for your stock. Select the Display check box to see the stock model boundaries in the graphics window.

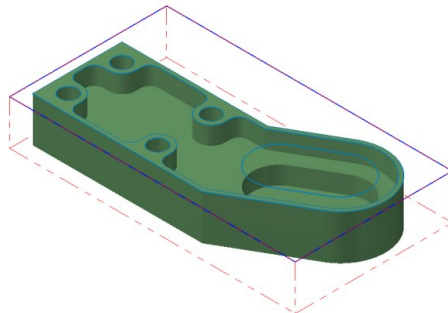


14



15-16

17. Click OK to complete the stock setup. The stock displays as dashed red lines around the part.

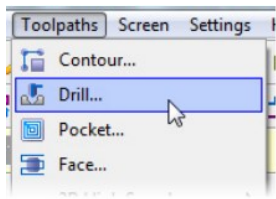


18. After complete work clean work area

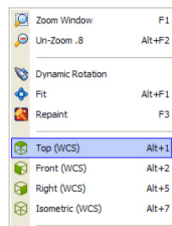
Activity 3.2: Create a Drill Toolpath That Drills All Four Holes of The Part

Procedure:

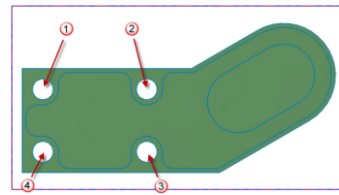
1. Collect and wear the PPE.
2. Read job sheet and specification sheet.
3. From the Mastercam menu, choose Toolpaths, Drill. The Drill Point Selection dialog box opens.
4. Right-click in the graphics window and choose Top (WCS) to switch to a Top graphics view. This will make it easier to select the drill holes.
5. Click each of the center points of the four holes of the part. Choose them in a clockwise order from top left to bottom left as shown in the numbered sequence below.



3

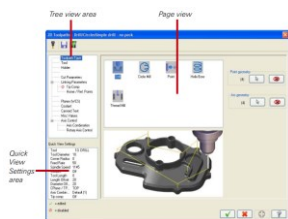


4

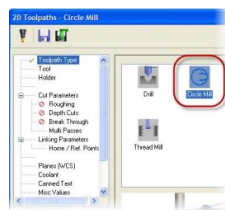


5

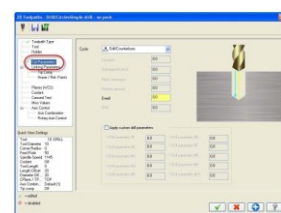
6. Click OK. The 2D Toolpaths - Drill/Circles dialog box opens (Exploring the Toolpath Dialog Boxes)
7. Summarizes key toolpath information from parameters you set on the different pages. It updates automatically as you make changes in the pages, and is always visible.
8. Click on the Circle Mill toolpath type icon. This changes the toolpath to a circle mill toolpath and changes the pages listed in the Tree View area.
9. Click the Drill toolpath type icon to change back to a drill toolpath.
10. Click the Cut Parameters page in the Tree View list. The Page area changes to list cut parameters for the toolpath.
11. Setting Drilling Parameters



7



8

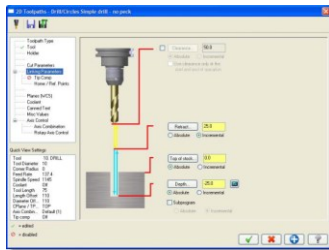


9-10

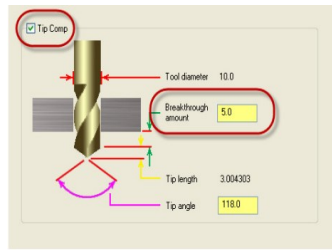
12. Click the Tool page in the Tree View list to pick a tool for the drill toolpath.
13. 2 Click the Select library tool button to pick a tool from one of Mastercam's tool libraries.
14. The Tool Selection dialog box opens and select the 10mm diameter drill from the tool list.

#	ToolName	Dia.	Cor. rad.	Len.
106	9.6 DRILL	9.6	0.0	50.0
107	9.7 DRILL	9.7	0.0	50.0
107	9.75 DRILL	9.75	0.0	50.0
108	9.8 DRILL	9.8	0.0	50.0
108	9.8 DRILL	9.8	0.0	50.0
109	9.9 DRILL	9.9	0.0	50.0
110	10 DRILL	10	0.0	50.0
110	10.0 DRILL	10.0	0.0	50.0
113	10.3 DRILL	10.3	0.0	50.0
115	10.5 DRILL	10.5	0.0	50.0
117	10.75 DRILL	10.75	0.0	50.0
118	10.8 DRILL	10.8	0.0	50.0
120	11 DRILL	11	0.0	50.0
122	11.25 DRILL	11.25	0.0	50.0
125	11.5 DRILL	11.5	0.0	50.0
130	12 DRILL	12	0.0	50.0
135	12.5 DRILL	12.5	0.0	50.0
137	12.75 DRILL	12.75	0.0	50.0

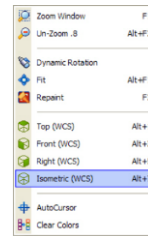
15. Click OK to return to the Tool page. And Click the Linking Parameters page in the Tree View list.
16. Enter -25.0 in the Depth field.
17. Click the Tip Comp page in the Tree View list. Notice that this page is off by default.
18. Select the Tip Comp check box to turn on this feature and enter 5.0 in the Breakthrough amount field.
19. Right-click in the graphics window and choose Isometric (WCS) from the menu to view the part and toolpath in the isometric view.
20. Your toolpath should look like this. The cyan lines are feed motion and the yellow lines are rapid motion.



16

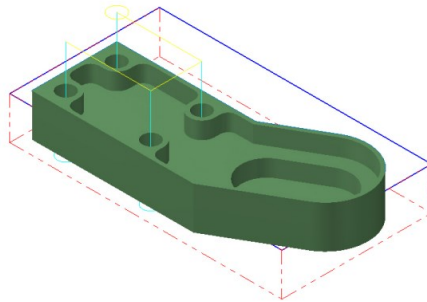


18



19

21. Right-click in the graphics window again, and choose Top (WCS).

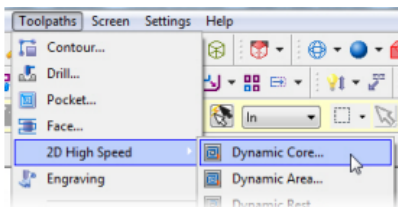


22. Choose File, save from the Mastercam menu or click the Save button to save your part.
23. After complete work clean work area

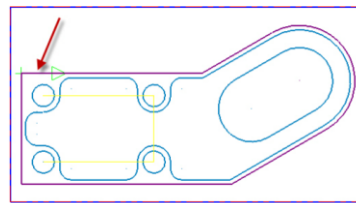
Activity 3.3: Creating a Dynamic Mill Toolpath

Procedure:

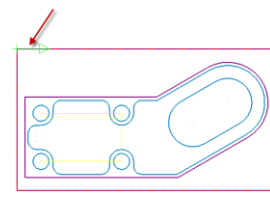
1. Collect and wear the PPE.
2. Read job sheet and specification sheet.
3. From the Mastercam menu, choose Toolpaths, 2D High Speed, Dynamic Core. The Chaining dialog box opens.
4. Press [Alt+S] to turn off shading and make it easier to select the chains. 3 Click the outside contour of the part to chain it.
5. The chaining arrow should point clockwise. If the chaining direction arrow is pointing counterclockwise, click the Reverse button on the Chaining dialog box.
6. Click the top of the stock boundary as the second chain. This second chain represents the size of the stock material. The chaining arrow should again point clockwise.



4



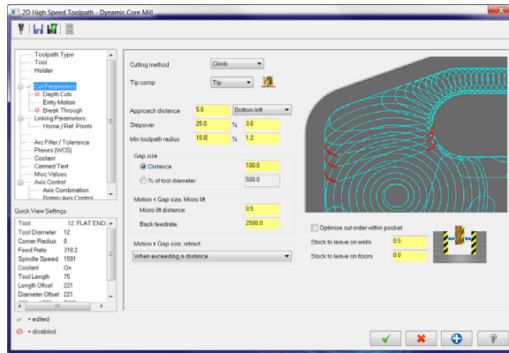
5



6

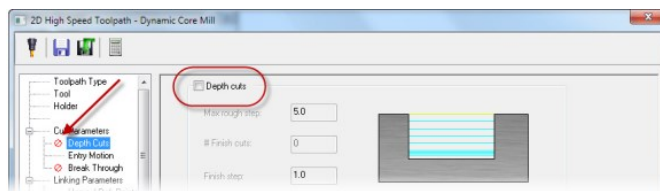
7. Click OK on the Chaining dialog box to chain the part. The Chaining dialog box closes and the Chaining Options dialog box opens.
8. Select Do not show this dialog again this session and click OK. The 2D High Speed Toolpath dialog box opens.
9. Click the Tool page in the Tree View list to select a tool for this toolpath.
10. Click the Select library tool button. The default metric tool library opens.
11. Select the 12mm diameter flat endmill, and click OK.12. Setting the Dynamic Mill Toolpath Parameters
13. Click the Cut Parameters page in the Tree View area to enter values for different cutting parameters and compensation options.
14. Set the following parameters:

- Enter 5.0 for the Approach distance, which adds the specified absolute distance to the beginning of the toolpath's first cut.
- Enter 10% for the Min toolpath radius. This parameter is used to calculate 3D arc moves and reduce sharp corner motion between passes.
- Enter 0.5 for the Micro lift distance. Micro lifts are slight lifts during back moves that help clear chips and minimize excessive tool heating.
- Select When exceeding a distance from the Retract drop-down list, select the Distance radio button in the Gap size group box, and enter 100.0. Mastercam adds retract motion when the next cut begins at a distance greater than the distance you define.
- Enter 0.5 for the Stock to leave on walls. This leaves 0.5mm of stock on the outer walls.
- Enter 0 for Stock to leave on floors.



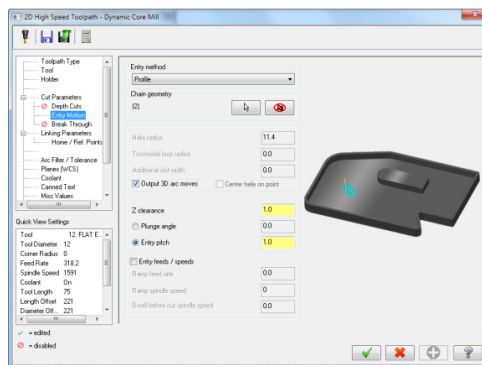
15. Click the Depth cuts page in the Tree View area.

16. Deselect the Depth cuts check box to turn off depth cuts for this toolpath.



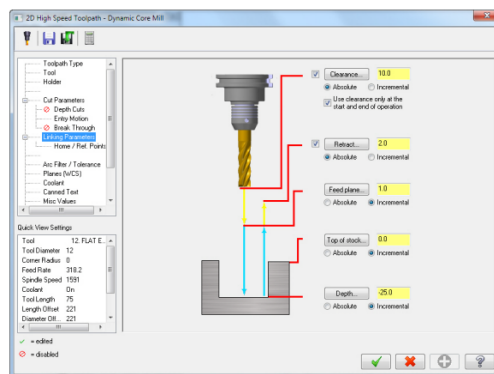
17. Click the Entry Motion page in the Tree View list to set how and where the tool enters the stock.

18. Set the following parameters:



19. Click the Linking Parameters page in the Tree View list to set important heights such as clearance, retract, and feed plane, as well as the final toolpath depth.

20. Set the following parameters:



- The toolpath cleans most of the material outside the part using smooth, efficient tool motion.
 - Click OK to generate the toolpath.
 - Right-click in the graphics window and choose Isometric (WCS) from the menu to view the part and toolpath in the isometric view.
-
21. Right-click in the graphics window again and choose Top (WCS) from the menu to view the part and toolpath in the top view.
 22. Choose File, save from the Mastercam menu or click the Save button to save your part.
 23. After complete work clean work area

Specification Sheet 3: Create A 2D Drill Toolpath That Drills All Four Holes of The Part

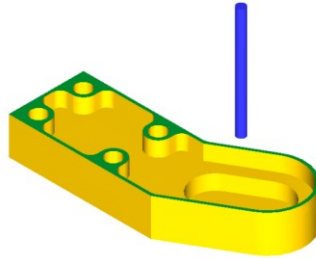
Job Name: Create a 2D drill toolpath that drills all four holes of the part

Activity 3.1: Configure Machine and Setting Up Stock for drilling

Activity 3.2: Create a drill toolpath that drills all four holes of the part.

Activity 3.3: Creating a Dynamic Mill Toolpath

Conditions for the job: You must practice safe operation procedures at all times during the assessment. You will be required to demonstrate OSH competencies to Create a 2D drill toolpath that drills all four holes of the part



Required Personal Protective Equipment (PPE): Pre/person

Sl.	Name of the PPE	Specification	Unit	Quantity	Remarks
a.	Hand Gloves	Cotton	pair	01	
b.	Mask	N 95	Nos	01	
c.	Safety shoe	As required	pair	01	
d.	Safety goggles	Standard	Nos	01	
e.	Apron	Standard	Nos	01	

Required Tools and Equipment

Sl.	Name of tools and equipment	Specification	Unit	Quantity	Remarks
7	Desktop computer	i5(8GB RAM)	No	01	

Learning Outcome -4: Create 3D Tool Paths

Learning Outcome -4: Create 3D tool paths	
Assessment Criteria	<ol style="list-style-type: none"> 1. 3D tool path is prepared 2. level as per matching strategy is set 3. Surface is created as per requirement 4. Silhouette command is executed 5. Edge curve command is executed 6. Drive surface, containment region, check surface is selected 7. 3D rough & finishing tool path is executed 8. Stock model & rest milling is executed 9. 3D tool path is verified 10. G&M code is generated
Conditions and Resources	<ol style="list-style-type: none"> 1. Workplace or Simulated Workplace 2. CBLM 3. Handout 4. Laptop 5. Multimedia Projector 6. Paper, Pen, Pencil, 7. Internet Facilities 8. White Board and 9. Audio Video Devices 10. Necessary Tools and Equipment 11. Necessary Materials 12. Necessary PPE
Contents	<ol style="list-style-type: none"> 1. Necessary machines 2. Model-Specific Machining 3. Tool path sequence/continuity 4. Suitable tool diameter and length <ul style="list-style-type: none"> ▪ Acknowledgment of CAM parameters ▪ Using measuring instruments ▪ Calculating cutting speed, feed rate, and depth of cut ▪ Simulating and editing program ▪ Saving and downloading program ▪ Writing CNC program ▪ Determining the optimum sequence of operations 5. Level <ul style="list-style-type: none"> ▪ Level number ▪ Level on/off ▪ Level increment ▪ Visibility ▪ Entities 6. Surface

	<ul style="list-style-type: none"> ▪ From solid ▪ Flat boundary ▪ Extend ▪ Fill holes ▪ Offset surface <p>7. Edge curve</p> <ul style="list-style-type: none"> ▪ Curve on one edge ▪ Curve on all edges ▪ Flowline curve ▪ Curve slice <p>8. 3D rough & finishing tool</p> <ul style="list-style-type: none"> ▪ Pocket ▪ Area rough ▪ Opti rough ▪ Contour ▪ Scallop ▪ Waterline ▪ Horizontal ▪ Raster <p>9. Hybrid</p>
Job/Task/Activity	<ol style="list-style-type: none"> 1. Identify the required machine by inspecting the workpiece 2. Select the appropriate tool path sequence for a model-specific machining operation 3. Select pre-machining operations as required 4. Prepare work holding and clamping to avoid tool clash <p>Select the appropriate cutting parameters</p>
Training Method	<ol style="list-style-type: none"> 1. Discussion 2. Presentation 3. Demonstration 4. Guided Practice 5. Individual Practice 6. Project Work 7. Problem Solving <p>Brainstorming</p>
Assessment Method	<ol style="list-style-type: none"> 1. Written Test 2. Demonstration 3. Oral questioning 4. Portfolio

Learning Experience 4: Create 3D Tool Paths

In order to achieve the objectives stated in this learning guide, you must perform the learning steps below. Beside each step are the resources or special instructions you will use to accomplish the corresponding activity.

Learning Activities	Recourses/Special Instructions
1. Trainee will ask the instructor about the learning materials	1. Instructor will provide the learning materials ‘Apply CAM Software for Machining
2. Read the Information sheet and complete the Self Checks & Check answer sheets on “Create program using G & M code”	2. Read Information sheet 1: Create 3D tool paths Answer Self-check 1: Create 3D tool paths 3. Check your answer with Answer key 1: Create 3D tool paths
3. Read the Job/Task Sheet and Specification Sheet and perform job/Task	4. Job/Task Sheet and Specification Sheet Job sheet 4: Perform 3D Machining in Mastercam as per selected Mechanical 3D Model. Activity 4.1: Assign a Machine Definition and Setting Stock for Machining Activity 4.2: Perform program for Drilling Holes Activity 4.3: Perform program for Roughing the Pocket Activity 4.4: Roughing the Slot Activity 4.5: Rest Roughing the Interior Activity 4.6: Finishing the Pocket Floor Activiety4. 7: Perform Program for Finishing the Slot Activiety4. 8: Perform Program for Roughing the Exterior Activity 4.9: Perform Program for Finishing the Exterior Specification sheet 4: Perform 3D Machining in Master CAM as per selected Mechanical 3D Model.

Information Sheet 4: Create 3D Tool Paths

Learning Objective: After completion of this information sheet, the learners will be able to explain, define and interpret the following contents:

- 4.1. Necessary machines
- 4.2. Model-Specific Machining
- 4.3. Tool path sequence/continuity
- 4.4. Suitable tool diameter and length
- 4.5. Level
- 4.6. Surface
- 4.7. Edge curve
- 4.8. 3D rough & finishing tool

4.1. Necessary Machines

For Mastercam 3D tool pathing, you will typically require CNC machines capable of 3-axis, 4-axis, or 5-axis machining, depending on the complexity of the parts being manufactured.

4.2. Model-Specific Machining

This involves programming toolpaths in Mastercam tailored to the specific geometry and features of the CAD model being machined. Each part may require unique toolpath strategies to achieve the desired results.

4.3. Tool Path Sequence/Continuity

Determining the order of operations and ensuring that toolpaths are created in a logical sequence to efficiently machine the part. Continuity ensures smooth transitions between toolpath operations, minimizing tool retractions and reducing cycle times.

4.4. Suitable Tool Diameter and Length

Selecting cutting tools with appropriate diameters and lengths based on the part geometry, material, and desired cutting parameters. Tools should be capable of reaching all required features without interference and should be suitable for the material being machined.

a. Acknowledgment of cam parameters

Understanding and adjusting CAM parameters such as toolpath settings, stepovers, feeds, speeds, and depths of cut to optimize machining operations for the specific material and machine setup.

b. Using measuring instruments

Employing measuring instruments such as calipers, micrometers, or height gauges to verify part dimensions, tool offsets, and machine setups, ensuring accurate machining results.

c. Calculating cutting speed, feed rate, and depth of cut

Calculating optimal cutting speeds, feed rates, and depths of cut based on factors such as material properties, tool material, tool geometry, and machine capabilities to achieve efficient material removal and desired surface finish.

d. Simulating and editing program

Simulating toolpaths in Mastercam to visualize machining operations and identify potential issues before running the program on the CNC machine. Editing programs to adjust as needed to optimize machining efficiency.

e. Saving and downloading program

Saving CNC programs in the appropriate file format and downloading them to the CNC machine for execution, ensuring compatibility and proper communication between Mastercam and the CNC machine.

f. Writing CNC program

Writing CNC programs using G-code or machine-specific programming languages to control the CNC machine's movements and machining operations, translating toolpaths generated in Mastercam into machine-readable instructions.

g. Determining the optimum sequence of operations

Planning the sequence of machining operations to minimize setup changes, tool changes, and machine idle time, maximizing efficiency and productivity while ensuring accurate and consistent machining results.

4.5. Level

a. Level number

Each level in Mastercam is assigned a unique number for identification and organization purposes. The level number helps users manage and reference different levels within the CAM environment.

b. Level on/off

Mastercam allows users to toggle individual levels on or off, controlling the visibility of entities (such as geometry, toolpaths, or annotations) assigned to each level. Turning a level on displays its associated entities, while turning it off hides them from view.

c. Level increment

The level increment specifies the numerical increment used when assigning new levels in Mastercam. For example, if the level increment is set to 10, the next available level number will be 10 units higher than the previous level number.

d. Visibility

Visibility settings control whether entities assigned to a specific level are visible or hidden within the Mastercam workspace. Users can adjust visibility settings to show or hide entities for easier editing, visualization, or analysis.

e. Entities

Entities refer to geometric elements, toolpaths, annotations, or other objects created or imported into the Mastercam environment. These entities can be assigned to specific levels for organization and management purposes, allowing users to control their visibility and editing options.

By effectively managing levels and their associated entities in Mastercam, users can organize complex projects, streamline workflow processes, and control the visibility of

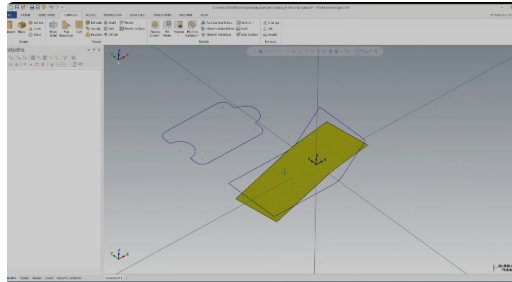
different components within the CAM environment.

4.6. Surface

a. From solid

This feature in Mastercam allows users to create surfaces directly from solid models. It automatically generates surfaces that match the geometry of the solid model, facilitating the creation of complex toolpaths based on solid geometry.

b. Flat boundary



The flat boundary operation in Mastercam allows users to define planar regions within a surface. This is useful for specifying areas where toolpaths should be confined or restricted to, such as flat surfaces or regions with specific height or angle requirements.

c. Extend

The extend operation in Mastercam extends selected surface edges to meet neighboring surfaces or boundaries. This is useful for ensuring continuity and smooth transitions between adjacent surfaces, which is crucial for generating accurate and visually appealing toolpaths.

d. Fill holes

This operation fills holes or voids in surfaces, either by automatically detecting and filling them or by specifying the hole boundaries manually. Filling holes is essential for creating watertight surfaces and ensuring that toolpaths are generated correctly across the entire surface geometry.

e. Offset surface

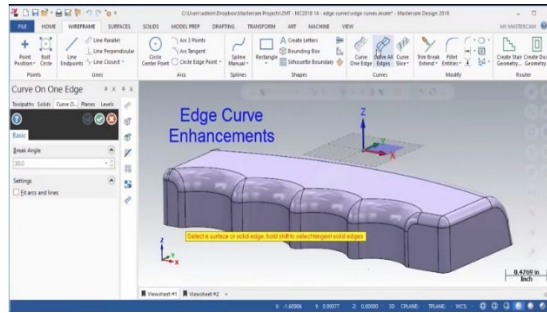
The offset surface operation in Mastercam creates new surfaces offset from existing ones by a specified distance. This is useful for creating clearance surfaces, machining allowances, or boundaries for toolpath generation. Offset surfaces can be used to define regions for roughing, finishing, or other machining operations.

By leveraging these surface operations in Mastercam, users can manipulate and prepare surface geometry effectively for 3D tool pathing. These operations help ensure that surfaces are properly defined, interconnected, and optimized for generating accurate and efficient toolpaths for CNC machining.

4.7. Edge Curve

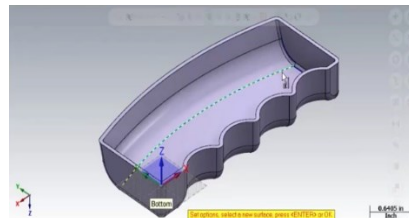
a. Curve on one edge

This operation allows users to create a curve along a single edge of a surface or solid model in Mastercam. It is useful for defining boundaries or guiding toolpaths along specific edges of the geometry.

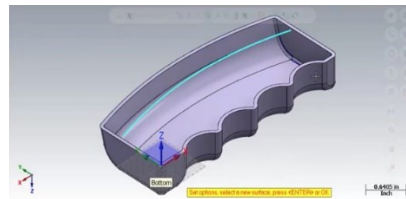


There are some new enhancements to the Curves commands in Mastercam 2018. The 2D/3D construction switch will determine if the curves are created on edge or projected to the current construction plane, and there are several new Selection and Filtering options available for both Curve One Edge and Curve All Edges.

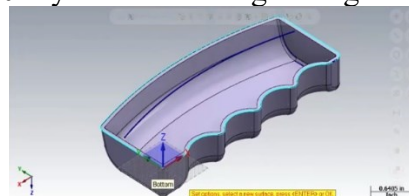
When curves are created on the 3D face edges in 3D mode the curves stay on the face edges.



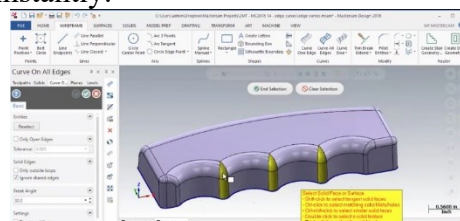
However, if you switch from 3D to 2D construction mode the curves are projected to the current plane and construction elevation.



You can also hold the **Shift** key and all the tangent edges will be picked with one click.



In the **Curve All Edges** command you can use **CTRL** and/or **Shift** to have tangent or matching radii selected instantly.



Finally, the option check boxes for **Ignore Shared Edges** and **Only Outside Loops** allow you to choose multiple faces and keep just the curves you need without redundant or extra geometry in the results.

As you can see, these new options will help get the results you want quickly and reduce your preparation time for making toolpath.

b. Curve on all edges

With this operation, users can generate curves along all edges of a surface or solid model in Mastercam. It automatically creates curves along every edge, making it efficient for defining complete boundaries or guiding toolpaths around the entire geometry.

c. Flowline curve

The flowline curve operation in Mastercam generates a curve that follows the flow of surface curvature. It is particularly useful for creating toolpath boundaries or guiding toolpaths along complex surface contours, ensuring smooth and consistent machining motions.

d. Curve slice

This operation slices a surface or solid model with a plane or curve, generating curves at the intersection between the geometry and the slicing element. It is useful for extracting specific cross-sections or profiles from 3D models, which can then be used for further analysis or toolpath generation.

4.8. 3D Rough & Finishing Tool

a. Pocket

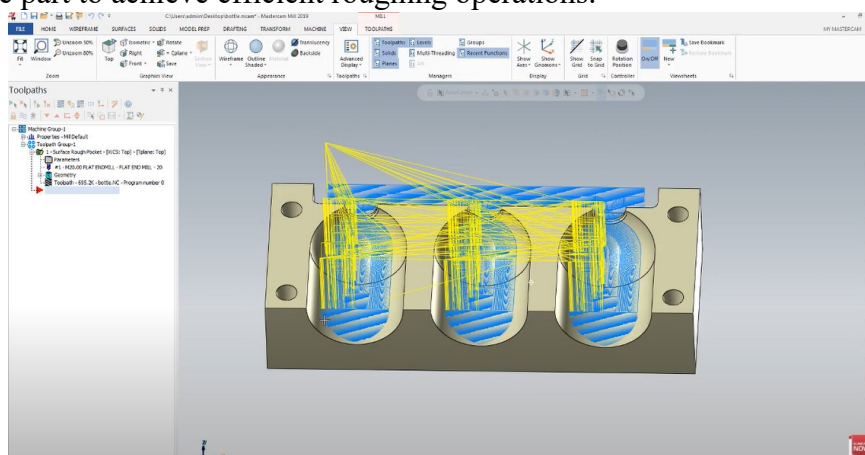
The pocket toolpath is used to remove material from within closed boundaries, such as pockets, cutouts, or recessed areas on the part. It removes material in multiple passes, starting from the outside and working towards the center, ensuring efficient material removal.

b. Area rough

The area rough toolpath is similar to the pocket toolpath but is optimized for roughing out larger, more irregularly shaped areas of material. It removes material using multiple passes, efficiently clearing away excess material while minimizing tool engagement and cycle time.

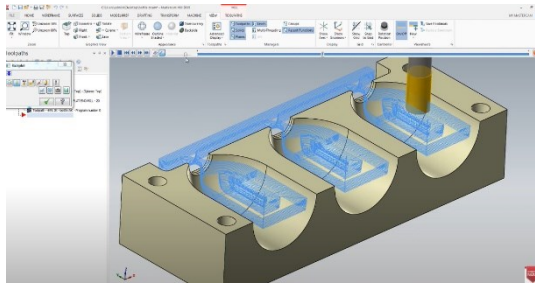
c. Opti rough

Opti rough is an advanced roughing toolpath strategy that optimizes toolpath motion and cutting conditions to maximize material removal rates while minimizing cutting forces and tool wear. It dynamically adjusts cutting parameters based on the geometry of the part to achieve efficient roughing operations.



d. Contour

The contour toolpath follows the contours of the part geometry, removing material along defined paths or boundaries. It is often used for semi-finishing or finishing operations to achieve precise part dimensions and surface finish.



e. Scallop

The scallop toolpath is a finishing strategy that removes the remaining material between adjacent passes to achieve a smooth surface finish. It minimizes the scallop height or stepover distance between toolpath passes, resulting in finer surface finishes.

f. Waterline

The waterline toolpath follows horizontal contours or levels within the part geometry, removing material layer by layer. It is particularly useful for machining shallow features or flat surfaces and is commonly used for finishing operations.

g. Horizontal

The horizontal toolpath removes material along horizontal planes within the part geometry, making it ideal for machining flat or gently sloping surfaces. It ensures consistent tool engagement and efficient material removal, particularly for parts with horizontal features.

h. Raster

The raster toolpath removes material in a back-and-forth pattern along a specified direction, typically perpendicular to the toolpath direction. It is suitable for machining flat or prismatic surfaces and helps achieve uniform material removal and surface finish.

i. Hybrid

The hybrid toolpath combines multiple cutting strategies, such as raster, contour, and waterline, to optimize material removal and surface finish for complex part geometries. It dynamically adjusts cutting parameters based on the geometry of the part to achieve efficient and effective machining results. These 3D rough and finishing toolpaths in Mastercam offer a range of strategies for efficiently machining complex part geometries while achieving the desired surface finish and dimensional accuracy. Each toolpath type has its advantages and is selected based on the specific requirements of the part and machining operation.

Self-Check 4 Create 3D Tool

1. What is the Procedure of Saving and Downloading Program?

Answer:

2. What are Entities?

Answer:

3. What is 3D Pocket?

Answer:

Answer Sheet 4 Create 3D Tool

1. What is the Procedure of Saving and Downloading Program?

Answer: Saving CNC programs in the appropriate file format and downloading them to the CNC machine for execution, ensuring compatibility and proper communication between Mastercam and the CNC machine.

2. What are Entities?

Answer: Entities refer to geometric elements, toolpaths, annotations, or other objects created or imported into the Mastercam environment. These entities can be assigned to specific levels for organization and management purposes, allowing users to control their visibility and editing options.

3. What is 3D Pocket?

Answer: The pocket toolpath is used to remove material from within closed boundaries, such as pockets, cutouts, or recessed areas on the part. It removes material in multiple passes, starting from the outside and working towards the center, ensuring efficient material removal.

Job Sheet 4: Perform 3D Machining in Mastercam as Per Selected Mechanical 3D Model

Job Name: Perform 3D Machining in Mastercam as per selected Mechanical 3D Model

Activity 4.1: Assigning a Machine Definition and Setting Stock for Machining

Activity 4.2: Perform program for Drilling Holes

Activity 4.3: Perform program for Roughing the Pocket

Activity 4.4: Roughing the Slot

Activity 4.5: Rest Roughing the Interior

Activity 4.6: Finishing the Pocket Floor

Activity 4.7: Finishing the Slot

Activity 4.8: Roughing the Exterior

Activity 4.9: Finishing the Exterior

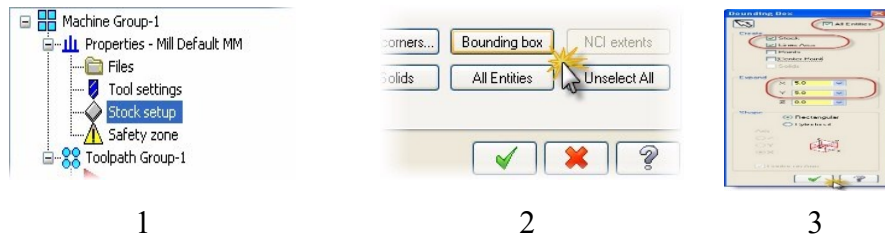
Activity 4.1: Assigning a Machine Definition and Setting Stock for Machining

Procedure:

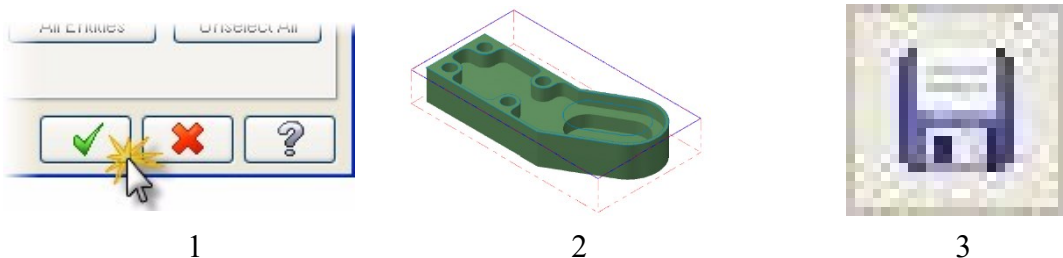
1. Open the tutorial part file Basic_3D_Machining_Part1_Start.MCX-5, which was provided with the tutorial.
2. Click OK if prompted to switch to a metric configuration.
3. Choose Machine Type, Mill, Default to open the default Mill machine definition.



4. Choose File, Save As, and save the part under a different file name. This protects the original tutorial file from being overwritten.
5. Select Stock setup. If necessary, click the [+] next to Properties to expand the list
6. Click Bounding box. The Bounding Box dialog box opens
7. Set the options and parameters as shown, then click OK.



8. Click OK in the Machine Group Properties dialog box
9. Press [Alt+F1], or right-click and select Fit, to fit the geometry to the screen.
10. Choose File, Save or click the Save button to save the part with the machine definition and defined stock.

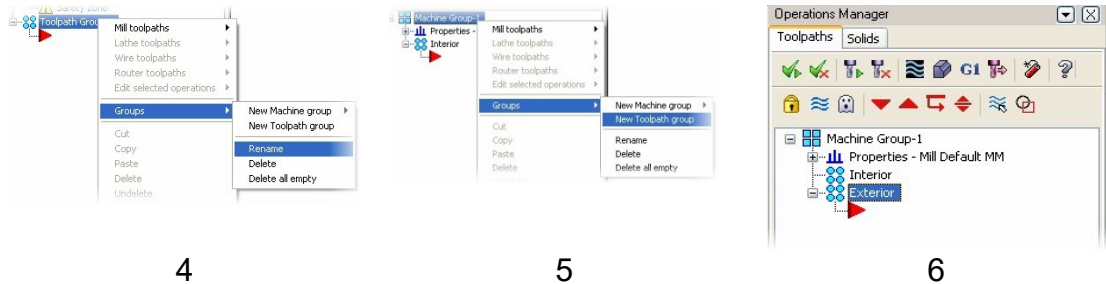


11. Save the file.

Activity 4.2: Perform Program for Drilling Holes

Procedure:

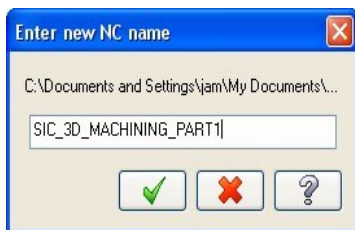
1. Open Master cam software
2. Import selected part model for CAM
3. Perform right-click Toolpath Group-1, select Groups, Rename.
4. Type Interior and press [Enter].
5. Toolpath Manager, right-click on Machine Group-1, select Groups, New Toolpath Group
6. Type Exterior and press [Enter]. The Toolpath Manager should look as shown



7. Choose File, Save or click the Save button to save the part.
8. Click the Moveinsert arrow up one item button Toolpath Manager should look as shown after performing this step.
9. Right-click in the graphics window and choose Isometric (WCS) from the menu to view the part and toolpath in the isometric view.
10. Choose Toolpaths, Drill



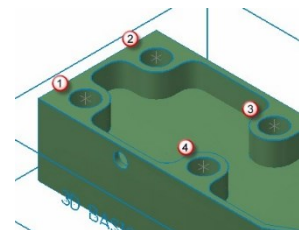
11. Click OK when prompted to enter new NC name
12. Use the default option for Select drill point position in the graphics screen in the Drill Point Selection dialog box
13. Select the arc centers of the four holes as indicated. The autocursor changes to indicate the arc center as you approach



11

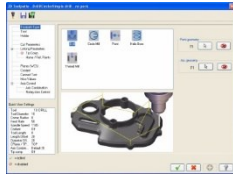


12

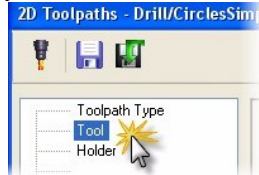


13

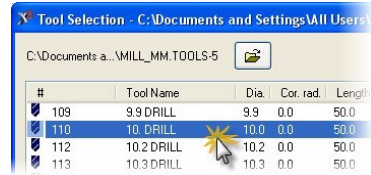
14. Click OK in the Drill Point Selection dialog box . The 2D Toolpaths dialog box opens to the Toolpath Type page. Drill is selected as the toolpath type.
15. Select Tool from the Tree View pane on the left.
16. Click the Select library tool button. Select the 10mm drill and click OK



14

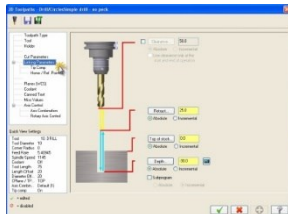


15



16

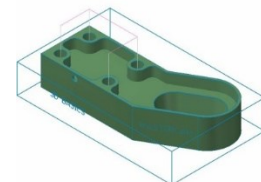
17. Select Linking Parameters from the Tree View pane and enter values as shown.
18. Select Tip Comp from the Tree View pane. Click the Tip Comp checkbox to activate this feature. Use default values for the tip comp parameters.
19. Click OK in the 2D Toolpath dialog box to generate the drill toolpath



17



18



19

20. Click the Toggle display on selected operations button. If necessary, select the Drill/Counterbore operation.
21. Save your part file

Activity 4.3: Perform Program for Roughing the Pocket

Procedure:

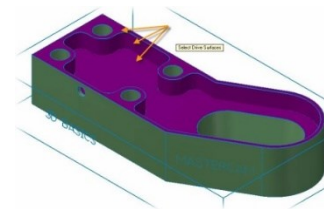
1. Open Master cam software
2. Import selected part model for CAM
3. Choose Toolpaths, Surface High Speed
4. select the option to eliminate this dialog box and make refinement available for use. Click OK to close the dialog box.
5. Follow the prompt to select drive surfaces as shown.



3



4



5

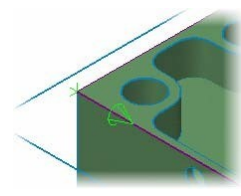
6. Press [Enter] or click the End Selection button to accept the selection.
7. Click the Select button in the Containment area of the Toolpath/Surface selection dialog box.
8. Click the C-plane radio button. C-plane selection limits chaining to entities that are parallel to the current construction plane.
9. Select the chain on the top of the part. This selection can be in either direction for a closed containment boundary



7

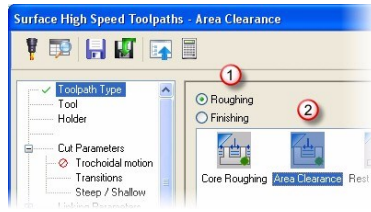


8

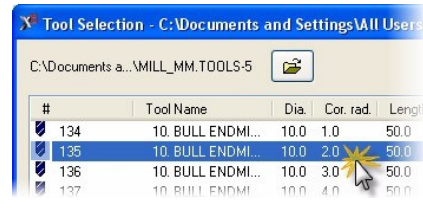


9

10. Click OK on the Chaining dialog box to return to the Toolpath/Surface selection dialog box.
11. Click OK on the Toolpath/surface selection dialog box.
12. The Surface High Speed toolpaths dialog box opens on the Toolpath Type page.
13. Select Roughing and Area Clearance on the Toolpath Type page.
14. Click the Select library tool button. The default metric tool library opens.
15. Select the 10mm bull endmill with 2mm corner radius and click OK.

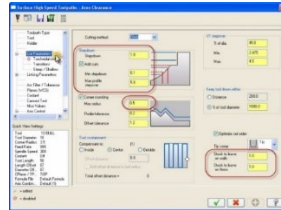


12

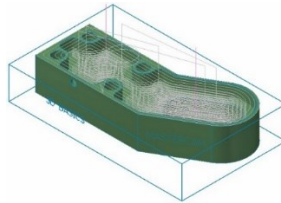


15

16. Select Tool from the Tree View pane.
17. Select Cut Parameters from the Tree View pane. Set the parameters as indicated.



18. Click OK to generate the area clearance toolpath.

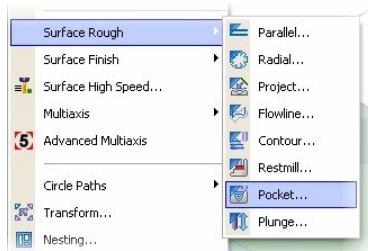


19. Turn off the toolpath display for the operation.
20. Save your part file.

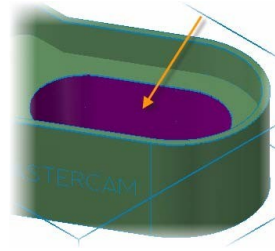
Activity 4.4: Perform CAM Programing (Roughing the Slot)

Procedure:

1. Open Master cam software
2. Import selected part model for CAM
3. Choose Toolpaths, Surface Rough, Pocket.
4. Follow the prompt to select drive surfaces as shown.

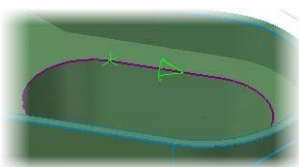


3

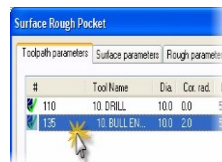


4

5. Press [Enter] or click the End Selection button.
6. Click the Select button in the Containment area of the Toolpath/Surface selection dialog box.
7. The Chaining dialog box opens.
8. Click the C-plane radio button in the Chaining dialog box.
9. Select the chain at the top of the slot as shown.
10. Click OK on the Chaining dialog box.
11. Click OK on the Toolpaths/surface selection dialog box.
12. The Surface Rough Pocket dialog box opens to the Toolpath parameters tab.
13. Select the 10mm bull endmill with the 2mm corner radius showing in the tool list.
14. Click the Surface parameters tab.



9

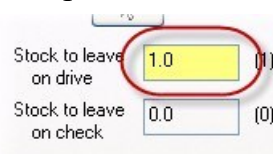


13

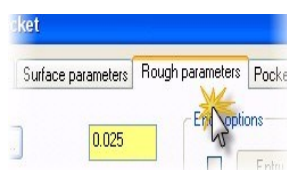


14

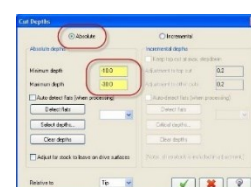
15. Set Stock to leave on drive to a value of 1mm.
16. Click the Rough parameters tab.
17. Click the Cut Depths button.
18. Set the parameters as shown and click OK.



15



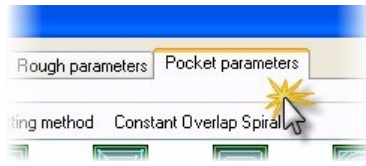
16



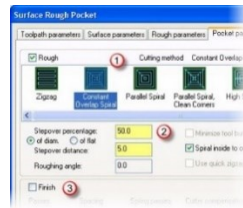
18

19. Click the Pocket parameters tab.

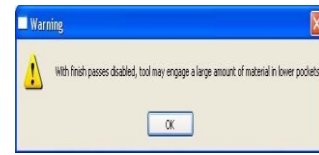
20. Click Constant Overlap Spiral, set the parameters as shown, and click OK.
21. The following warning message displays because the finish passes are turned off. Click OK to continue.



19

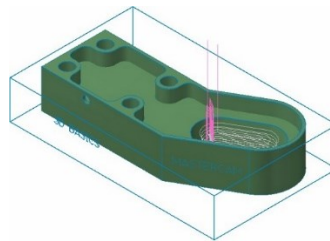


20



21

22. The surface rough pocket toolpath is generated on the selected geometry.

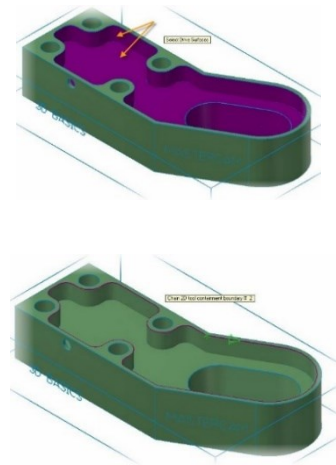


23. Turn off the toolpath display for the operation.
24. Save your part file.

Activity 4.5: Rest Roughing the Interior

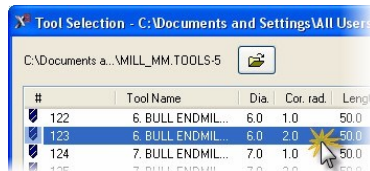
Procedure

1. Choose Toolpaths, Surface High Speed.
2. Follow the prompt to select drive surfaces as shown.
3. Press [Enter] or click End Selection to accept the selection.
4. Click the Select button in the Containment area of the Toolpath/Surface selection dialog box.
5. The Chaining dialog box opens.
6. Click the C-plane radio button.
7. Follow the prompt to chain a containment boundary as shown.
8. Click OK in the Chaining and the Toolpath/surface selection dialog boxes The Surface High Speed toolpaths dialog box opens.
9. Select Toolpath Type from the Tree View pane, and then select Roughing and Rest Roughing on the Toolpath Type page
10. Select Tool from the Tree View pane.
11. Click the Select library tool button. The tool library defined in the Machine Group Properties opens

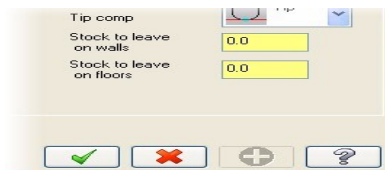


Select library tool...

12. Select the 6mm bull endmill with the 2mm corner radius and click OK



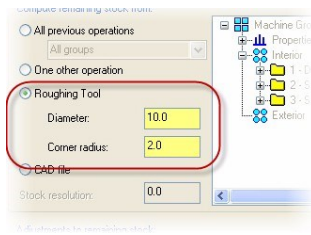
13. Select Cut Parameters from the Tree Viewpane. Set the stock to leave values as indicated



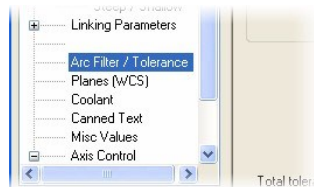
14. Select Rest Material from the Tree Viewpane. Click the plus sign in front of Cut Parameters to expand the tree if necessary.



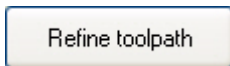
15. Select Roughing Tool and set the values as shown.



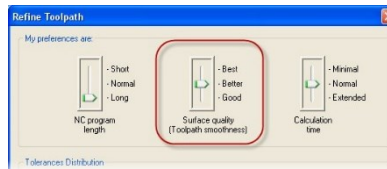
16. Select **Arc Filter/Tolerance** from the Tree View pane.



17. Click the **Refine toolpath** button. The Refine Toolpath dialog box opens



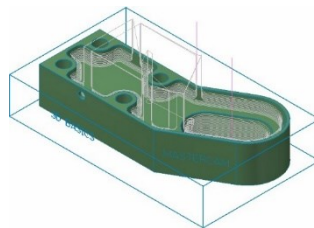
18. Move the slider for Surface quality to the **Better** setting.



19. Click OK to accept the settings on the Refine Toolpath dialog.

20. Click OK in the Surface High Speed Toolpath dialog box.

21. The rest roughing toolpath is generated on the selected geometry.



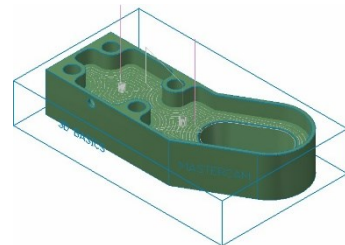
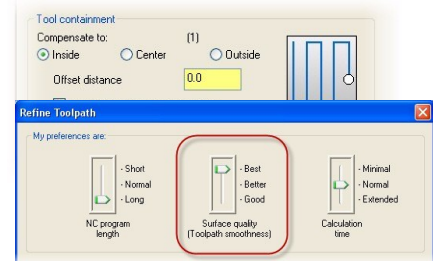
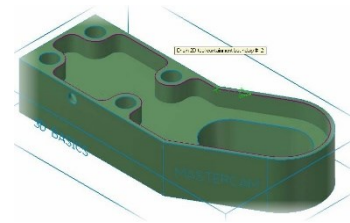
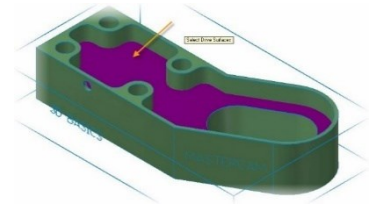
22. Turn off the toolpath display for the operation.

23. Save your part file.

Activity 4.6 Perform Program for Finishing Pocket Floor

Procedure:

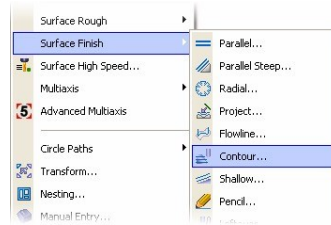
1. Choose Toolpaths, Surface High Speed. The prompt to select Drive Surfaces appears.
2. Follow the prompt to select drive surfaces as shown
3. Press [Enter] or click the End Selection button to accept the selection.
4. Click the Select button in the Containment area of the Toolpath/Surface selection dialog box. The Chaining dialog box opens.
5. Click the C-plane radio button.
6. Follow the prompt to chain a containment boundary as shown.
7. Click OK in the Chaining and the Toolpath/surface selection dialog boxes. The Surface High Speed toolpaths dialog box opens on the Toolpath Type page.
8. Select the Toolpath Type page, and then Finishing and Horizontal Area.
9. Horizontal Area is designed to quickly and efficiently machine flat areas of a part.
10. Select Tool from the Tree View pane.
11. Select the 6mm bull endmill with the 2mm corner radius showing in the tool list.
12. Select Cut Parameters from the Tree View pane. Set the tool containment as shown.
13. Select Arc Filter/Tolerance from the Tree View pane.
14. Click the Refine Toolpath button.
15. Move the slider for Surface quality to the Best setting.
16. Click OK to accept the settings on the Refine Toolpath dialog.
17. Click OK in the Surface High Speed Toolpath dialog box to generate the horizontal area toolpath.
18. Turn off the toolpath display for the operation.
19. Save your part file.



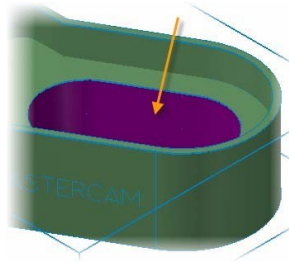
Activity4.7 Perform Program for Finishing the Slot

Procedure:

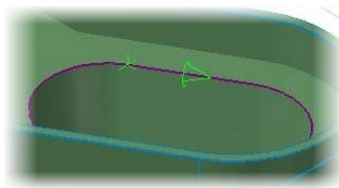
1. Choose **Toolpaths, Surface Finish, Contour**.



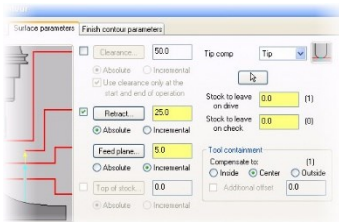
2. Follow the prompt to select drive surfaces as shown.



3. Press **[Enter]** or click the **End Selection** button to accept the selection.
4. Click the **Select** button in the Containment area of the Toolpath/Surface selection dialog box. The Chaining dialog box opens.
5. Click the **C-plane** radio button.
6. Follow the prompt to chain a containment boundary as shown.



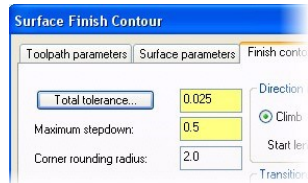
7. Click **OK** on the Chaining and Toolpaths/surface selection dialog box. The Surface Finish Contour dialog box opens to the Toolpath parameters tab.
8. Select the 6mm bull endmill with the 2mm corner radius showing in the tool list.
9. Click the **Surface parameters** tab. Set the parameters as shown.



10. Click the **Finish contour parameters** tab.



11. Set the parameters as shown

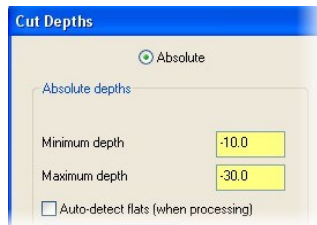


12. Click the **Total tolerance** button.

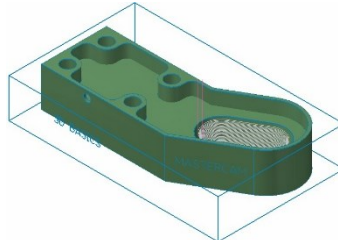
13. Move the slider for Surface quality to the Best setting and click **OK**.

14. Click the **Cut Depths** button.

15. Set the parameters as shown and click **OK**



16. Click **OK** from the Surface Finish Contour dialog box to generate the finish contour toolpath



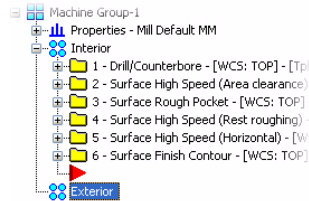
17. Turn off the toolpath display for the operation.

18. Save your part file.

Activity 4.8 Perform Program for Roughing the Exterior

Procedure:

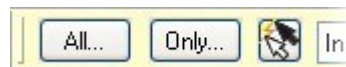
1. Select the toolpath group Exterior.
2. Click the minus (-) in front of the Interior toolpath group to collapse the tree



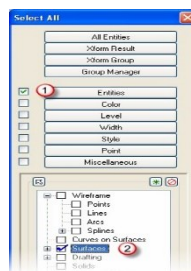
3. Click the Position insert arrow after selected operation or after selected group button. The insertion arrow is placed in the Exterior group.



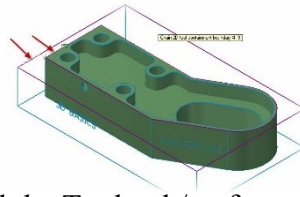
4. Choose Toolpaths, Surface High Speed.
5. Click the All button on the General Selection ribbon bar when prompted to select drive surfaces



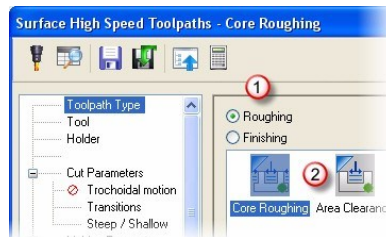
6. Select the Entities and Surfaces checkboxes in the Select All dialog box as shown.



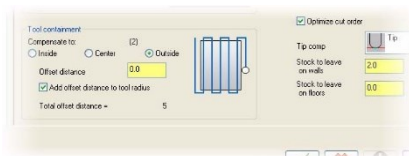
7. Click OK in the Select
8. All dialog boxes.
9. Press [Enter] or click the End Selection button.
10. Click the Select button in the Containment area of the Toolpath/Surface selection dialog box. The Chaining dialog box opens.
11. Click the C-plane radio button.
12. Select two chains as shown to be used as containment boundaries. The chains lie on the top of the part and the top of the stock boundary created in the Toolpath Setup lesson



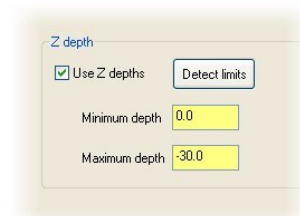
13. Click OK in the Chaining and the Toolpath/surface selection dialog boxes. The Surface High Speed toolpaths dialog box opens on the Toolpath Type page.
14. Select the Toolpath Type page, and then Roughing and Core Roughing.



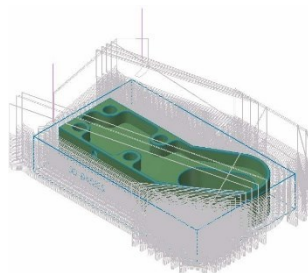
15. Select Tool from the Tree View pane.
16. Select the 10mm bull endmill with the 2mm corner radius showing in the tool list.
17. Select Cut Parameters from the Tree View pane. Set the tool containment and stock to leave values as indicated



18. Select Steep/Shallow from the Tree View pane. Set the parameters as shown.
19. Click the plus [+] in front of Cut Parameters if necessary, to expand the tree



20. Click OK in the Surface High Speed Toolpath dialog box to generate the core roughing toolpath.

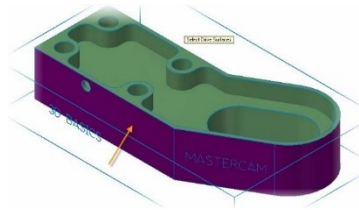


21. Turn off the toolpath display for the operation.
22. Save your part file.

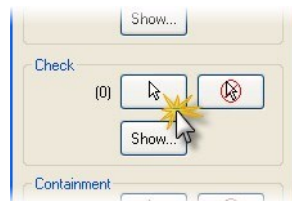
Activity 4.9 Perform Program for Finishing the Exterior

Procedure:

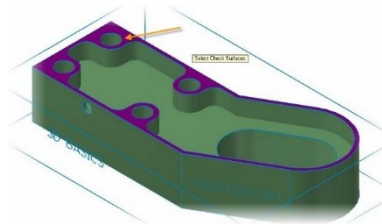
1. Choose Toolpaths, Surface Finish, Contour. Follow the prompt to select drive surfaces as shown



2. Press [Enter] or click the End Selection button to accept the selection.
3. Click the Select button in the Check Surfaces area of the Toolpath/Surfaceselection dialog box.



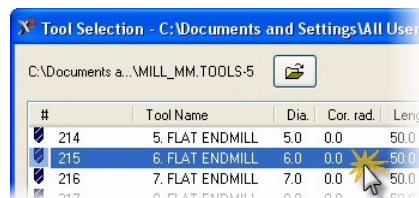
4. Follow the prompt to select check surfaces as shown.



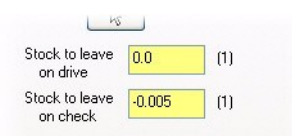
5. Press [Enter] or click the End Selection button to accept the selection.
6. Click OK on the Toolpath/Surface selection dialog box.
7. Click the Select library tool button

Select library tool...

8. Select the 6mm flat endmill and click OK

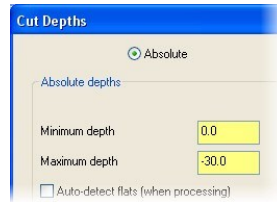


9. Click the Surface parameters tab and set the stock to leave values as shown.

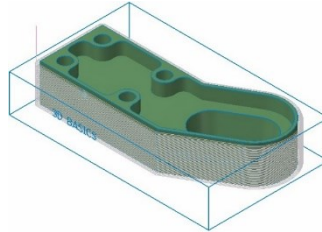


10. Click the Finish contour parameters tab.

11. Click the Total tolerance button.
12. Move the slider for Surface quality to the Best setting and click OK.
13. Click the Cut Depths button.
14. Set the parameters as shown and click OK



15. Click OK to generate the surface finish contour toolpath.



16. Turn off the toolpath display for the operation.
17. Save your part file.

Specification Sheet 4: Perform 3D Machining in Mastercam as Per Selected Mechanical 3D Model

Job Name: Perform 3D Machining in Mastercam as per selected Mechanical 3D Model.

Activity 4.1: Assigning a Machine Definition and Setting Stock for Machining

Activity 4.2: Perform program for Drilling Holes

Activity 4.3: Perform program for Roughing the Pocket

Activity 4.4: Roughing the Slot

Activity 4.5: Rest Roughing the Interior

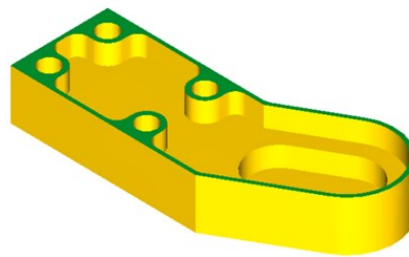
Activity 4.6: Finishing the Pocket Floor

Activity 4.7: Finishing the Slot

Activity 4.8: Roughing the Exterior

Activity 4.9: Finishing the Exterior

Conditions for the job: You must practice safe operation procedures at all times during the assessment. You will be required to demonstrate OSH competencies to Perform 3D Machining in Mastercam as per selected Mechanical 3D Model.



Required Personal Protective Equipment (PPE): Pre/person

Sl.	Name of the PPE	Specification	Unit	Quantity	Remarks
j.	Hand Gloves	Cotton	pair	01	
k.	Mask	N 95	Nos	01	
l.	Safety shoe	As required	pair	01	
m.	Safety goggles	Standard	Nos	01	
n.	Apron	Standard	Nos	01	

Required Tools and Equipment

Sl.	Name of tools and equipment	Specification	Unit	Quantity	Remarks
4.	Desktop computer	i5(8GB RAM)	No	01	

Review Of Competency

Below is yourself assessment rating for module “Applying CAM Software for Machining”

Assessment of performance Criteria	Yes	No
1. Computer-aided machining software is installed and verify the system requirement	<input type="checkbox"/>	<input type="checkbox"/>
2. Basic parameter is set of the CAM software	<input type="checkbox"/>	<input type="checkbox"/>
3. Menu functions are identified	<input type="checkbox"/>	<input type="checkbox"/>
4. Appropriate tools and equipment are used to produce a drawing as per job requirements	<input type="checkbox"/>	<input type="checkbox"/>
5. CAD files are imported as required	<input type="checkbox"/>	<input type="checkbox"/>
6. CAD models are created as required	<input type="checkbox"/>	<input type="checkbox"/>
7. The workpiece is verified and required machines are identified	<input type="checkbox"/>	<input type="checkbox"/>
8. Appropriate tool path sequence is selected for a model-specific machining operation	<input type="checkbox"/>	<input type="checkbox"/>
9. Appropriate tool diameter and length is selected to avoid chattering	<input type="checkbox"/>	<input type="checkbox"/>
10. Pre-machining operation is selected if required	<input type="checkbox"/>	<input type="checkbox"/>
11. Work holding & clamping is prepared to avoid tool clash	<input type="checkbox"/>	<input type="checkbox"/>
12. Appropriate cutting parameter is selected	<input type="checkbox"/>	<input type="checkbox"/>
13. Planes are identified	<input type="checkbox"/>	<input type="checkbox"/>
14. Machine definition and post-processor is selected	<input type="checkbox"/>	<input type="checkbox"/>
15. Origin & stock setup is defined	<input type="checkbox"/>	<input type="checkbox"/>
16. 2D tool paths are executed	<input type="checkbox"/>	<input type="checkbox"/>
17. Tapping and thread milling is executed	<input type="checkbox"/>	<input type="checkbox"/>
18. Tool paths are verified	<input type="checkbox"/>	<input type="checkbox"/>
19. G&M codes are generated	<input type="checkbox"/>	<input type="checkbox"/>
20. 3D tool path is prepared	<input type="checkbox"/>	<input type="checkbox"/>
21. level as per matching strategy is set	<input type="checkbox"/>	<input type="checkbox"/>
22. Surface is created as per requirement	<input type="checkbox"/>	<input type="checkbox"/>
23. Silhouette command is executed	<input type="checkbox"/>	<input type="checkbox"/>
24. Edge curve command is executed	<input type="checkbox"/>	<input type="checkbox"/>
25. Drive surface, containment region, check surface is selected	<input type="checkbox"/>	<input type="checkbox"/>
26. 3D rough & finishing tool path is executed	<input type="checkbox"/>	<input type="checkbox"/>
27. Stock model & rest milling is executed	<input type="checkbox"/>	<input type="checkbox"/>
28. 3D tool path is verified	<input type="checkbox"/>	<input type="checkbox"/>
29. G&M code is generated	<input type="checkbox"/>	<input type="checkbox"/>

now feel ready to undertake my formal competency assessment.

Signed:

Date:

Development of CBLM

The Competency based Learning Material (CBLM) of ‘**Applying CAM Software for Machining**’ (Occupation: **CNC Maching Centre Operation with CAD CAM, Level-4**) for National Skills Certificate is developed by NSDA with the assistance of SIMEC System Ltd., ECF Consultancy & SIMEC Institute of Technology JV (Joint Venture Firm) in the month of June, 2024 under the contract number of package SD-9B dated 15th January 2024.

SL No.	Name & Address	Designation	Contact Number
1	Rofiqun Nabi	Writer	01841604582
2	Uttam Kumar Das	Editor	01716220932
3	Engr. Md. Zuwel Parves	Co-Ordinator	01737278906
4	Engr. Md. Nazrul Islam	Reviewer	01711273708

Reference:

1. https://www.mastercam.dk/download/mastercam/tutorials/Basic_2D_Machining_Tutorial.pdf,
2. https://uwaterloo.ca/engineering-student-shops/sites/default/files/uploads/files/intro_to_mastercam_0.pdf,
3. https://caminstructor.com/files/www/TechTipVideos/MCAMX5_RefGuide.pdf,