ΠΑΝΕΠΙΣΤΗΜΙΟ ΠΑΤΡΩΝ
UNIVERSITY OF PATRAS

ΑΝΟΙΚΤΑ ακαδημαϊκά μαθήματα ΠΠ

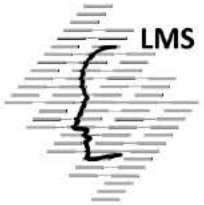
Αριθμητικός Έλεγχος Εργαλειομηχανών

Ενότητα 10: Cutter Diameter Compensation

Δημήτρης Μούρτζης, Επίκουρος Καθηγητής
Πολυτεχνική Σχολή
Τμήμα Μηχανολόγων & Αεροναυπηγών Μηχανικών

COMPUTER NUMERICAL CONTROL OF MACHINE TOOLS

Laboratory for Manufacturing Systems and Automation
Department of Mechanical Engineering and Aeronautics
University of Patras, Greece



Dr. Dimitris Mourtzis

Assistant Professor

Patras, 2015

Objectives of section 10

- Define **cutter diameter compensation**
- Describe **ramp off** moves and explain their importance
- List the **precautions** necessary when using cutter diameter compensation
- Write programs in word address that **utilize cutter diameter compensation**

Definitions and Codes

- Programs presented in previous sections required an **allowance** for the cutter radius in the programmed coordinates

- Most CNC machines have a built-in feature called ***cutter diameter compensation (cutter comp)*** that allows the part line to be programmed. (Confusion may be caused by use of the terms "**offset**" and "**compensation**") In this text, **"compensation"** refers to cutter diameter offset

- The term "**offset**" refers to ***tool length offset*** and the change in axis coordinates when programming arcs and angles.) Cutter comp is also called ***cutter radius offset (CRO)*** by some controller manufacturers

- In computer-aided programming languages (such as APT) and some CAD/CAM systems it is also called **cutcom**. These terms all refer to the same thing: **a built-in cycle** in the MCU that, when activated, ***alters*** the tool path by an amount contained in the cutter comp register

- The value in the register is entered in by the setup person when the job is being prepared

Setting Tool Length Offsets

- **Basic principles** of CNC machines :
  - **Table movement** and / or tool holder by a **predetermined** length from a reference point
  - A datum per movement axis → setting tool cutting edge
  - Need to use more than one cutting tools → set each tool length offset or equilibration
  - Cutting tool offset
- **Change in diameter** due to:
  - Deterioration
  - Change cutter
  - Rounding of the edge radius of the cutting tool

Definitions and Codes

Cutter comp is accomplished through the use of G codes : G40, G41, G42

G40 – Cutter diameter compensation **cancel**. Upon receiving a **G40**, cutter diameter compensation **is turned off**. The tool will change from a compensated position to an uncompensated position on the next X, Y, or Z axis move

G41 – Cutter diameter compensation **left**. Upon receiving a **G41**, the tool will compensate to the left of the programmed surface. The tool will move to a compensated position on the next X, Y, or Z axis move after the **G41** is received

G42 – Cutter diameter compensation **right**. Compensates to the right of the programmed surface

Codes G41, G42

- The orbit of the center of the cutting tool **is not identical** with the piece geometry

Compensation: **Automatic calculation** of the cutter path based on the diameter / radius of the cutting tool

- **G41,G42**
- Command format

N.. G01 G41 X.. Y.. D..

N.. G02 G41 X.. Y.. I.. J.. D..

- Where
  - **D** is the memory address of machine's MCU where the compensation value is registered

G Codes:G41, G42

- **G40**
- **Compensation cancel** (**G41** and **G42**) of cutter radius
- Activated **automatically** by machine at the beginning of each program
- «**Modal**» command

Definitions and Codes

- Most controllers allow compensation to be performed on any two axes. A **G code** is used to determine which **axes combination** is to be used

  - If the part is to be machined using the X and Y axes, compensation is desired in the X/Y plane

  - If using the X and Z axes, compensation in the Z/X plane is needed

  - If using the Y and Z axes, compensation is needed in the Y/Z plane. The X/Y plane is used most commonly.

The G codes used to select the desired work plane are:

G17 – X/Y plane

G18 – Z/X plane

G19 – Y/Z plane

Special Considerations

- Sometimes, a controller requires a **vector** to be commanded with the **G41** or **G42** to orient the cutter correctly prior to the ramp-on move

  - Technically, a **vector** is a geometric entity that has both **magnitude** (length) and **direction**

  - In NC programming, vectors are simply mathematical arrows that point the cutter in a given direction

  - To utilize a vector the **I** and **J addresses** are used

Fine Tuning With Cutter Diameter Compensation

- Up to this point, **cutter diameter compensation** has been used to program the part line; the **program coordinates have matched the part dimensions**
- Another way **cutter comp** is employed is to **fine tune the cutter path**
- In this type of programming, the part is programmed using the parallel path method
- Cutter comp is used to compensate for the difference between the programmed and actual cutter diameter
  - For example, if a program is written for a .500-diameter end mill, but a resharpened end mill measuring .490 diameter is used, the .020 diameter difference can be compensated by using cutter comp.

Fine Tuning With Cutter Diameter Compensation

- In the **fine tune method**, **cutter comp** is usually used to ***compensate for a cutter which is smaller than the programmed diameter***. When using the part line method exactly the opposite is the case

- **Cutter comp** is used to ***compensate for a cutter that is larger than the zero diameter cutter programmed*** (the part line)

- For this reason, **it is necessary to use a minus (-) value** in the **cutter comp** register when using the **fine tune method**

- Note that allowance is once again being made for the cutter radius. The cutter diameter compensation allows reground, undersize cutters to be used

Summary

The important concepts presented in this section are:

- Cutter diameter **compensation** is the automatic calculation of the cutter path by the machine control unit, based on the part line and cutter information contained in the program

- Cutter diameter compensation is **instituted** and **canceled** through use of the codes **G40**, **G41**, and **G42**. **G41** is cutter **compensation** left, **G42** is cutter compensation **right**, and **G40** is cutter compensation **cancel**

- The "**ramp on**" move is the initial compensation of the cutter. The compensation occurs 90 degrees to the next axis movement following the **G41** or **G42**. Care must be taken with the spindle position prior to the ramp on move to avoid cutting the part in the wrong area

- The "**ramp off**" move is the opposite operation. Ramp off will occur 90 degrees to the next axis movement following a **G40**. The compensation will be completely eliminated by the end of this move.

Vocabulary Introduced in this section

- Approach angle
- Cutter diameter compensation (cutter comp)
- Cutter radius offset (CRO)
- Ramp off move
- Ramp on move

End of Section

Funding

- This educational material has been developed in the teaching duties of the respective educator.
- The Project "**Open Academic Courses at the University of Patras**" has funded only the reformation of the educational material.
- The Project is implemented within the context of the Operational Programme "Education and Lifelong Learning" (EdLL) and is co-funded by the European Union (European Social Fund) and national resources.

Reference Note

License Note

Notes Preservation

Any reproduction or modification of this material must include:

- the Reference Note
- the License Note
- the Notes Preservation statement
- the Third Parties' Works Note (if exists)

as well as the accompanying hyperlinks.