MT107

SINUMERIK HANDBOOK

How to Measure **Tool offset in Milling**

This handbook provides a step by step guide on how to take tool offset in Milling machine technology.

SIEMENS

Higher productivity with SINUMERIK

Whether it’s a basic part or a complex workpiece, the Siemens control system offers you a wealth of functionality with a very high degree of operator friendliness and manufacturing efficiency.

Description of the document:

This document highlights the concept of Creating and Measuring tool in the Milling machine.

- Create Tool — Creation of setup tools
- Measure Tool — Measure tool offset /test tool offset

Applicability:

- Technology — 3 axis Milling & Milling 3+1, & 3+2 Machine
- SINUMERIK controllers — 808D, 828D, 840Dsl & SINUMERIK ONE

Index

Creation of **setup tools**

Load tool into the Magazine

- While creating new tool ensure the location of the Magazine is empty as depicted below.

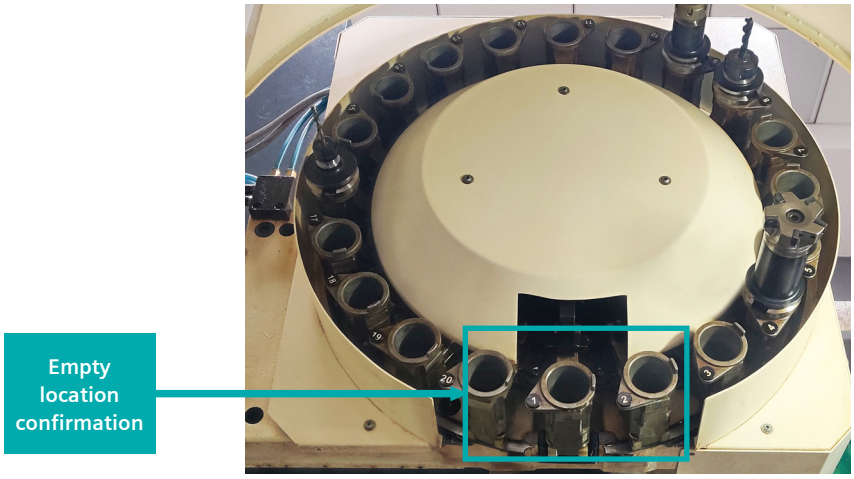


Figure 1.1

- Load the tool into the spindle at the required location of the Magazine as shown below.

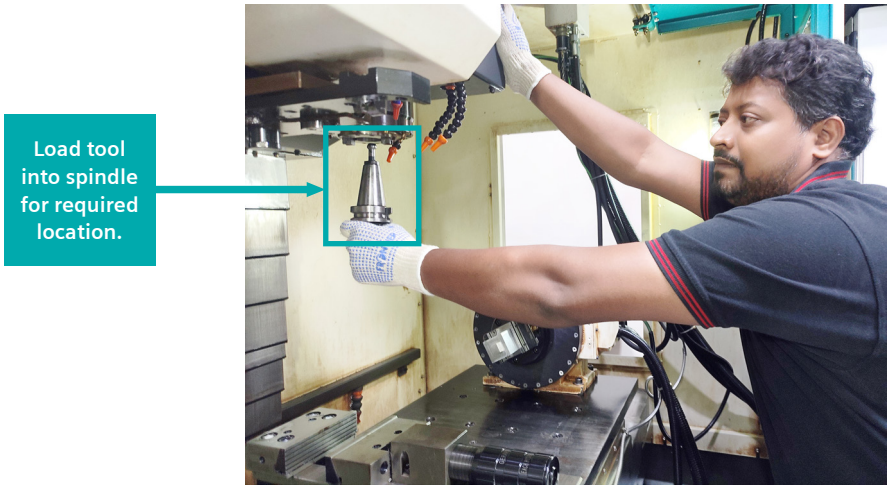
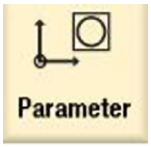


Figure 1.2

Create a new tool in the tool list

A tool must be created in the tool list before measuring the tool.

Press "**Parameter**" in the operating area. → 

Then press the "**Tool list**" softkey → 

By pressing the above softkey "**Tool list**" the tool list window will open as depicted in the subsequent page.

Selecting the function “New Tool”

Move the cursor on the loaded tool location in the list as depicted below.

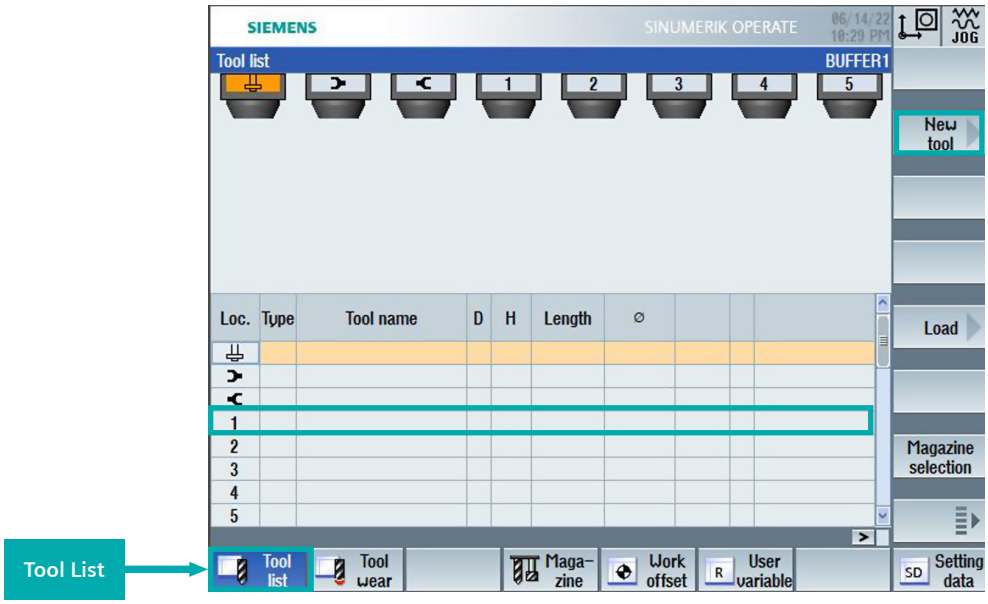


Figure 1.3

For Example: Location **Number 1** is highlighted above.

Press the “**New tool**” softkey as depicted.

After pressing the **New tool** in the empty location, the tool library window will open as shown below.

Figure 1.4

Select the tool type of the required tool from the tool library. After selecting the tool in the tool list as shown in in the subsequent page

For Example: Creating Milling tools for Location Number 1
Create required Milling tools in the list as shown below picture.

- Milling Tool (End mill _Type_120) as shown above picture.

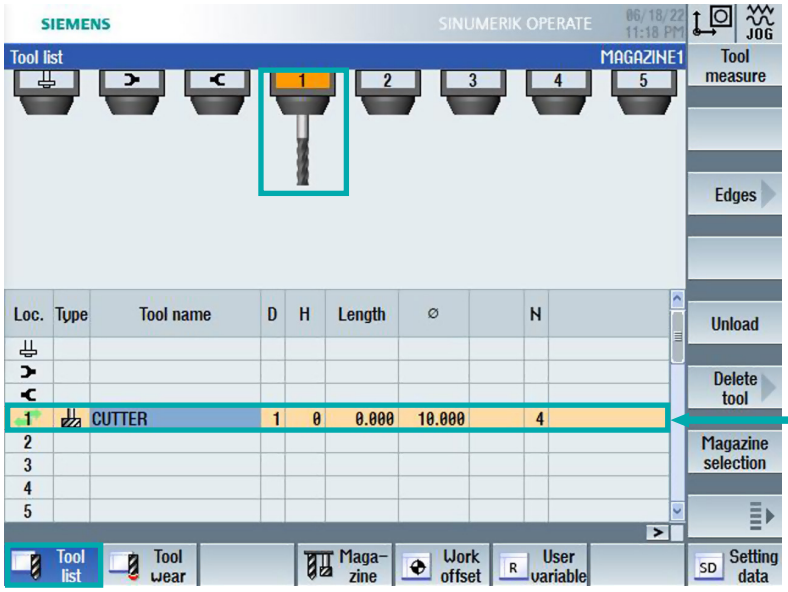
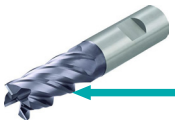
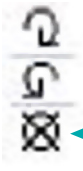


Figure 1.5

Parameter for "Tool list" as shown in below.

Parameters	Meanings
Location	Locations of the Magazine.
Type	Define **type of tool** while selecting Milling tools as shown in the picture in the inset.  Type of tool (End mill)
Tool Name	Name of tool (To identify a tool, Tool name can be entered as text or number).
D1	**Too offset Number** for selecting tool.
Length Z	Tool length for Z axis.
Radius	Define specific **radius** of tool as shown in thepicture in the inset.  Radius of (End mill)
N	Number of Tool cutting edges.  Number of Cutting Edges (4)
	Spindle direction  Direction of spindle M03/ M04

Measure tool offset

Measure Length

A tool must be created and loaded before it can be measured.

Press "**Machine**" in the operating area. → 

Press the "**JOG**" operating mode. → 

Then press the "**TSM**" softkey. → 

Select measure tool, work offset, enter a spindle & spindle direction CW. Then press "**Cycle Start**" as shown below.

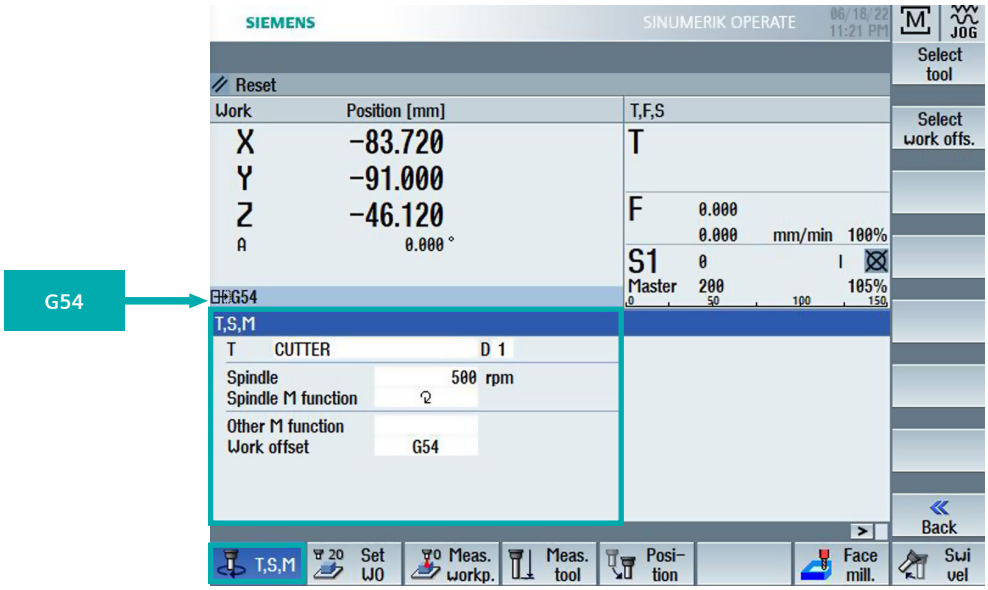


Figure 1.6

Press the "**Measure tool**" softkey. → 

After pressing the "**Measure tool**" softkey, various functions for measuring will be available in the vertical softkey bar as depicted below.

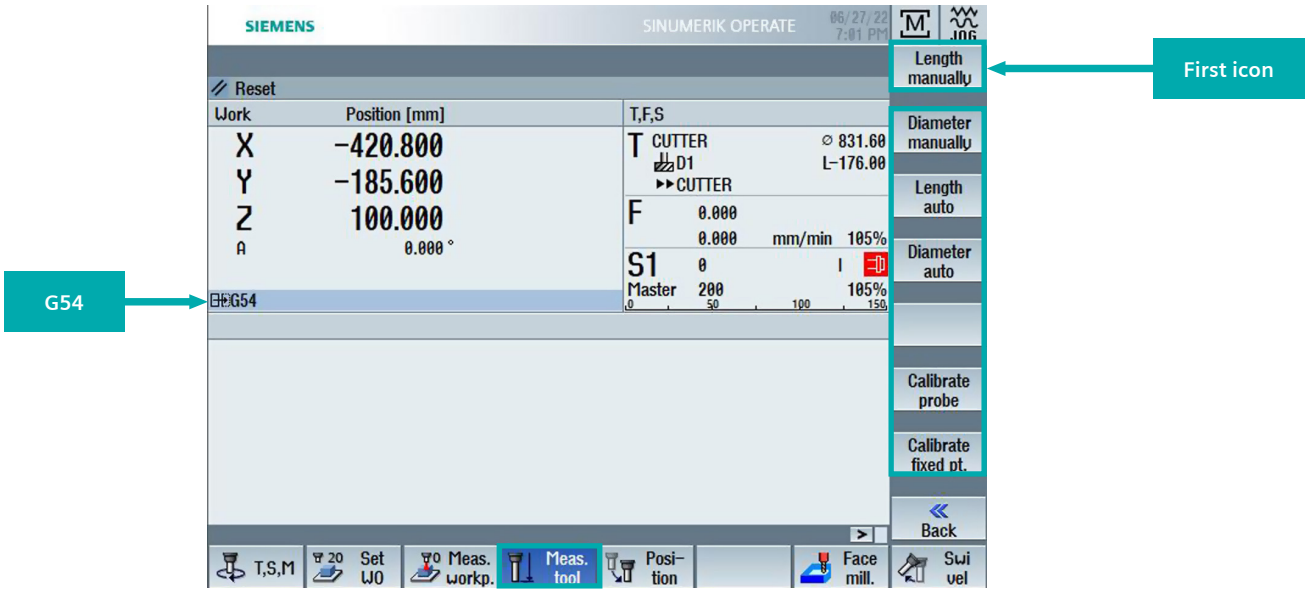


Figure 1.7

Press the corresponding softkey of the **first icon** on the right-hand side of the screen.

Press the "**Hand wheel**" key on the MCP and position the tool at location Z=0 of the work piece.
Workpiece.

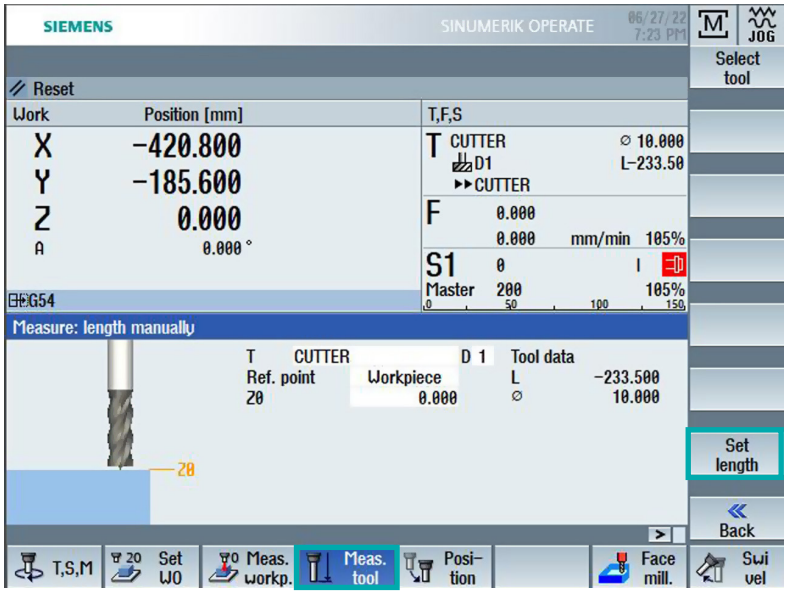


Figure 1.8

Press the "**set length**" softkey as depicted above

The measured tool length is now shown in “Tool data (L)”. This value is also saved in the length value column of the corresponding tool list at the same time depicted below.

Test the tool offset results

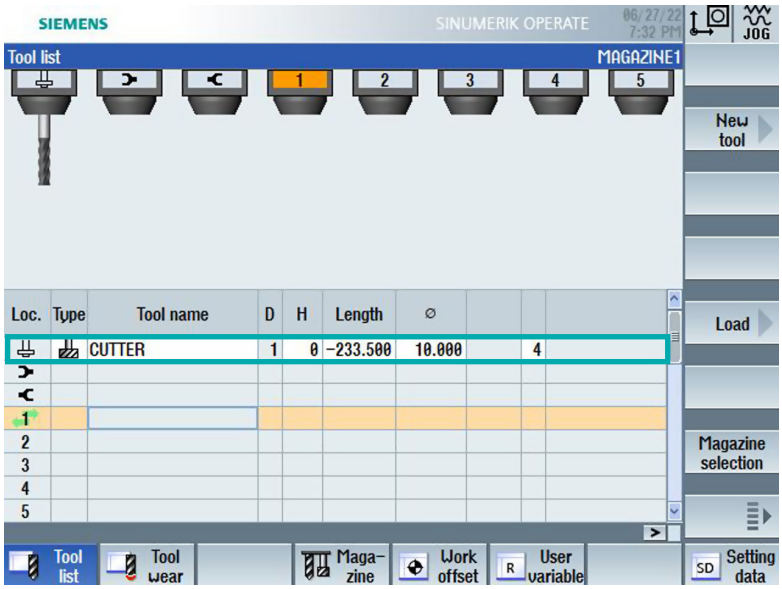


Figure 1.9

The tool offset must be performed correctly for it to be tested as follows.

Press the “**MDA**” operating mode →

Press the “**Delete blocks**” softkey →

Enter the test program recommended on the right side.

```
G00X0Y0Z50
G01Z0.0F200
```

Press “**Cycle Start**” operating mode →

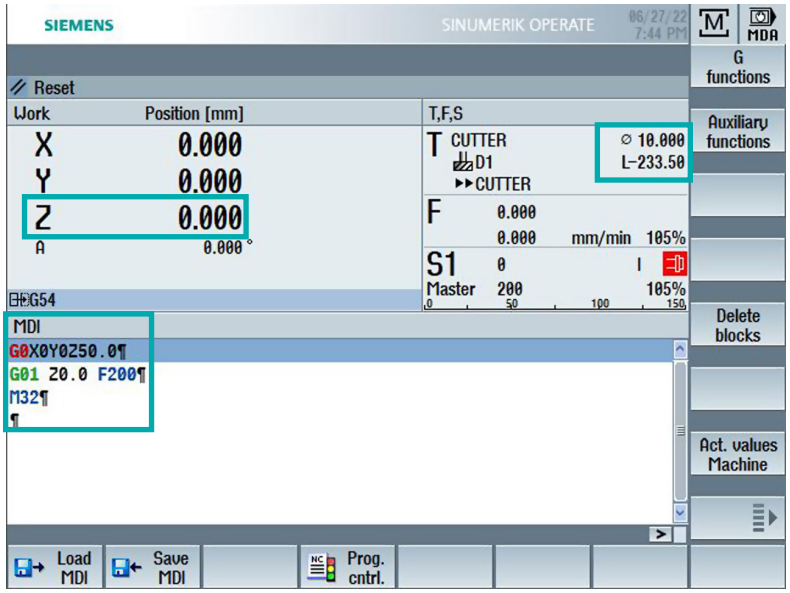
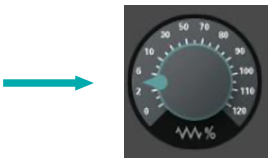


Figure 2.0

Increase the feedrate override gradually to avoid accidents caused by an axis moving too fast and observe whether the axis moves to the set position as depicted above.



To know more about SINUMERIK

Enrol for Application & Commissioning trainings
🌐: www.siemens.co.in/dex
☎: 080-40975628
✉: tac.training.in@siemens.com

▶ Watch SINUMERIK Video Tutorials Online

▶ SINUTRAIN—Free Download

Published by
Siemens Limited
Digital Industries
Motion Control

XX-XXX-XXX

For online application support
Prakash Somannavar
☎: +91-8980023525
✉: prakash.somannavar@siemens.com

Syed Imtiyaz
☎: +91-9945293532
✉: syed.imtiyaz@siemens.com

For Service Support
:18002090987
✉: ics.india@siemens.com
🌐: www.siemens.co.in/industry-services

S18473