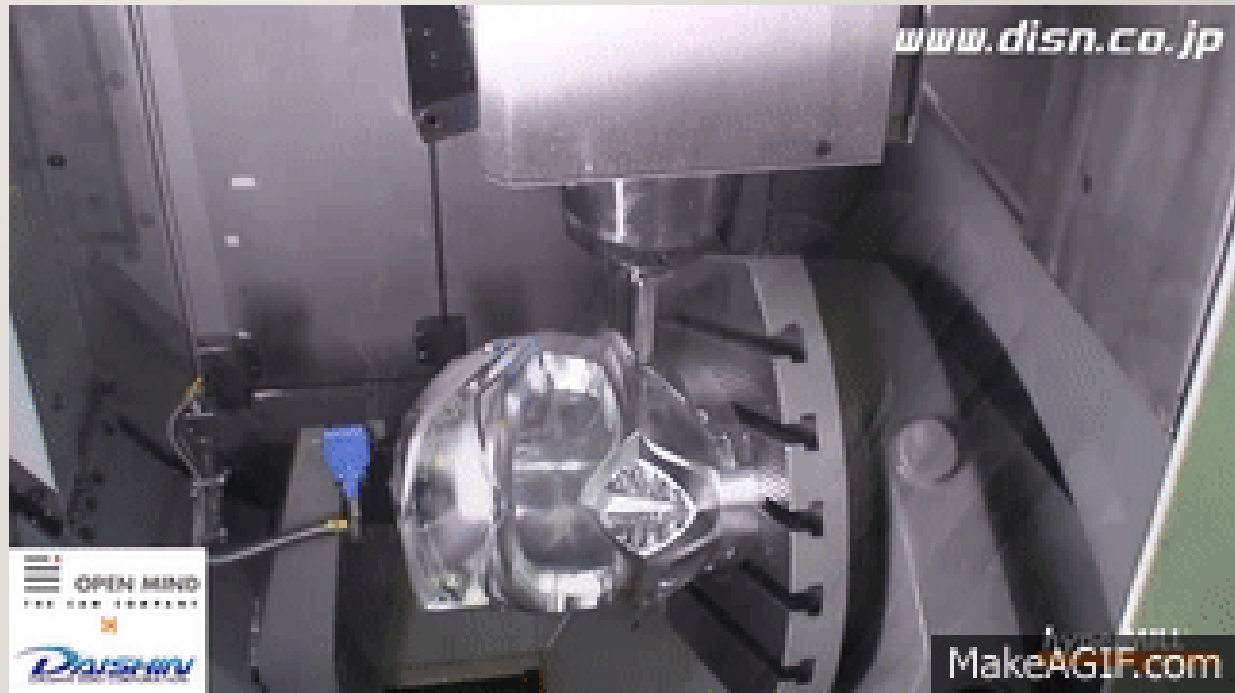


INTRO TO CNC PROGRAMMING

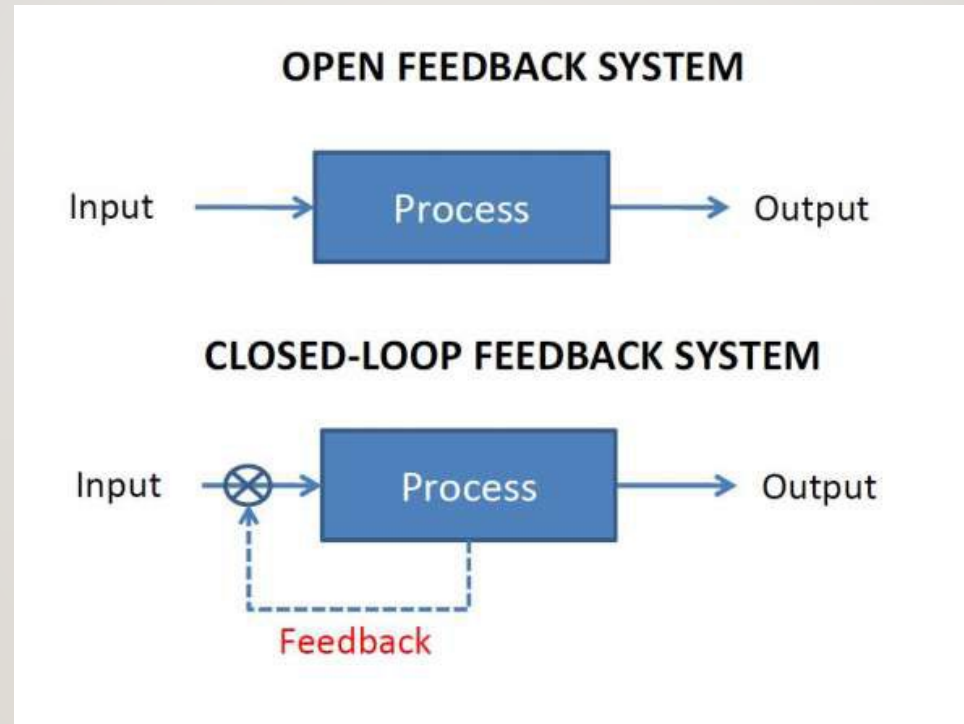


WHY DO WE USE CNC CONTROLS?

- Repeatability
- Accuracy
- Cycle time (Time=\$\$\$)
- Complex movements

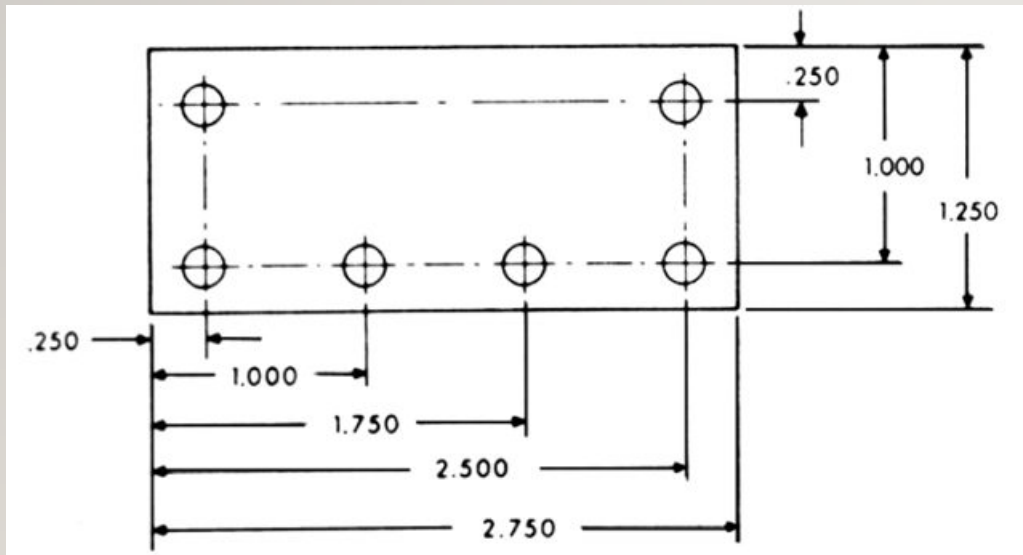


NC vs CNC (OPEN LOOP VS CLOSED LOOP)

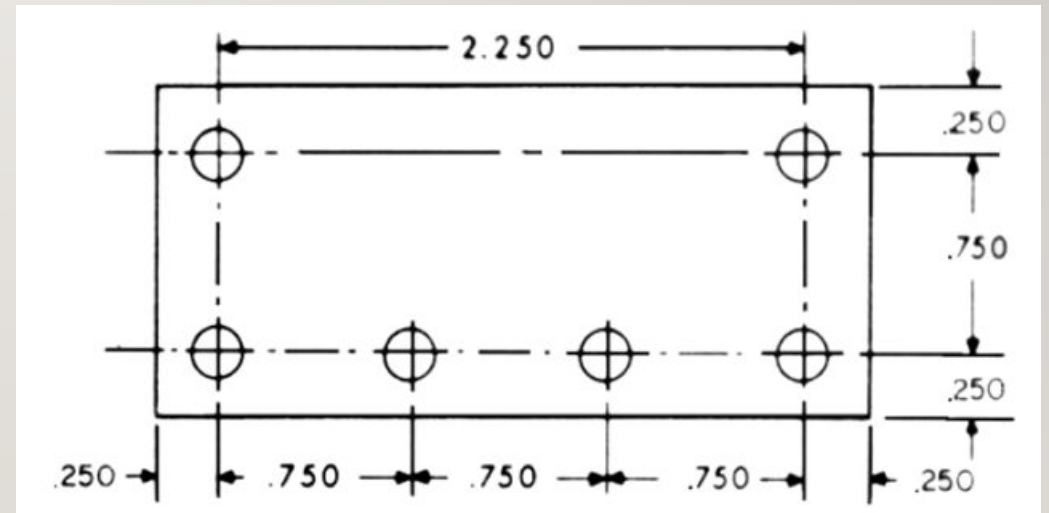


ABSOLUTE vs INCREMENTAL DIMENSIONS

- Absolute (X,Z)
 - Dimensions from work offset position



- Incremental (U,W)
 - Dimensions from your last position



OFFSETS AND MACHINE HOME

- **Work Offsets**

- Tells the machine where the point in space where the part is programmed from.
- Typically G54-G59
- Haas also provides G154 P1-P99

THIS IS NOT HOME!!!!

- Machine home is the physical limits of the machine

- **Tool Offsets**

- Tells the machine where the tip of the tool is in relation to your Work Offset.

Not typically associated with a specific G-code. This is recorded in the controller of the machine.

- Lathes can have multiple tool offsets
- Mills will only have one tool offset

BASIC CODES

- 1) Percent signs
- 2) Program numbers
- 3) Nxx – line numbers
- 4) Gxx – Preparatory/Movement codes
- 5) Mxx – misc. codes
- 6) Safe Line
 - a. G18 G20 G40 G80 G97 G99
- 7) Canned Cycles
- 8) Auxiliary Codes
 - a. These are codes within a Canned Cycle or M-code. (Tool numbers, Speeds, Feeds, Depth of cut, Movements or Dimensions)
 - b. Examples:
 - i. M06 T01 (M06 = tool change for “T1” or Tool #1)
 - ii. G81 X3.6 Y-1.2425 Z-0.5294 R0.2 F6.8
(Drilling Cycle. X and Y are positional. Z = final depth. R = return plane after drilling. F = feed rate)

PROGRAM STOPS

- M00 – mandatory pause in the program at a logical stopping point. Can be after a specific operation or between tool changes. The controller cannot ignore this.
- M01 – optional stop in the program. This works just like M00 but uses a button on the face of the controller to turn on or off the “optional stops” within a program.

SAFE LINE

- Sets the machine up and resets any outstanding codes
 - Absolute vs incremental
 - What plane your working in
 - Turn off canned cycles
- Not required for program to run but it's a good practice to be in
- “G18 G20 G40 G54 G80 G97 G99”
- Used to ensure a safe start condition

BASIC PROGRAM STRUCTURE

- Program Example