Competency Based Learning Material (CBLM)

CNC Machining Centre Operation with CAD CAM

Level-4

Module: Performing Basic Multi-Axis Machining

Code: CBLM-OU-LE-CNCCDM-06-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.nsda.gov.bd.
National Skills Portal: http:\\skillsportal.gov.bd

This Competency Based Learning Materials (CBLM) on "**Performing Basic Multi-Axis 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.

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, Performing Basic Multi-Axis 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 you 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 though 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 ..... **i**

How to use this Competency Based Learning Material (CBLM) ..... **v**

Module Content **8**

Learning Outcome -1: Set Dynamic Work Offset **9**

Learning Experience 1: Set Dynamic Work Offset 10
Information Sheet 1: Set Dynamic Work Offset 11
Self-Check Sheet 1: Set Dynamic Work Offset 17
Answer Key - 1: Set Dynamic Work Offset 18
Job Sheet 1-1: Mark the 5 Axes on the Machine 19
Specification Sheet 1-1: Mark the 5 Axes on the Machine 20
Job Sheet 1-2: Mark the Machine Rotary Zero Point (MRZP) 21
Specification Sheet 1-2: Mark the Machine Rotary Zero Point (MRZP) 22
Job Sheet 1-3: Set the Dynamic Work Offset 23
Specification Sheet 1-3: Set the Dynamic Work Offset 25

Learning Outcome -2: Perform (3+2)/Positional Machining **26**

Learning Experience 2: Perform (3+2)/Positional Machining 28
Information Sheet 2: Perform (3+2)/Positional Machining 29
Self-Check Sheet 2: Perform (3+2)/Positional Machining 45
Answer Key - 2: Perform (3+2)/Positional Machining 46
Job Sheet 2-1: Complete the Stock Model 47
Specification Sheet 2-1: Complete The Stock Model 49
Job Sheet 2-2: (3+2)/for positional machining generate G&M code 50
Specification Sheet 2-3: (3+2)/For Positional Machining Generate G&M Code 52

Review of Competency **53**

Reference: **55**

Module Content

Unit of Competency	**Perform Basic Multi-Axis Machining**
Unit Code	**OU-LE-CNCCDM-06-L4-V1**
Module Title	Performing Basic Multi-Axis Machining
Module Descriptor	This module covers the skills, knowledge and attitudes required to perform basic multi-axis machining. It specifically includes setting dynamic work offset and performing (3+2)/positional machining.
Nominal Hours	**40** Hours
Learning Outcome	After completing the practice of the module, the trainees will be able to perform the following jobs: 1. Set dynamic work offset 2. Perform (3+2)/positional Machining

Assessment Criteria

1.1 5 axis is identified in the machine
1.2 Machine rotary zero points (MRZP) are identified
1.3 Dynamic work offset is set
2.1 Stock model is performed
2.2 Collision control is checked
2.3 G&M code is created for (3+2)/positional machining
2.4 Required plane is selected for (3+2)/positional machining in CAM software
2.5 Positional machining is executed using CAM software
2.6 Cutting in the machine is executed.

Learning Outcome -1: Set Dynamic Work Offset

Assessment Criteria	1. 5 axis is identified in the machine 2. Machine rotary zero points (MRZP) are identified 3. Dynamic work offset is set
Conditions and Resources	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 PPE
Contents	1. 5 axis ▪ X-axis ▪ Y-axis ▪ Z-axis ▪ B axis ▪ C axis 2. Machine Rotary Zero Point (MRZP) 3. Dynamic work offset
Job/Task/Activity	1. Mark the 5 axes on the machine 2. Mark the machine rotary zero point (MRZP). 3. Set the dynamic work offset
Training Method	1. Discussion 2. Presentation 3. Demonstration 4. Guided Practice 5. Individual Practice 6. Project Work 7. Problem Solving 8. Brainstorming Role Play
Assessment Method	1. Written Test 2. Demonstration 3. Oral questioning 4. Portfolio

Learning Experience 1: Set Dynamic Work Offset

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. Student will ask the instructor about the required learning materials	1. Instructor will provide the learning materials "Perform Basic Multi-Axis Machining"
2. Read the Information sheet and complete the Self Checks & Check answer sheets on "Set Dynamic Work Offset"	3. Read Information Sheet 1: Set Dynamic Work Offset Answer Self-check 1: Set Dynamic Work Offset Check your answer with Answer key 1: Set Dynamic Work Offset
2. Read the Job/Task Sheet and Specification Sheet and perform job/Task	4. Job/Task Sheet and Specification Sheet Job Sheet 1.1: Mark the 5 axes on the machine Specification Sheet 1.1: Mark the 5 axes on the machine Job Sheet 1.2: Mark the machine rotary zero point (MRZP). Specification Sheet 1.2: Mark the machine rotary zero point (MRZP). Job Sheet 1.3: Set the dynamic work offset Specification Sheet 1.3: Set the dynamic work offset

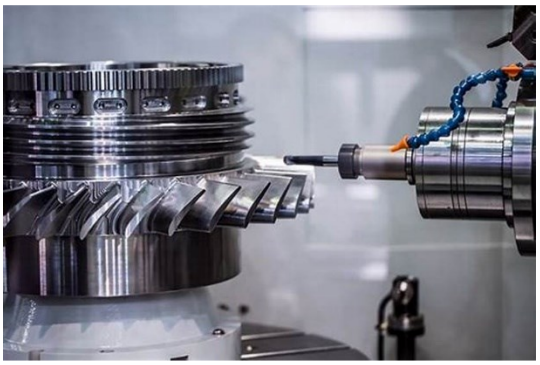
Information Sheet 1: Set Dynamic Work Offset

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

1.1 5 axis
1.2 Machine rotary zero points (MRZP)
1.3 Dynamic work offset

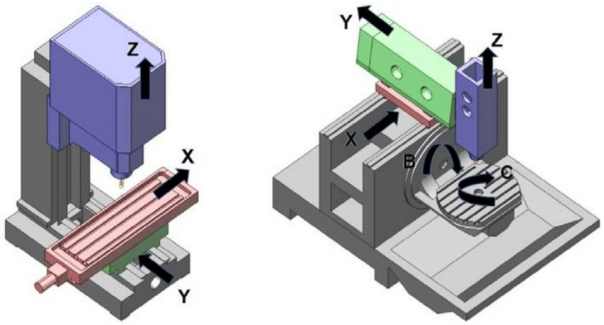
1.1 **5 axis**

When people talk about multi-axis CNC machining or multiple machining, 5-axis CNC machining is always mentioned in the advanced manufacturing methods. Full 5-axis CNC machining means a machine's processing ability to move a tool or a workpiece in five different axes simultaneously. 5-axis CNC machining is considered a more evolved and sophisticated version of the CNC machining process. shown down:



CNC 5-axis Machining.

As we all know, 5-axis CNC machining is achieved by 5-axis CNC machines. Nowadays 5-axis CNC machines are considered incredibly excellent for creating complex shapes with better surface finishing. People can learn much about 5-axis CNC machining once they understand how 5-axis CNC machines work in a CNC machine shop. In the 5-axis machining process, there are two additional axes as compared to the X, Y, and Z-axis. The two additional axes can be either A and B, A and C, or B and C Axis. They are involved to complete the machining process of highly complex shapes.



The extra two axes are utilized by the movement of a table on which the job material is fixed. Thus, the table can rotate in either way to 360 degrees around the cutting tool giving the machine accessibility to side cutting. Meanwhile, if we consider a single-axis movement like either in A, B, or C position the rotation is limited to 180 degrees. In the 5-axis machining process either there is swilling of the tool head against the still job material or there is the rotation or movement of a table at an angle that gives the table a tilt position. Depending upon the feature requirement, the table can tilt at different angles to achieve complex shapes. According to the size and weight of the object that is subjected to the required manufacturing product, the selection of either the swivelling or rotation of the table is decided. If the job is heavier, then it is subjected to swivel, otherwise, 5-axis machining goes with table tilt to perform the required machining process. There are also different varieties of 5-axis machining due to the types of 5-axis CNC machines. Now the question is how the other two axes are deployed in the 5-axis CNC machining process.

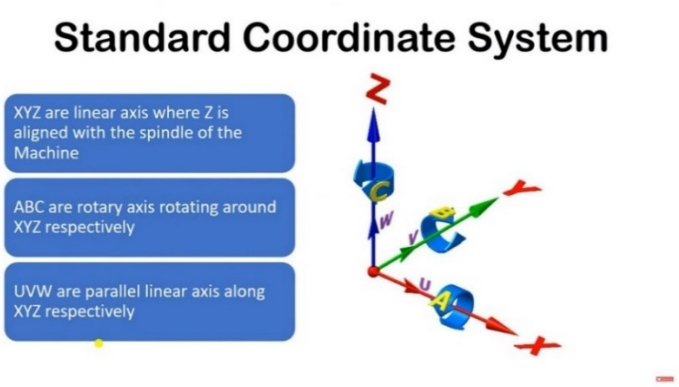
A. Benefits of 5-Axis Machining in Mastercam

a. **Machining complex shapes:** Ideal for components with curved or angled surfaces that would be difficult or impossible to machine with a 3-axis.
b. **Improved surface finish:** By constantly adjusting the tool angle, you can achieve smoother finishes.
c. **Reduced machining time:** Certain features can be machined in a single operation compared to multiple setups in 3-axis machining.
d. **Extended tool life:** Maintaining a proper cutting angle reduces stress on the tool, improving its lifespan.

B. X-axis

Diagram of in the standard coordinate system there are **X, Y**, and **Z** linear axes. There are additional rotary axes for each linear axis in a multiaccess environment.

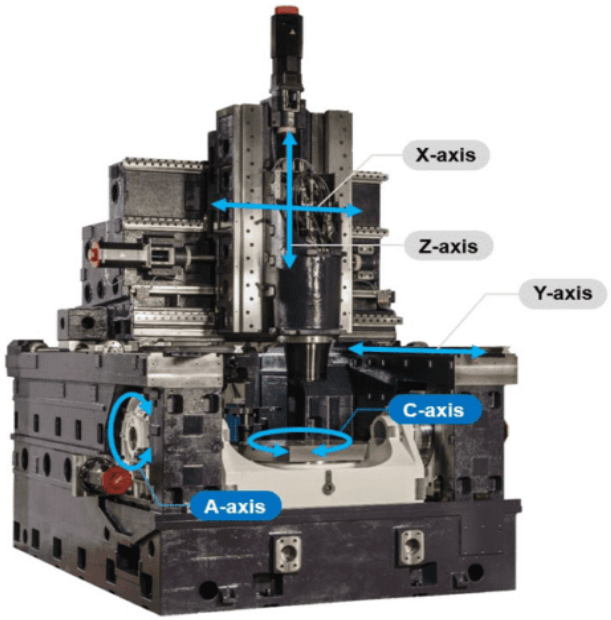
The X-axis represents horizontal movement across the length of the machine bed. Imagine a table moving left and right. This movement is fundamental for most CNC machines, including 5-axis variants. It provides the base for positioning the tool relative to the workpiece.



standard coordinate system

C. Y-axis

In a 5-axis CNC machine, the Y-axis remains the same as in a traditional 3-axis machine. It controls the **forward and backward** movement of the tool or the workpiece (depending on the machine configuration). Shown in Fig 2. It allows for linear movement in a straight line, typically along the table of the machine. Imagine the machine facing you. Moving the Y-axis forward would push the tool or workpiece away from you, and moving it backward would pull it toward you.



5-axis CNC machine

D. Z-axis

The Z-axis in a 5-axis CNC machine functions similarly to a 3-axis machine: it controls the **up-and-down movement** of the cutting tool. This vertical motion allows the tool to plunge into the workpiece and reach different depths for machining features. As shown in Fig. 2, the Z-axis does not operate independently. It is programmed to coordinate with the X, Y, and rotational axes to achieve the desired toolpath.

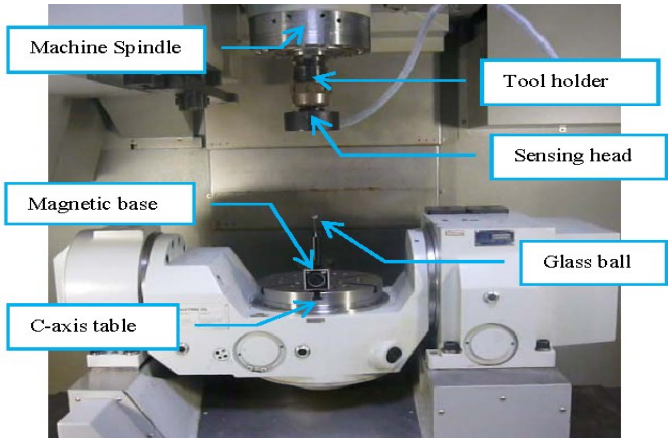
E. B axis

The B-axis is one of the two additional rotational axes that grant the machine increased flexibility for machining complex shapes.

As shown in Figure 2, the B-axis allows for **rotation around the Y-axis** of the machine. Imagine the Y-axis as a vertical pole, and the B-axis enables the tool or the table (depending on the machine configuration) to tilt sideways along this pole. This tilting motion provides access to features that would be difficult or impossible to reach with just linear X, Y, and Z movements.

F. C axis

The C-axis in a 5-axis CNC machine is one of the two rotary axes that provide additional machining flexibility compared to 3-axis machines.
The C-axis enables the **rotation of the workpiece around the Z-axis**. This Z-axis is the same one used for the up-and-down movement of the cutting tool. Imagine the workpiece mounted on a turntable - the C-axis controls the spinning of this turntable.



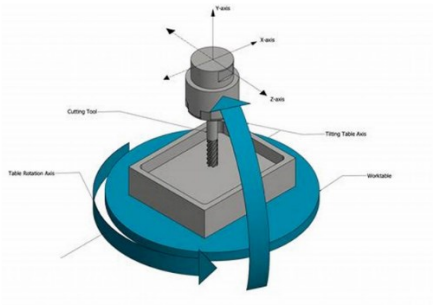
C-axis

By rotating the workpiece on the C-axis, the tool can access various features at different angles without physically repositioning the tool itself. This allows for the machining of complex, 3D geometries that would be difficult or impossible with just linear movements.

The **A** axis rotates about **X**. The **B** axis rotates about **Y**. Finally, the **C** axis rotates about **Z**. Additionally, **U**, **V**, and **W** are generally used for incremental axis positions to specify machining along parallel axes of **X**, **Y**, and **Z**.

1.2 Machine Rotary Zero Point (MRZP)

The Machine Rotary Zero Point (MRZP) in a 5-axis CNC machine is a critical concept for ensuring machining accuracy. It refers to the **exact centre of rotation** of the rotary axes. As shown in



Machine Rotary Zero Point

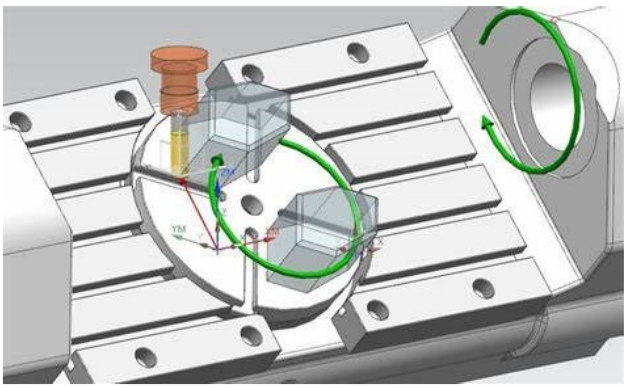
a. Setting The Machine Rotary Zero Point MRZP:

- There are various techniques for setting the MRZP, often involving the use of a **dial indicator**. The indicator is strategically positioned to measure the runout (eccentricity) of the rotary table or other rotating elements.
- By rotating the axes and taking measurements at different points, the technician can identify the centre of rotation and define that as the MRZP in the machine controller.
- Manufacturer instructions and specific machine configurations will determine the exact calibration procedures.

Understanding and accurately setting the Machine Rotary Zero Point (MRZP) is crucial for achieving precise results in 5-axis CNC machining. It establishes the foundation for all rotary movements and ensures the tool follows the programmed path relative to the centre of rotation. By following proper calibration procedures, you can guarantee the accuracy and repeatability of your 5-axis machining operations.

b. Dynamic Work Offset

DWO is a feature offered by some CNC machine controllers (like Haas with G254 code) that allows you to program toolpaths with more flexibility regarding the workpiece's initial position. Unlike traditional work offsets that require precise pre-alignment of the part, DWO offers some leeway. Dynamic work offset:



Dynamic work offset:

c. How Dynamic Work Offset (DWO) Works:

- **Set a Baseline Offset:** You define a single work offset in the CAM software and machine controller. This offset serves as a reference for the DWO function.
- **Machine handles adjustments:** When you activate DWO in the machine program, the controller automatically calculates the necessary adjustments based on the current tilt and rotary angles of the workpiece.
- **Toolpath follows adjusted path:** the machine adjusts the toolpath execution in real time to compensate for any discrepancies between the programmed offsets and the actual part position.

Dynamic Work Offset (DWO) is a valuable tool for simplifying toolpath creation and setup in 3+2 and 3+1 machining on 5-axis CNC machines. It offers more flexibility regarding the initial part position and reduces setup times. However, DWO has limitations and is not suitable for all 5-axis scenarios. Understanding the concept and its alternatives will help you choose the most appropriate approach for your specific machining needs.

Self-Check Sheet 1: Set Dynamic Work Offset

Questionnaire

1. What is 5-axis machining?
   Answer:

2. Why do we use 5-axis machine tools?
   Answer:

3. Write down the axes of 5-axis machine tools.
   Answer:

4. How do we set the Machine Rotary Zero Point?
   Answer:

5. Define the Dynamic work offset.
   Answer:

Answer Key - 1: Set Dynamic Work Offset

1. What is 5-axis machining?
   Answer: 5-axis CNC machining means a machine's processing ability to move a tool or a workpiece in five different axes simultaneously. 5-axis CNC machining is considered a more evolved and sophisticated version of the CNC machining process.

2. Why do we use 5-axis machine tools?
   Answer: Nowadays 5-axis CNC machines are considered incredibly excellent for creating complex shapes with better surface finishing.

3. Write down the axes of 5-axis machine tools.
   Answer: X axis, Y axis, Z axis, A axis, B axis, C Axis.

4. How do we set the Machine Rotary Zero Point?
   Answer: Setting the Machine Rotary Zero Point MRZP:
   There are various techniques for setting the MRZP, often involving the use of a dial indicator. The indicator is strategically positioned to measure the runout (eccentricity) of the rotary table or other rotating elements. > By rotating the axes and taking measurements at different points, the technician can identify the center of rotation and define that as the MRZP in the machine controller. > Manufacturer instructions and specific machine configurations will determine the exact calibration procedures.

5. Define the Dynamic work offset.
   Answer: Dynamic Work Offset (DWO) is a valuable tool for simplifying toolpath creation and setup in 3+2 and 3+1 machining on 5-axis CNC machines.

Job Sheet 1-1: Mark the 5 Axes on the Machine

Job name: Mark the 5 axes on the machine

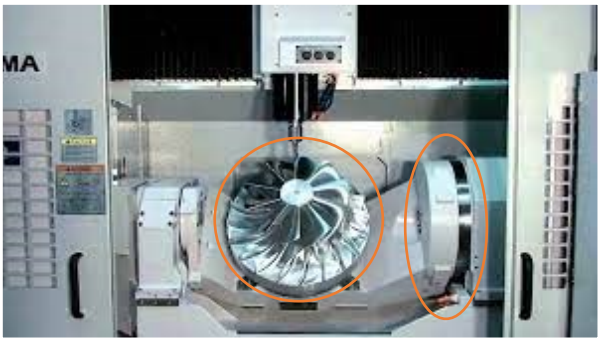
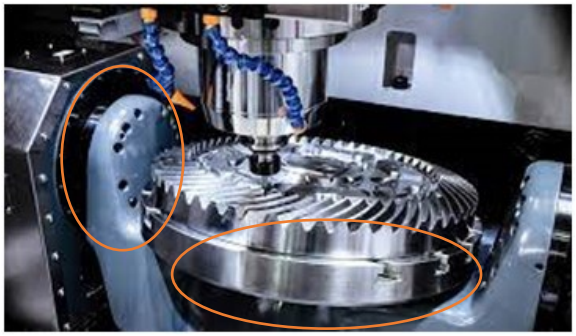
Procedure:

1. Wear appropriate PPE for the job
2. Read the Job sheet and Specification sheet provided.
3. Collect all needed materials, supplies, and equipment.
4. Identify the X(Horizontal Movement), Y(Depth Movement), and Z (Vertical Movement)Axes
5. Identify the Rotational Axes A-Axis (Rotation Around the X-Axis), B-Axis (Rotation Around the Y-Axis)
6. Verify the Axes Movement, X-Axis Movement , Y-Axis Movement and A-Axis Rotation
7. Move the machine in the X direction and ensure it travels horizontally left and right.
8. Check the display to confirm the axis movement matches the direction.
9. Move the machine in the Y direction and ensure it travels front to back.
10. Confirm on the machine's display that the Y movement is accurate.
11. Move the machine in the Z direction to ensure it travels up and down.
12. Verify the Z movement on the display matches the physical movement.
13. Rotate the A-axis and observe the part or tool rotating around the X-axis.
14. Ensure the display reflects the rotation accurately.
15. Rotate the B-axis to observe the part or tool rotating around the Y-axis.
16. Verify the rotation on the machine's display.
17. Step 4: Mark the Axes on the Machine
18. Clearly label the X, Y, and Z axes on the machine bed or relevant parts.
19. Use stickers, paint, or engravings to mark the directions.
20. Mark the A-axis near the rotational mechanism that rotates around the X-axis.
21. Mark the B-axis near the rotational mechanism that rotates around the Y-axis.
22. Perform Test and Calibrate
23. Perform test runs to move each axis individually and observe the behavior.
24. Make sure the movements correspond accurately to the control commands.
25. Clean workplace.
26. Store tools and equipment in a safe place.

Specification Sheet 1-1: Mark the 5 Axes on the Machine

Job name: Mark the 5 axes on the machine

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 Mark the 5 axes on the machine

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	

Job Sheet 1-2: Mark the Machine Rotary Zero Point (MRZP)

Procedure:

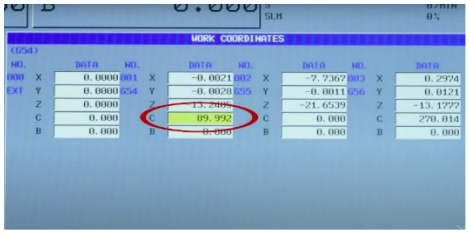
1. Wear appropriate PPE for the job
2. Read the Job sheet and Specification sheet provided.
3. Collect all needed materials, supplies, and equipment.
4. Prepare the tools, equipment, and materials for use.
5. Follow hygiene and safety requirements during the demonstration processes.
6. Clean the table for more precise the job.
7. Set up the Vise for mounting the job.

6

7

8. Use of a dial indicator is strategically positioned to measure the runout (eccentricity) of the rotary table or other rotating elements.
9. take measurements at different points, and identify the centre of rotation and define that as the MRZP in the machine controller.

8



9

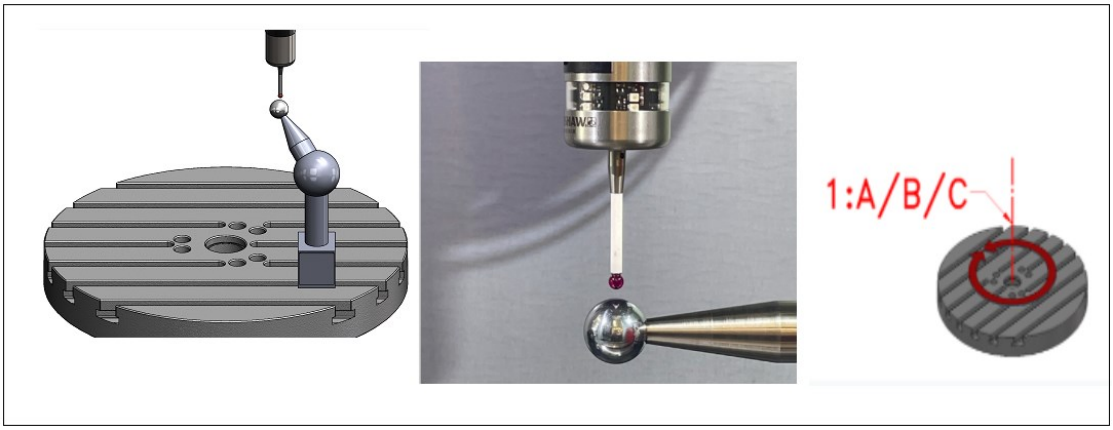
10. Follow Manufacturer instructions and specific machine configurations will determine the exact calibration procedures.
11. Apply the relevant machine selection process.
12. Follow the process of machine rotary zero point.
13. Machine Setup: Ensure the machine is in a stable and level condition. Secure a suitable workpiece or fixture on the rotary table.
14. Reference Point Establishment: Choose a reference point on the workpiece or fixture. This point will be used as a basis for measurements.
15. Data Collection: Use a dial indicator or other precision measurement tool to measure the distance between the reference point and the rotary axis at multiple positions.
16. Calculation: Analyze the collected data to determine the center of rotation. This often involves mathematical calculations or using specialized software.
17. MRZP Offset Input: Input the calculated MRZP values into the machine control's offset parameters. These values will compensate for the rotary axis's position relative to the linear axes.
18. Clean workplace.
19. Store tools and equipment in a safe place

Specification Sheet 1-2: Mark the Machine Rotary Zero Point (MRZP)

Job Name: Mark the machine rotary zero point (MRZP).

Conditions for the job: You must practice safe operation procedures at all times during the assessment. You will be required to demonstrate OSH competencies **Mark the machine rotary zero point (MRZP).**

Diagram



Required Personal Protective Equipment (PPE): Per/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's

Sl.	Name of tools and equipment	Specification	Unit	Quantity	Remarks
1	Probe Tool		No	01	
2	Edge Finder		No	01	
3	Tool Holder and Collect		No	01	

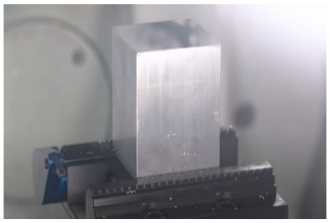
Required Materials

Sl.	Name of materials	Specification	Unit	Quantity	Remarks
1	Aluminum Bar	100 x 100 x 60	Pcs	01	

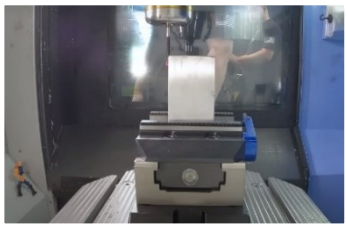
Job Sheet 1-3: Set the Dynamic Work Offset

Procedure:

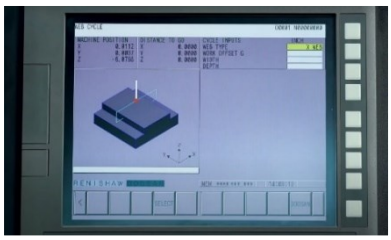
1. Wear appropriate PPE for the job
2. Read the Job sheet and Specification sheet provided.
3. Collect all needed materials, supplies, and equipment.
4. Prepare the tools, equipment, and materials for use.
5. Follow hygiene and safety requirements during the demonstration processes.
6. Apply the relevant machine selection.
7. Set up a job: First, you need to set up a job on Vise.
8. Set single work offset in the CAM software and machine controller. This offset serves as a reference for the DWO function.
9. use the prove equipment to take the values of offsets.
10. Perform Machine Handles Adjustments to activate DWO in the machine program, the controller automatically calculates the necessary adjustments based on the current tilt and rotary angles of the workpiece.
11.



7



9



10

12. Adjusts the toolpath execution in real time to compensate for any discrepancies between the programmed offsets and the actual part position.
13. Define Work Coordinate System (WCS): Establish a WCS at a convenient location on the workpiece or fixture. This is typically the part's zero point.
14. Measure Tool Length Offset (TLO): Determine the distance between the tool tip and the machine's reference point. This information is crucial for accurate tool positioning.
15. Set Up Rotary Axis: Ensure the rotary axis is calibrated and its zero point is accurately defined.
16. Activate DWO(Dynamic work offset) Function: Enable the DWO(Dynamic work offset) function in your CNC control's settings. This is usually done through specific parameters or codes.
17. Establish Reference Points: Define reference points on the workpiece that correspond to different rotary axis positions. These points will be used to calculate the DWO (Dynamic work offset)values.
18. Measure and Calculate DWO(Dynamic work offset): Use a probe or other measurement tool to determine the position of the reference points relative to the

WCS for different rotary axis angles. Calculate the DWO values based on these measurements. Many CNC controls have built-in functions to assist with this calculation.

19. Enter the calculated DWO (Dynamic work offset) values into the CNC control's DWO (Dynamic work offset) parameters. These values will compensate for the workpiece's position changes as the rotary axis rotates.
20. Perform test cuts to verify the accuracy of the DWO (Dynamic work offset)values. Make any necessary adjustments.
21. Clean workplace.
22. Store tools and equipment in a safe place.

Specification Sheet 1-3: Set the Dynamic Work Offset

Job Name: Set the dynamic work offset

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 the dynamic work offset**

Diagram


Required Personal Protective Equipment (PPE): Per/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	Probe Tool		No	01	
2	Edge Finder		No	01	
3	Tool Holder and Collect		No	01	

Required Materials

Sl.	Name of materials	Specification	Unit	Quantity	Remarks
1	Aluminum Bar	100 x 100 x 60	Pcs	01	

Learning Outcome -2: Perform (3+2)/Positional Machining

Assessment Criteria	1. Stock model is performed 2. Collision control is checked 3. G&M code is created for (3+2)/positional machining 4. Required plane is selected for (3+2)/positional machining in CAM software 5. Positional machining is executed using CAM software 6. Cutting in the machine is executed.
Conditions and Resources	1. Workplace or Simulated Workplace 2. CBLM 3. Handout 4. **L**aptop 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	1. Stock model ▪ Consolidation of stocks ▪ Name of stock ▪ Stock color ▪ Material ▪ Additional offset ▪ Source operations ▪ Stock Compare 2. Collision Control ▪ Checking flute, solder, shank, holder

	▪ Strategy and parameters ▪ Check geometry ▪ Check clearance 3. Requirement to test collision control 4. (3+2)/Positional Machining 5. Planes required for (3+2)/positional machining in CAM software ▪ WCS plane ▪ C plane ▪ T plane ▪ Top ▪ Front ▪ Right ▪ Left ▪ Back ▪ Bottom ▪ Customized 6. Use of CAM software ▪ Positional machining ▪ Strategy and parameters ▪ Check geometry ▪ Check clearance 7. Positional machining
Job/Task/Activity	1. Complete the stock model 2. Check Collision Control 3. Create G&M code for (3+2)/positional machining 4. Select required plane for (3+2)/positional machining in CAM software 5. Perform positional machining using CAM software
Training Method	1. Discussion 2. Presentation 3. Demonstration 4. Guided Practice 5. Individual Practice 6. Project Work 7. Problem Solving 8. Brainstorming
Assessment Method	1. Written Test 2. Demonstration 3. Oral questioning 4. Portfolio

Learning Experience 2: Perform (3+2)/Positional Machining

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
Trainee will ask the instructor about the learning materials	1. Instructor will provide the learning materials ‘Perform (3+2)/positional Machining
Read the Information sheet and complete the Self Checks & Check answer sheets on “Perform (3+2)/positional Machining”	2. Read Information sheet 1: Perform (3+2)/positional Machining 3. Answer Self-check 1: Perform (3+2)/positional Machining 4. Check your answer with Answer key 1: Perform (3+2)/positional Machining
Read the Job/Task Sheet and Specification Sheet and perform job/Task	5. Job/Task Sheet and Specification Sheet Job Sheet 2-1: Complete the stock model Specification Sheet 2-1: Complete the stock model. Job Sheet 2-2: Create G&M Code for (3+2)/ positional machining Specification Sheet 2-2: Create G&M Code for (3+2)/ positional machining

Information Sheet 2: Perform (3+2)/Positional Machining

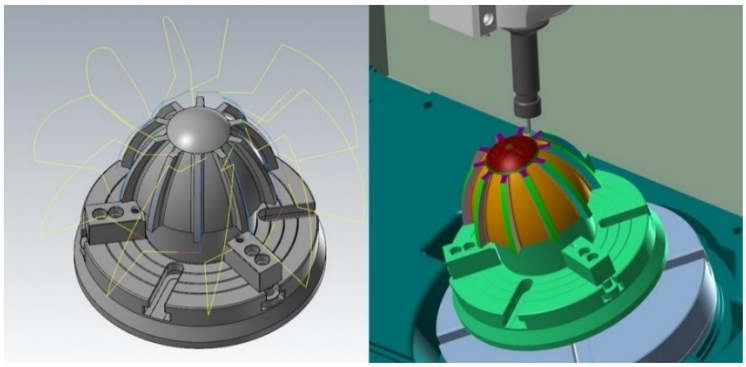
Learning Objective: After completion of this information sheet, the learners will be able to explain, define and interpret the following contents

2.1 Stock model
2.2 Collision Control
2.3 Requirement to test collision control
2.4 (3+2)/Positional Machining
2.5 Planes required for (3+2)/positional machining in CAM software
2.6 Use of CAM software
2.7 Positional machining

2.1 Stock Model

Stock typically refers to the raw material used to create the final part. This could refer to a digital 3D model that represents the raw material (stock) used for machining a part on a 5-axis CNC machine. This model would typically be created in CAD (Computer-Aided Design) software and imported into the CAM (Computer-Aided Manufacturing) software.

Stock model might refer to a pre-defined model of a particular 5-axis CNC machine within the CAM software. This model is not related to the raw material but provides information about the machine's capabilities for collision avoidance and reachability checks during toolpath generation.



Stock model part on a 5-axis CNC machine

The stock model is crucial for the CAM programmer to define the starting point for machining. It allows them to visualize the material removal process and ensure the toolpaths stay within the boundaries of the stock.

a. Consolidation of stocks

consolidation of stock likely refers to strategies for optimizing material usage and minimizing waste during the machining process.

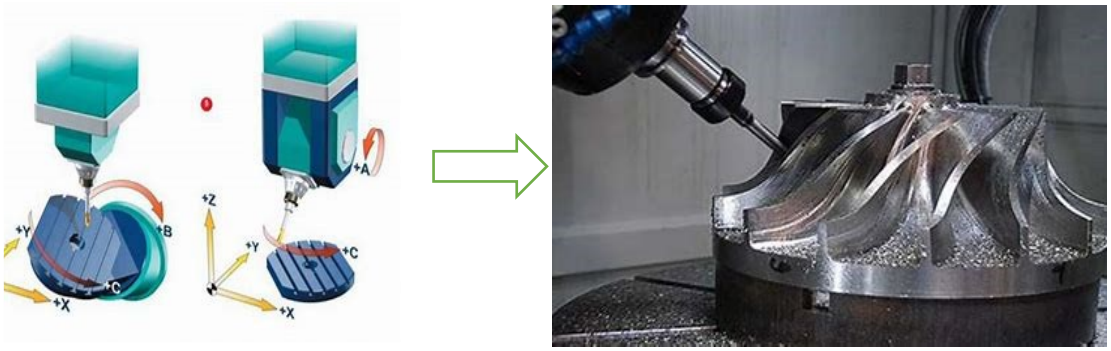


Figure 6: Consolidation of stocks working axes

Overall, consolidation in 5-axis machining can refer to either minimizing leftover material waste or combining multiple machining operations into a single setup. Both approaches aim to optimize the 5-axis machining process for efficiency, reduced costs, and improved production outcomes.

b. Name of stock

Mastercam for 5-axis machining, focus on defining the stock's geometry accurately using the stock model. You can then assign a clear name to your stock model for better project management.

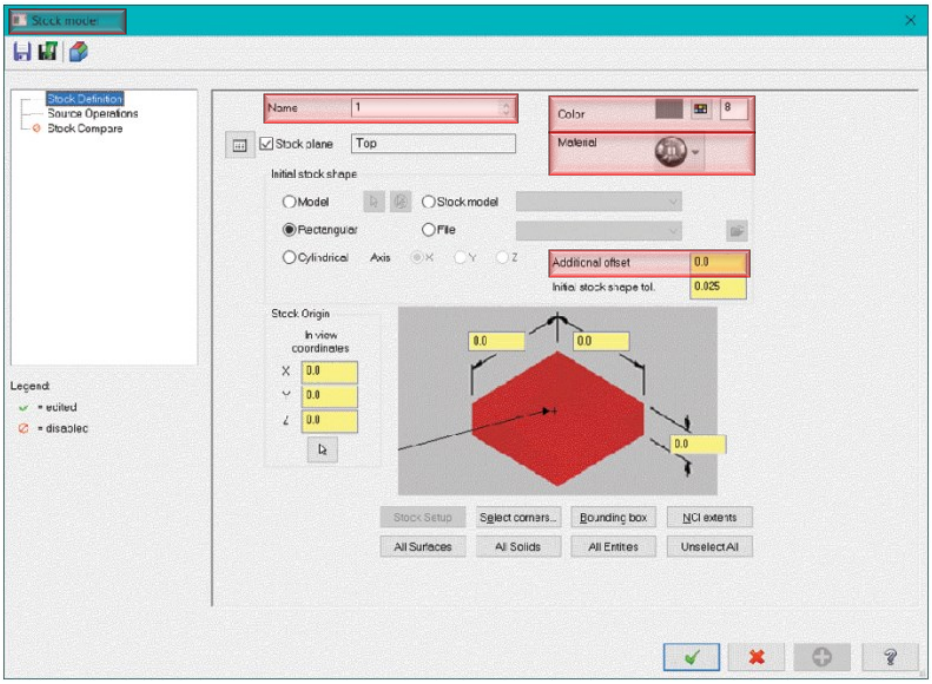


Figure 7: Name of stock, Stock colour, Material, Additional offset.

c. Stock colour

This color-coding system provides valuable insights into the effectiveness of your toolpaths and helps identify potential issues. The stock model itself doesn't have a

designated colour in Mastercam. You might be able to customize the display colour for better visualization, As shown in Figure 7.

Stock Compare in Verify utilizes colours to represent the remaining material, excessive removal, and insufficient removal after toolpath simulation.

d. Material

The stock material can be various metals, plastics, wood, or other machinable materials depending on the desired final part. Common examples of metalworking include aluminium, steel, titanium, etc.

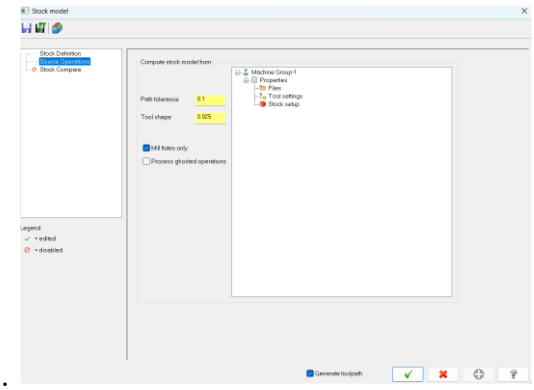
e. Additional offset

Most stock materials aren't perfectly perfect in terms of geometry. They might have slight variations in size, flatness, or surface finish. As shown in Figure 7, When Mastercam creates the stock model based on your dimensions, it assumes a perfect geometric shape.

The initial shape tolerance allows Mastercam to account for these potential real-world imperfections by adding a small offset to the stock model's geometry. This ensures the toolpaths are generated considering these minor variations and avoids gouging into the actual stock material.

f. Source operations

This section within the Stock Model properties defines which existing toolpath operations contribute to the initial stock shape, as shown in Figure 8. Essentially, you're telling Mastercam which previous machining processes have already been performed on the raw material before it reaches the stage represented by this Stock Model.

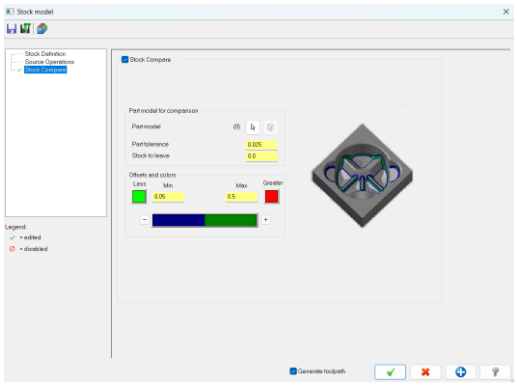


Source Operations

By effectively utilizing Source Operations in your Mastercam Stock Model, you can create a more accurate representation of the starting point for your machining process. This leads to improved toolpath simulation, better collision avoidance checks, and ultimately, more efficient, and reliable 5-axis machining.

g. Stock Compare

Mastercam Verify is a module that simulates the machining process based on your programmed toolpaths,



Stock Compare.

Within Verify, the "Stock Compare" function comes into play. It compares the virtual machined part (created by simulating the toolpaths on the stock model) with the original stock model.

By effectively utilizing Stock Compare with your Mastercam stock model, you can gain valuable insights into your 5-axis toolpaths, identify potential issues, and ensure accurate and efficient machining of your parts

2.2. Collision Control

Collision control is crucial for any CNC (Computer Numerical Control) machine, particularly for 5-axis machines due to their increased complexity and freedom of movement. Here are several strategies and technologies employed for collision control on 5-axis CNC machines.

A. Checking flute, solder, shank, holder

a. **Flute:** Yes, Collision Control can identify collisions involving the flute of the cutting tool. The flute is the part of the tool that creates the cutting edge. Collisions with the machine or fixture could damage the flute and affect machining performance.

b. **Solder:** In some cases, Collision Control might account for solder connections on the tool shank, especially if the toolholder has a specific design that requires checking clearance around the solder area. However, it's not as common as checking the flute or shank itself.

c. **Shank:** Yes, Collision Control definitively checks for collisions involving the shank of the tool. The shank is the cylindrical body of the tool that fits into the tool holder. Collisions with the machine or fixture could damage the shank and cause runout or tool breakage.

d. **Holder:** Yes, Collision Control also checks for collisions involving the tool holder. The tool holder grips the shank of the tool and connects it to the machine

spindle. Collisions between the holder and the machine components could damage the holder or restrict its movement.

Strategy and Parameters

Figure 11: Strategy and Parameters in Collision Control.

B. Collision Control Strategies

These strategies define the overall approach Mastercam takes when checking for collisions:

a. **Full Toolpath Simulation:** This is the most comprehensive strategy where Mastercam simulates the entire toolpath movement along the programmed path, checking for collisions at every point. This offers the most thorough collision detection but can be time-consuming for complex toolpaths.

b. **Critical Points Check:** This strategy focuses on checking collisions at critical points along the toolpath, such as start/end points, transitions between rapid movements and cutting motions, and areas with tight clearances. This is a good balance between accuracy and speed, suitable for most machining scenarios.

c. **Area Checking:** This strategy allows you to define specific areas within the workspace where you want to focus the collision check. This can be useful for situations where you're concerned about collisions in a particular region due to complex toolpath manoeuvres or workpiece geometry.

C. Collision control parameters

These parameters allow you to fine-tune how Mastercam identifies and reports potential collisions:

a. **Minimum clearance:** This defines the minimum acceptable distance between the tool and the machine components. Values are typically set in millimetres or inches. A smaller clearance value increases collision detection sensitivity but might flag harmless near misses.

b. **Stock model consideration:** You can choose whether to include the stock model (raw material) in the collision check. This can be helpful for identifying potential

gouging of the stock due to tool holder or fixture interference during approach/retract motions.

c. **Machine component selection:** Some advanced Mastercam versions allow you to selectively enable/disable collision checks for specific machine components. This can be useful for focusing on critical areas or excluding components that are not relevant to the current machining operation.
d. **Collision reporting:** Mastercam offers various options for reporting collision events. You can choose to display visual alerts within the 3D model, generate text reports with collision details, or even pause the simulation upon encountering a collision.

Collision control is crucial for any CNC (Computer Numerical Control) machine, particularly for 5-axis machines due to their increased complexity and freedom of movement. Here are several strategies and technologies employed for collision control on 5-axis CNC machines.

e. **Toolpath Simulation:** One of the primary methods for collision control is to simulate the toolpath before actual machining. This involves using specialized software that generates a virtual model of the machining process, allowing operators to identify potential collisions between the tool, the workpiece, and the machine itself.

f. **Collision Detection Systems:** Many modern CNC machines, including 5-axis ones, are equipped with collision detection systems. These systems use sensors

and software algorithms to monitor the machine's movements in real-time and detect any unexpected collisions. Once detected, the machine can automatically stop or adjust its movements to avoid damage.

g. **Dynamic Workpiece and Tool Modelling:** Advanced CNC systems incorporate dynamic models of both the workpiece and the cutting tool. These models take into account factors such as tool deflection, workpiece deformation, and machine kinematics to predict and prevent collisions during machining operations.
h. **Safe zones and limits:** Operators can define safe zones and limits within the CNC machine's software to prevent the tool or workpiece from approaching critical areas where collisions are likely to occur. These limits can include physical boundaries of the machine, as well as clearance distances from fixtures or clamps.
i. **Real-time monitoring and feedback:** Some CNC machines feature real-time monitoring and feedback systems that continuously track the machine's performance and detect any deviations from the programmed toolpath. This allows operators to intervene quickly in the event of a collision or other unexpected event.
j. **Automatic tool length measurement:** Accurate measurement of tool length is essential for collision avoidance. Many CNC machines are equipped with automatic tool length measurement systems that ensure the tool is properly positioned and can adjust the machining parameters accordingly.
k. **Adaptive control systems:** Adaptive control systems use feedback from sensors and actuators to adjust machining parameters in real-time, optimizing performance and avoiding collisions. These systems can dynamically adjust feed rates, spindle speeds, and toolpaths based on changing conditions during the machining process.
l. **Machine kinematics optimization:** Optimizing the machine's kinematics, such as acceleration and deceleration profiles, can help minimize the risk of collisions during rapid movements between machining operations.
m. By implementing these strategies and technologies, operators can effectively control collisions on 5-axis CNC machines, ensuring safe and efficient machining operations.

D. Check Geometry

Collision Control in Mastercam inherently checks the geometry of various elements to identify potential collisions during 5-axis machining.

E. Collision-Checked Geometries:

a. Tool geometry

This includes the complete 3D definition of the cutting tool you'll be using. Mastercam considers:

- **Flute diameter:** This is the diameter of the cutting portion of the tool that creates the machined surface. Collisions involving the flute can damage the cutting edge and affect machining performance.

- **Shank diameter:** This is the cylindrical body of the tool that fits into the tool holder. Collisions with the shank can damage the tool and cause runout or breakage.
- **Overall tool length:** The complete length of the tool from tip to end is considered to ensure clearance throughout the toolpath.
- **Corner Radius (optional):** If applicable, the corner radius of the tool (the rounded edge where the flute meets the shank) might also be factored in for more precise collision detection.

b. Toolholder geometry:

The 3D model of the tool holder you've selected for the machining operation is also included in the collision check. Mastercam considers:

- **Overall shape and size:** The entire geometry of the tool holder is checked for collisions with the machine structure and other components.
- **Toolholder pocket:** The specific area within the tool holder that grips the tool shank is crucial to ensure proper clearance and avoid interference during tool rotation.

F. Machine geometry:

A virtual 3D model of your specific 5-axis CNC machine is used for collision detection. This model includes:

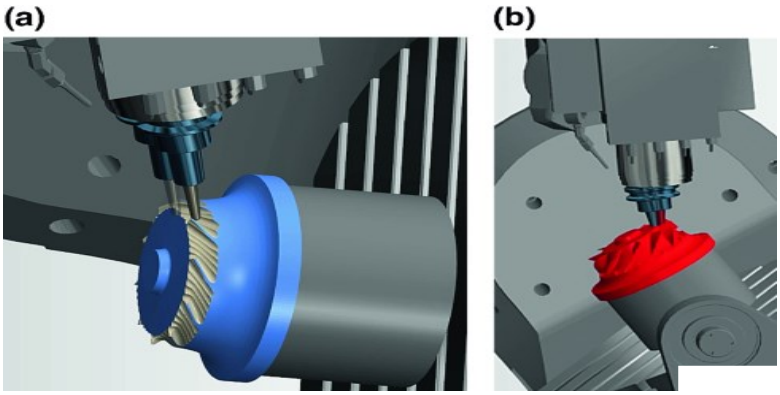
a. Workspace limitations: The maximum travel ranges of the machine's linear axes and rotary axes are considered to define the boundaries within which the tool can safely move.

b. Machine structure: The physical components of the machine, such as the table, column, rotary heads, and other elements, are included to check for potential collisions with the tool or tool holder.

c. Specific components (optional): Some advanced Mastercam versions allow you to enable/disable collision checks for specific machine components. This can be helpful for focusing on critical areas or excluding components not relevant to the current machining operation.

G. Stock model geometry (optional):

a. You can optionally include the stock model (raw material) in the collision check. This is beneficial for identifying potential gouging of the stock material due to tool holder or fixture interference during the approach/retract motions of the toolpath.



Stock Model Geometry (Optional)

H. Check clearance

To determine a collision, Mastercam compares the geometries of various elements and evaluates the **minimum distance** between them throughout the toolpath. This minimum distance is referred to as **clearance**.

2.3 Requirement to Test Collision Control

Testing collision control for a 5-axis machine is crucial to ensure the safety, accuracy, and efficiency of machining operations. Here's a structured approach to testing collision control for such machines:

a. Virtual Simulation Testing

- Utilize specialized software for virtual simulation of machining operations.
- Import CAD models of the workpiece, cutting tools, and machine components.
- Generate toolpaths and simulate machining operations.
- Analyse the simulation results to identify any potential collisions between the tool, workpiece, and machine components.

b. Physical Testing:

- Conduct physical testing with mock setups or sacrificial materials.
- Use dummy workpieces and cutting tools to perform test runs.
- Gradually increase cutting parameters to verify collision control effectiveness under different operating conditions.
- Employ sensors or monitoring systems to detect collisions during physical testing.

c. Boundary and Limit Testing:

- Define safe zones and limits within the CNC machine's software.
- Perform test runs near these boundaries to ensure that the machine halts or adjusts its movements appropriately when approaching critical areas.
- Test the effectiveness of boundary and limit settings under various machining scenarios**.**

d. Dynamic Modelling Validation:

- Validate dynamic models of the workpiece, tool, and machine components used for collision prediction.
- Compare simulation results with physical testing data to ensure accuracy and reliability.
- Make adjustments to dynamic models based on discrepancies observed during validation.

e. Real-time Monitoring and Feedback Testing:

- Evaluate real-time monitoring and feedback systems during actual machining operations.
- Monitor the machine's performance and response to unexpected events or deviations from the programmed toolpath.
- Test the responsiveness of the system in detecting and mitigating collisions in real-time.

f. Adaptive Control System Testing:

- Test adaptive control systems by intentionally introducing disturbances or changes in machining conditions.
- Assess the system's ability to dynamically adjust machining parameters to avoid collisions while maintaining machining efficiency.
- Verify the effectiveness of adaptive control algorithms through both simulation and physical testing.

g. Documentation and Reporting:

- Document test procedures, results, and any observed issues or improvements needed.
- Generate reports summarizing the effectiveness of collision control measures and recommendations for enhancements.
- Maintain comprehensive records for future reference and compliance purposes.

By following this systematic approach to testing collision control for 5-axis CNC machines, manufacturers can ensure the safety and reliability of their machining processes while maximizing productivity and minimizing downtime due to collisions.

2.4. (3+2)/Positional Machining

Positional machining in the context of 5-axis machining refers to the ability to precisely control the position and orientation of the cutting tool relative to the workpiece. This capability is essential for manufacturing complex parts with intricate geometries and surfaces. Here's a breakdown of positional machining for 5-axis machining:

a. Positional Control

3+2 Machining: In 3+2 machining, the machine operates as a 3-axis machine with the added capability of positioning the workpiece in two additional axes before machining begins. This approach allows for complex parts to be machined with improved accuracy and surface finish by orienting the workpiece optimally for each operation.

Benefits

- Enhanced Surface Finish: Positioning the workpiece optimally allows the cutting tool to maintain a consistent angle of engagement with the surface, resulting in improved surface finish.

- Reduced Setup Time: By positioning the workpiece in multiple orientations in a single setup, 3+2 machining reduces the need for manual repositioning and alignment, thereby reducing setup time.

  Increased Accuracy: Precise control over the position and orientation of the cutting tool enables machining of complex geometries with high accuracy, minimizing errors and scrap.

2.5. Planes Required For (3+2)/Positional Machining In CAM Software

In (3+2) positional machining, CAM software is utilized to generate toolpaths that optimize the positioning of the workpiece and tool to achieve complex geometries and surfaces. To effectively program these operations, specific planes are often utilized.

Here are the key planes typically required for (3+2) positional machining in CAM software:

a. Machine Coordinate System (MCS):

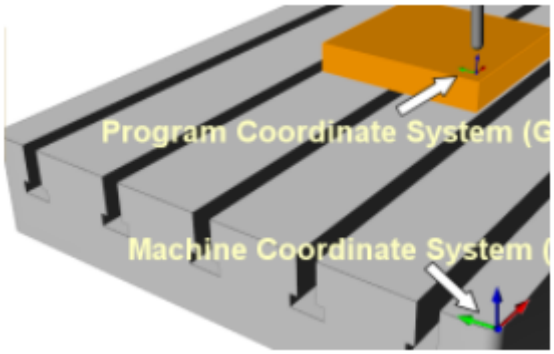
The MCS defines the reference frame of the CNC machine. It establishes the origin and orientation of the machine's axes. All toolpaths are generated relative to this coordinate system.

b. Workpiece Coordinate System (WCS):

The WCS is used to define the reference frame of the workpiece. It specifies the origin and orientation of the workpiece within the machine's workspace. Aligning the WCS with the desired features of the workpiece facilitates accurate machining.

c. Tool Coordinate System (TCS):

The TCS establishes the reference frame for the cutting tool. It defines the tool's position and orientation relative to the workpiece. Aligning the TCS with specific features of the workpiece allows for precise tool engagement.



d. Part Zero Plane (P0 Plane):

Also known as the reference plane, the P0 plane is a flat surface on the workpiece used as the reference for establishing the WCS. It typically corresponds to a specific feature or surface of the workpiece that serves as the primary datum for machining operations.

e. Toolpath Plane:

The toolpath plane is the surface or plane along which the toolpath is generated. It represents the cutting plane where the tool engages with the workpiece. The orientation of the toolpath plane determines the direction of tool movement relative to the workpiece.

f. Safety Plane:

The safety plane is a virtual plane that defines the maximum allowable distance between the tool and the workpiece to prevent collisions. It ensures that the tool remains within a safe operating envelope throughout the machining process.

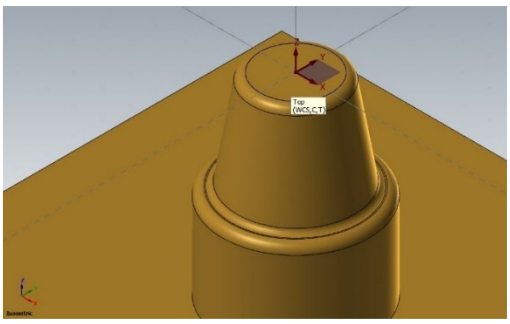
g. Indexing Planes:

Indexing planes are additional planes used for positioning the workpiece during (3+2) positional machining. They enable rotations or tilting of the workpiece to access different machining features or angles.

By utilizing these planes effectively within the CAM software, programmers can create optimized toolpaths for (3+2) positional machining operations. Proper alignment and orientation of these planes ensure accurate machining and efficient utilization of the CNC machine's capabilities.

h. WCS plane

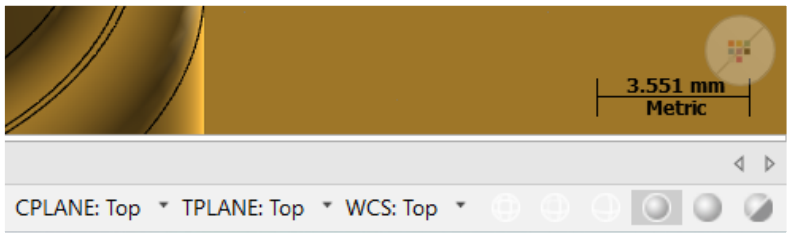
The WCS (Work Coordinate System) plane is a fundamental concept that defines the origin and orientation for your entire machining project. It acts as the reference point for all toolpath movements and positioning within your 3D model. As shown WCS plane in



WCS Plane, C Plane, T Plane.

i. C plane

The C Plane (Construction Plane) is a crucial element that serves as a temporary 2D reference plane within your 3D machining environment. It provides a convenient platform for defining and creating 2D geometry elements that will eventually be used for toolpath generation.



CPlane, TPlane, WCS Plane.

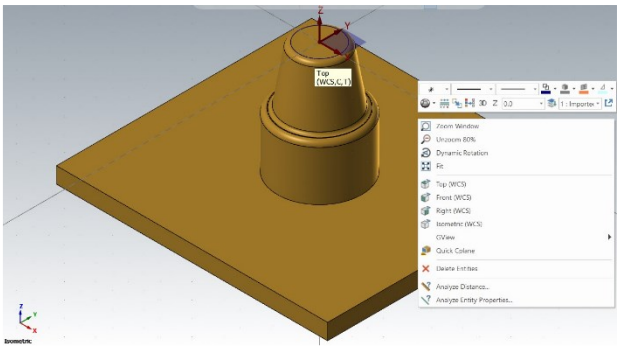
you can significantly enhance your 2D geometry creation workflow and achieve greater control over your toolpath generation for successful CNC machining projects.

j. T plane

T Plane (Tool Plane) is a concept specifically related to 5-axis machining functionalities. It is a virtual plane that defines the normal direction (perpendicular orientation) for the cutting tool during machining operations,

k. Top

Top view is a fundamental and versatile tool for viewing, analysing, and creating geometry within your 3D machining projects. By effectively utilizing the Top view along with other functionalities like C Planes and toolpath verification, you can streamline your design and machining workflow in Mastercam.



Top, Front, Right view.

l. Front

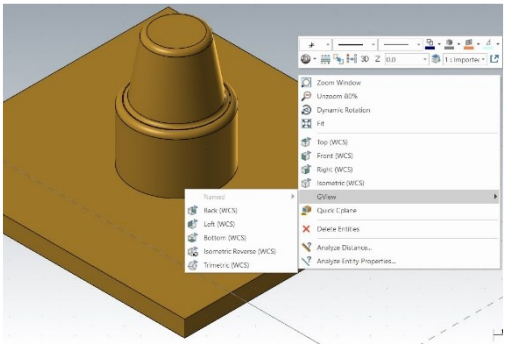
Front view displays your model as if you're looking directly at it from a positive X-axis direction. Imagine yourself standing in front of the model, facing its positive X-axis side.

m. Right

Right view is a pre-defined view aligned with the right side that displays your 3D model from a specific perspective within the graphics window,

n. Left

Left view in Mastercam is a pre-defined view aligned with the left side that displays your 3D model from a specific perspective within the graphics window.



Left, Back, Bottom view.

o. Back

Mastercam offers the flexibility to achieve a "Back" perspective of your model through rotation tools and CPlanes. This allows you to effectively analyse and create geometry from various angles within your 3D machining projects. As shown the option in

p. Bottom

Bottom view, you can always mentally consider the opposite direction of the Top view as the "Bottom" for better visualization.

q. Customized

Custom views allow you to create viewpoints tailored to focus on specific features, machining regions, or areas of interest within your model. By saving frequently used custom views, you can quickly switch between them without having to manually re-position the model every time. You can share custom views with colleagues or clients to provide specific perspectives of the model during project discussions.

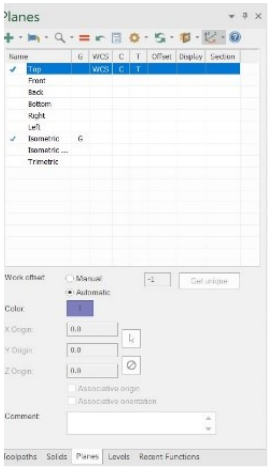


Figure 19: Customized Plane.

While custom views offer flexibility, some advanced functionalities like clipping planes (focusing on a specific section) might require additional tools depending on your Mastercam version.

2.6 Use of CAM Software

CAM software (Computer-Aided Manufacturing) is a crucial tool used in manufacturing to bridge the gap between design and production. Here's a breakdown of its applications:

a. **Prototyping and machining of complex parts:** CAM software empowers the creation of intricate 3D models in CAD software and translates them into precise toolpaths for machining complex parts with high accuracy and repeatability.

b. **Improved machining efficiency:** By optimizing toolpath strategies and simulating the machining process, CAM software helps identify areas for improvement and reduce machining time. This leads to faster production cycles and cost savings.

c. **Reduced errors and improved quality:** CAM software's verification tools help prevent toolpath errors and collisions that could damage the machine or workpiece. This contributes to maintaining consistent part quality and reducing production waste.

d. **Enhanced machining capabilities:** CAM software unlocks the full potential of CNC machines by enabling advanced machining techniques like 3-axis, 4-axis, and 5-axis machining for creating complex geometries with greater flexibility.

CAM software plays a vital role in modern manufacturing by enabling efficient and precise machining of complex parts. By understanding its functionalities and applications, you can appreciate its role in streamlining workflows, optimizing production processes, and achieving high-quality machined components.

2.7. Positional Machining

Positional machining is a valuable technique for utilizing 5-axis CNC machines to their partial potential. It offers significant advantages in terms of reduced setups, improved accuracy, and cost-effectiveness for workpieces with complex geometries that can be machined from multiple fixed orientations. However, it's important to consider the limitations when dealing with highly intricate features that require full 5-axis toolpath capabilities. It is close to industrial robotics preferably with 5 axis CNC machines.

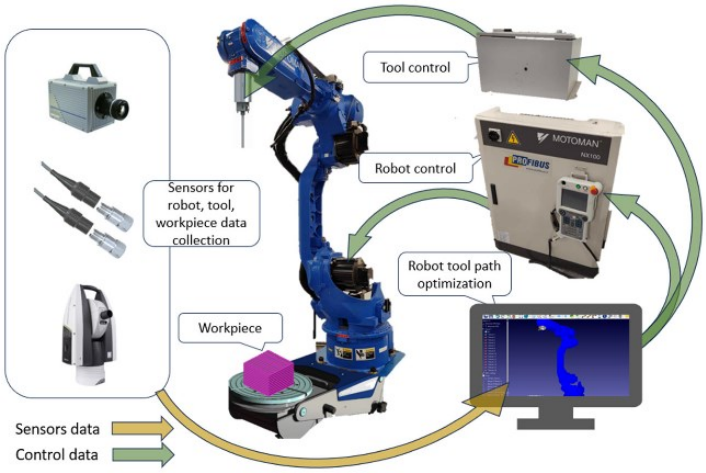


Figure 20: Positional machining.

Self-Check Sheet 2: Perform (3+2)/Positional Machining

1. What is the Stock Model?
   Answer:

2. Define Collision Control?
   Answer:

3. What do you mean by (3+2)/Positional Machining?
   Answer:

4. What is WCS Plane?
   Answer:

5. Describe some use of CAM software?

   Answer:

Answer Key - 2: Perform (3+2)/Positional Machining

1. What is the Stock Model?
   Answer: Stock typically refers to the **raw material** used to create the final part. This could refer to a **digital 3D model** that represents the raw material (stock) used for machining a part.

2. Define Collision Control?
   Answer: Collision Control allows you to virtually simulate the toolpath movements within a 3D model of your machine to identify potential collisions between the tool, tool holder, fixtures, and other machine components. By detecting potential collisions before running the machining process on the actual machine.

3. What do you mean by (3+2)/Positional Machining?
   Answer: This is a 5-axis machining approach where the tool is positioned (indexed) at a fixed angular orientation (not necessarily 90 degrees) for multiple machining operations. It utilizes 3-axis toolpath logic but allows for tilting and rotating the tool around the X and Y axes for machining complex features.

4. What is WCS Plane?
   Answer: The WCS (Work Coordinate System) plane is a fundamental concept that defines the origin and orientation for your entire machining project. It acts as the reference point for all toolpath movements and positioning within your 3D model.

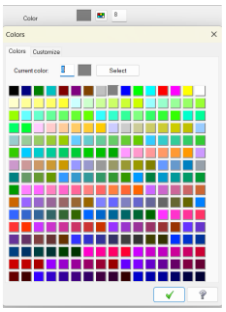
5. Describe some use of CAM software?
   Answer: CAM software (Computer-Aided Manufacturing) is a crucial tool used in manufacturing to bridge the gap between design and production. Here's a breakdown of its applications:

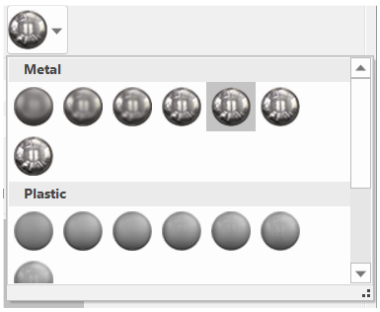
   a. Prototyping and Machining of Complex Part
   b. Improved Machining Efficiency
   c. Reduced Errors and Improved Quality
   d. Enhanced Machining Capabilities.

Job Sheet 2-1: Complete the Stock Model

Procedure:

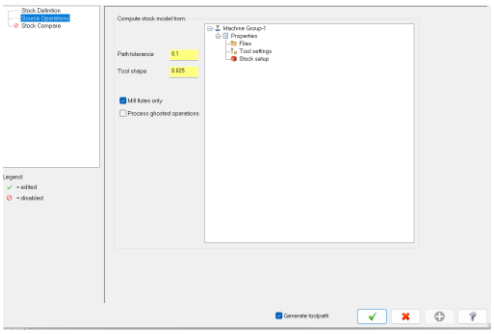
1. Wear appropriate PPE for the job
2. Read the Job sheet and Specification sheet provided.
3. Collect all needed materials, supplies, and equipment.
4. Prepare the tools, equipment, and materials for use.
5. Follow hygiene and safety requirements during the demonstration processes.
6. Apply the Stock Model option to Visualize the stock material.
7. Set up if you have your Stock name, Stock colour, Material, Additional offset, Source operations, Stock Compare as per your requirement. Note that all of information you had been used.
8. Set Up Name, Stock Colour and Materials



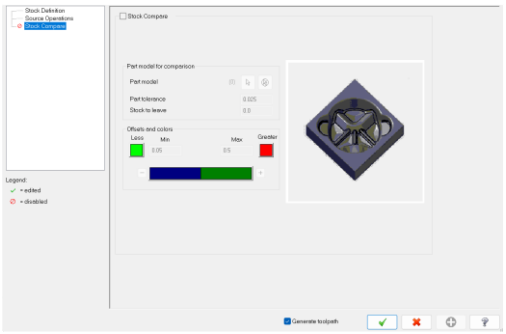


8. Stock color and Material:

9. Perform Additional offset
10. Perform Source operations: If before you have any operation and you need to remove from stock.
11. Perform Stock Compare

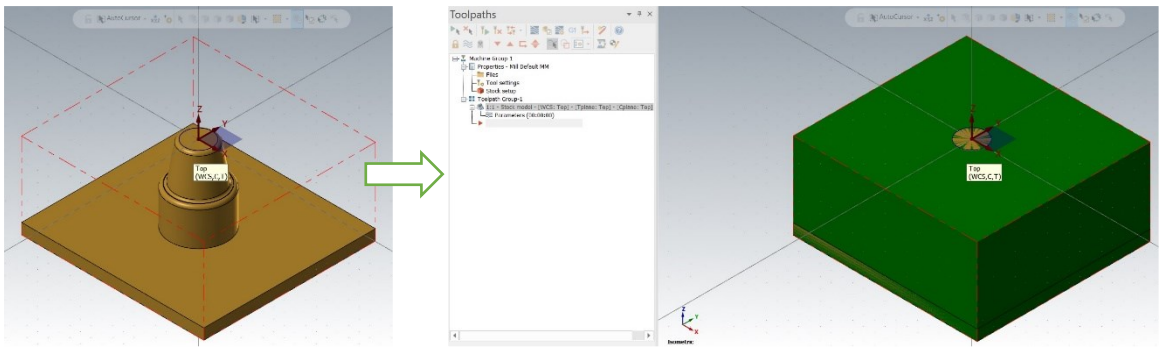


10



11

12. Perform setting the stock model, job will show like this:



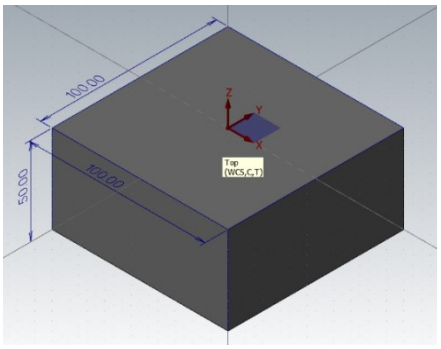
13. Customize the Stock Model options like Name of stock, Stock colour, Material, Additional offset, Source operations, Stock Compare.
14. Clean workplace.
15. Store tools and equipment in a safe place.

Specification Sheet 2-1: Complete the Stock Model

Job Name: Complete the stock model

Conditions for the job: You must practice safe operation procedures at all times during the assessment. You will be required to demonstrate OSH competencies **take the Stock Model.**

Diagram



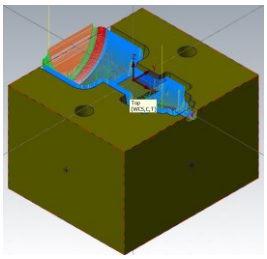
Tools, Equipment, Materials and PPE:

SL	Name of Tools, Equipment, Materials and PPE	Unit	Quantity
1	**Workstation computer** Operating System: Min Win 10 (64bit) Processor: Intel Xeon 3.3 GHz or equivalent **workstation grade** Ram: min 16GB Graphics Card: Min 4GB Nvidia Quadro SSD: Min 512GB HDD: 1TB Others: Monitor, Keyboard, Mouse, etc. Mastercam software Installed Application	Set	1
2	UPS	No	1
3	Plotter	No	1
4	Printer	No	1
5	USB / CD / DVD / Portable HD	No	1
6	Apron	No	1
7	Mask	No	1
8	Anti-Static mat	No	1

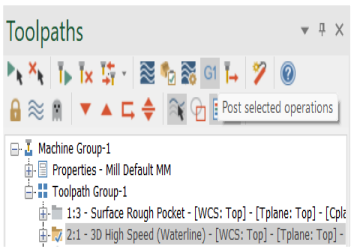
Job Sheet 2-2: Generate G&M Code for (3+2)/for positional machining

Procedure:

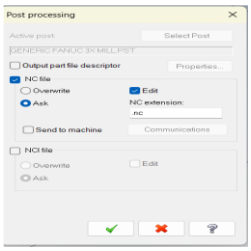
1. Wear appropriate PPE for the job
2. Read the Job sheet and Specification sheet provided.
3. Collect all needed materials, supplies, and equipment.
4. Prepare the tools, equipment, and materials for use.
5. Follow hygiene and safety requirements during the demonstration processes.
6. Apply the G1 option from the Mastercam to generate G&M codes.
7. Generating G&M codes, first you will need to check your toolpath operation carefully and select the toolpath one or more, then press the 'G1' option in the toolpath tree manager.
8. Pressing 'G1', a tab will show the code format:



6

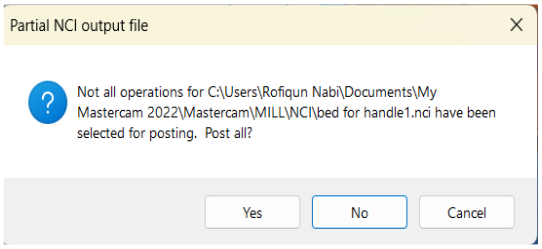


7

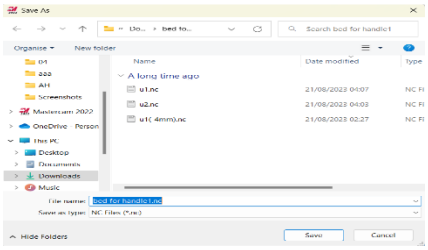


8

9. press ok but if an option shows like this:
10. Press no to generate your single operation codes.

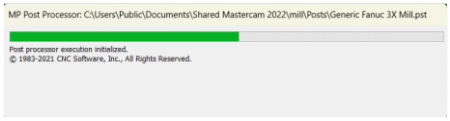


9

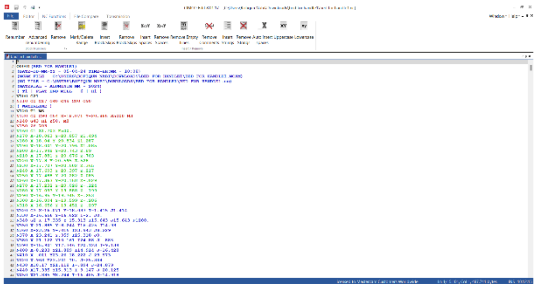


10

11. save it as anywhere as you like and click save:
12. Perform Generate code use post processor for selected machine.



13. Lastly Code Expert or CIMCO will generate your code which you will run in machine.



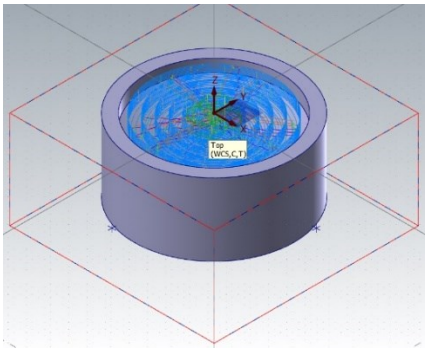
14. Clean workplace.
15. Store tools and equipment in a safe place.

Specification Sheet 2-3:
Generate G&M Code for (3+2)/for positional machining

Job Name: (3+2)/for positional machining generate G&M code

Conditions for the job: You must practice safe operation procedures at all times during the assessment. You will be required to demonstrate OSH competencies in **positional machining to generate G&M code.**

Diagram



Tools, Equipment, Materials and PPE:

SL	Name of Tools, Equipment, Materials and PPE	Unit	Quantity
1	**Workstation computer** Operating System: Min Win 10 (64bit) Processor: Intel Xeon 3.3 GHz or equivalent **workstation grade** Ram: min 16GB Graphics Card: Min 4GB Nvidia Quadro SSD: Min 512GB HDD: 1TB Others: Monitor, Keyboard, Mouse, etc. MasterCAM software Installed Application	Set	1
2	UPS	No	1
3	Plotter	No	1
4	Printer	No	1
5	USB / CD / DVD / Portable HD	No	1
6	Apron	No	1
7	Mask	No	1
8	Anti-Static mat	No	1

Review of Competency

Below is yourself assessment rating for module "**Perform Basic Multi-Axis Machining"**

Assessment of performance Criteria	Yes	No
5 axis is identified in the machine	☐	☐
Machine rotary zero points (MRZP) are identified	☐	☐
Dynamic work offset is set	☐	☐
Stock model is performed	☐	☐
Collision control is checked	☐	☐
G&M code is created for (3+2)/positional machining	☐	☐
Required plane is selected for (3+2)/positional machining in CAM software	☐	☐
Positional machining is executed using CAM software	☐	☐
Cutting in the machine is executed.	☐	☐

I now feel ready to undertake my formal competency assessment.

Signed:

Date:

Development of CBLM

The Competency based Learning Material (CBLM) of '**Performing Basic Multi-Axis 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	01737-278906
4	Engr. Md. Nazrul Islam	Reviewer	01711 273708

Reference:

1. **https://academy.titansofcnc.com/files/Fundamentals_of_CNC_Machining.pdf.**
2. **https://cache.industry.siemens.com/dl/files/454/37335454/att_110322/v1/SIN_WF5_0509_en.pdf,**
3. **https://www.slideshare.net/slideshow/5-axis-cnc-machine/137176160,**
4. **https://www.worthyhardware.com/news/multi-axis-cnc-machining/,**