

|            |  |
|------------|--|
| Week<br>03 | <ul style="list-style-type: none"> <li>• Explain Automatic tool exchanger using Multimedia</li> <li>• Explain the importance of Tool length compensation, Tool nose Radius compensation and Tool Wear compensation.</li> <li>• Explain Machine Zero and Work Zero</li> </ul> |
|------------|--|

### Automatic tool exchanger:

To make an automatic tool change, an M6 code is used. This code can be used in a program or in Manual Data Input (MDI) mode.

**In a Program** - When another tool is needed, an M6 followed by a tool location. (T\_) is entered and the tool changer will make the appropriate tool change and then continue on with the program.

**In MDI** - The operator can make a tool change with out running a program. If the operator is testing the tool changer or setting tool lengths, the M6 code followed by the tool location will command the tool changer to make the necessary tool change.

### Tool length compensation :

Tool length compensation is vital in CNC machining for enhancing precision by accounting for variations in tool length, improving tool life by preventing collisions, reducing rework and scrap by minimizing errors, and increasing efficiency by allowing seamless tool changes without program recalibration.

### Importance of Tool Length Compensation

- I. **Accuracy:** It eliminates dimensional errors by ensuring the tool tip, rather than the spindle nose, cuts to the correct position.
- II. **Efficiency:** By eliminating the need to manually adjust machining programs for each tool change, it streamlines operations and reduces machining time.
- III. **Cost Reduction:** Minimizing errors leads to less rework and scrap, while protecting tools from collisions extends their life, saving on replacement costs.
- IV. **Tool Life:** By accurately positioning the tool and preventing it from cutting at the wrong depth, it reduces wear and the likelihood of tool breakage.
- V. **Consistency:** It ensures consistent part quality across multiple machining operations by maintaining accurate tool positioning.
- VI. **Flexibility:** It allows CNC machines to use a variety of tools of different lengths without modifying the entire machining program for each tool.
- VII. **Tool Wear Management:** It automatically adjusts for the natural wear of tools, which can shorten them over time, ensuring consistent cutting quality.

### Cutter Radius Compensation

**Definition and Purpose :** The part program is developed for the cutter path with reference to the centre of the tool rather than periphery where the actual cutting takes place. At the time of writing a part program, a cutter of suitable diameter is selected.

The difference in the programmed diameter of the cutter and the diameter of the actual cutter is accounted for by cutter radius compensation. The following three G-codes are used cutter radius compensation.

|   |  |
|---|--|
| <ul style="list-style-type: none"> <li>• G-40: Cancel cutter radius compensation</li> <li>• G-41: Compensation applied to shift the cutter on the left of the programmed path.</li> <li>• G-42: Compensation applied to shift the cutter on the right of the programmed path.</li> </ul> <p>The direction in which the cutter path has to be shifted is decided by looking in the direction of cut.</p> |  |
|---|--|

Cutter Radius Compensation (CRC) is a CNC machining technique used to adjust the tool's path based on its radius. It ensures that the tool follows the intended path, taking into account its size. Imagine a milling tool with a 6mm radius; without CRC, the machine would interpret tool movement based on the tool's center, potentially causing errors in the final machined part. CRC corrects this by offsetting the toolpath by half the tool's diameter or radius.

## Implementing Cutter Radius Compensation -

### Explanation of the G41 and G42 G-codes

Cutter Radius Compensation in CNC machining is activated using the G41 and G42 G-codes. Here's a breakdown of these codes:

- **G41:** When you use G41 in your CNC program, you are instructing the machine to apply left-hand compensation. This means the toolpath will be offset to the left of the programmed path by half the tool's radius.
- **G42:** Conversely, G42 enables right-hand compensation. With this code, the toolpath will be offset to the right of the programmed path by half the tool's radius.
- **G41/G42 Commands:** In your CNC program, insert either the G41 or G42 command, followed by the tool number. For instance, G41 D2 would enable left-hand compensation for tool number 2. G42 D3 would activate right-hand compensation for tool number 3.

### Importance of Cutter Radius Compensation:

- I. **Simplified Programming:** It allows programmers to use the desired part surface coordinates rather than calculating the precise centerline path for every motion, which is especially helpful for complex shapes.
- II. **Accounts for Tool Wear:** It enables machinists to make precise adjustments to account for tool wear and other variations, ensuring the final part dimensions meet specifications.
- III. **Reduces Errors :** By accounting for the tool's diameter, it minimizes errors that would otherwise occur if the programmer had to manually account for the radius, leading to better accuracy and quality.
- IV. **Easier Part Size Adjustments:** Using cutter compensation, it's simple to adjust the finished part size by changing the tool radius offset, which is more efficient than rewriting the entire program, according to the CNC Training Centre.
- V. **Improved Efficiency:** Eliminates the need for numerous complex calculations, especially when dealing with intricate contours, saving significant time and effort in the programming stage, says ScienceDirect.
- VI. **Flexibility with Tooling:** Allows for the use of different-sized cutters on the same programmed path by simply changing the offset in the controller, which would otherwise require re-programming the entire toolpath.
- VII. **Accuracy for Matching Parts:** Helps ensure that multiple parts machined with the same program are finished to the same accurate dimensions, improving consistency

### Key Differences Between Tool Length and Cutter Radius Compensation –

| Tool Length Compensation (G43)   | Cutter Radius Compensation (G41/G42)  |
|--|---|
| <ul style="list-style-type: none"> <li>• <b>Focus:</b> Tool Length Compensation primarily deals with adjusting the tool's position along its axis to account for variations in tool length.</li> </ul>                                     | <ul style="list-style-type: none"> <li>• <b>Focus:</b> Cutter Radius Compensation centers around adjusting the toolpath to accommodate the tool's geometry, specifically its radius, ensuring that the machined feature matches the intended dimensions.</li> </ul> |
| <ul style="list-style-type: none"> <li>• <b>Parameter:</b> G43 is the G-code used for enabling Tool Length Compensation, with an additional parameter (H) specifying the tool number.</li> </ul>   | <ul style="list-style-type: none"> <li>• <b>Parameters:</b> G41 enables left-hand compensation, while G42 enables right-hand compensation. The D parameter is used to specify the tool's radius.</li> </ul>   |
| <ul style="list-style-type: none"> <li>• <b>Applications:</b> It is critical for operations where the tool's length is pivotal, such as drilling to a precise depth or machining parts that require consistent tool engagement.</li> </ul> | <ul style="list-style-type: none"> <li>• <b>Applications:</b> CRC is crucial for tasks involving complex contours, pocketing, and any operation where the tool's radius affects the final part's accuracy.</li> </ul>   |

## Tool Wear compensation

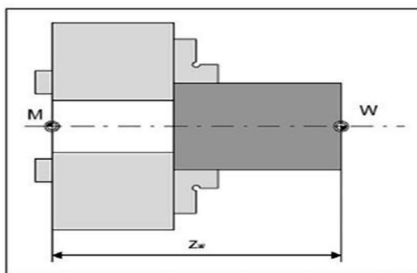
Tool wear compensation is vital for maintaining precision and efficiency in manufacturing by correcting for tool degradation during machining, which ensures part accuracy, reduces waste, extends tool life, and improves process consistency.

### Importance of Tool Wear Compensation

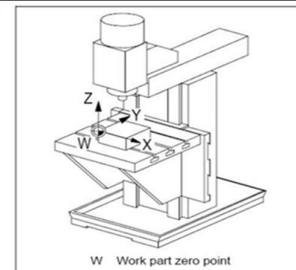
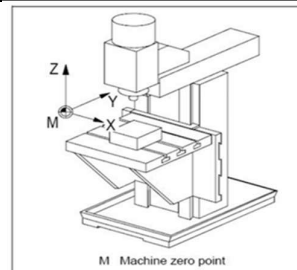
- I. **Maintains Part Accuracy:** By counteracting the effects of wear, compensation ensures that parts are machined to the required dimensions and tolerances, preventing out-of-spec products.
- II. **Reduces Rework and Scrap:** It minimizes dimensional errors, thereby reducing the need for scrap or rework, which directly lowers manufacturing costs.
- III. **Increases Efficiency:** Compensation allows machines to operate for longer periods without interruption for manual adjustments, leading to higher productivity and a more efficient workflow.
- IV. **Extends Tool Life:** By protecting tools from damage and ensuring proper positioning, the life of cutting tools is extended, reducing the frequency and cost of tool replacement.
- V. **Improves Process Consistency:** It ensures consistent quality and repeatability across successive machining operations, leading to more stable and reliable manufacturing processes.
- VI. **Reduces Machining Time:** By minimizing errors like overcutting and tool collisions, it reduces the time required for machining, further boosting productivity.

### Machine Zero and Work Zero

- In CNC machining, machine zero refers to the fixed origin point of the machine's coordinate system, established by the manufacturer and unchangeable.
- Work zero, also known as part zero, is the origin point defined by the programmer on the workpiece for a specific machining job.



M = Machine zero, W = Work Piece zero  
Fig. Machine zero and work piece zero on lathe machine



M Machine zero point  
W Work part zero point  
Machine Zero Point and Work part zero point on  
CNC milling machine

### Machine Zero:

- **Fixed Point:** Machine zero is a pre-defined, unchanging point on the CNC machine.
- **Manufacturer Defined:** The machine's manufacturer sets this point during the machine's construction.
- **Reference for All Axes:** It serves as the reference point for the machine's coordinate system, encompassing all axes (X, Y, Z).
- **Example:** On a CNC lathe, machine zero is often located at the center of the spindle nose.
- **Not User Adjustable:** You cannot directly change machine zero.

### Work Zero:

- **User Defined:** Work zero is a point on the workpiece that the programmer selects as the origin for the current machining operation.
- **Varies by Job:** The location of work zero can change depending on the specific part being machined.
- **Coordinate System Origin:** It establishes the coordinate system relative to the workpiece.
- **Example:** For a turning part, work zero is often set at the front face and center of the part.
- **Offset:** The distance between machine zero and work zero is accounted for using offsets, like G54 in CNC programming.