

# Task Assignment for G-Code Interpreter

## Task Description

Implement a program that reads the attached G-Code program from a file and parses the instructions. The program is expected to call *MachineClient's* stub methods to simulate execution. MachineClient implements a simplified interface to control a CNC machine.

G-Code is a programming language for CNC that instructs machines where and how to move. The language is not too complicated but it introduces number of instructions. Purpose is not to implement fully compliant G-Code interpreter but a limited subset of functions required to carry out the given task. It's not required to implement all the G-Code instructions used to initialise the machine (e.g., "*N1 G00 G17 G21 G40 G49 G80 G94*") - but it would be nice to see stub functions that are called to handle also these commands.

Input file containing the G-Code instructions should be passed to the program as a command line argument.

```
$ python cnc.py rectangle.gcode
```

If you are not familiar with G-Code you may find some background information e.g., from <https://www.autodesk.com/industry/manufacturing/resources/manufacturing-engineer/g-code> to understand the basics.

## Simplified CNC Machine Client

Simplified CNC machine client API.

```

class MachineClient:
    def home(self):
        """ Moves machine to home position. """
        print("Moving to home.")

    def move(self, x, y, z):
        """ Uses linear movement to move spindle to given XYZ coordinates.

        Args:
            x (float): X axis absolute value [mm]
            y (float): Y axis absolute value [mm]
            z (float): Z axis absolute value [mm]
        """
        print("Moving to X={:.3f} Y={:.3f} Z={:.3f} [mm]".format(x, y, z))

    def move_x(self, value):
        """ Move spindle to given X coordinate. Keeps current Y and Z unchanged.

        Args:
            value (float): Axis absolute value [mm]
        """
        print("Moving X to {:.3f} [mm]".format(value))

    def move_y(self, value):
        """ Move spindle to given Y coordinate. Keeps current X and Z unchanged.

        Args:
            value(float): Axis absolute value [mm]
        """
        print("Moving Y to {:.3f} [mm]".format(value))

    def move_z(self, value):
        """ Move spindle to given Z coordinate. Keeps current X and Y unchanged.

        Args:
            value (float): Axis absolute value [mm]
        """
        print("Moving Z to {:.3f} [mm]".format(value))

    def set_feed_rate(self, value):
        """ Set spindle feed rate.

        Args:
            value (float): Feed rate [mm/s]
        """
        print("Using feed rate {} [mm/s]".format(value))

    def set_spindle_speed(self, value):
        """ Set spindle rotational speed.

        Args:
            value (int): Spindle speed [rpm]
        """
        print("Using spindle speed {} [mm/s]".format(value))

    def change_tool(self, tool_name):
        """ Change tool with given name.

        Args:
            tool_name (str): Tool name.
        """
        print("Changing tool '{:s}'".format(tool_name))

    def coolant_on(self):
        """ Turns spindle coolant on. """
        print("Coolant turned on.")

    def coolant_off(self):
        """ Turns spindle coolant off. """
        print("Coolant turned off.")

```

## G-Code Program

G-Code program defines a rectangular shape to be milled.

The program is simplified for educational purposes.

You can view the G-Code program in use by e.g., downloading [CAMotics](#) application.

### Some tips

%	Initialisation code used at the beginning and end of the program.
(*)	Comment
Oxxxx	Program number
Nx	Optional line number at the beginning of a line.

```
%
O0001
(DIA 20.0 END MILL - NO CUTTER RADIUS COMP USED)
(MACHINE OUTSIDE OF 100 X 200 RECTANGLE)
(X0.0 Y0.0 - BOTTOM LEFT CORNER)
N1 G00 G17 G21 G40 G49 G80 G94
(SET AND CHANGE TOOL 01)
N4 T01 M06
N5 S2000 M03
N6 G90 G54 G00 X-12.000 Y-12.000
(CUTTING STARTS)
N9 G01 Z-5.000 F100.
(LINEAR FEED TO XY WITH GIVEN FEED RATE)
N10 G01 X-12.000 Y-10.000 F600.
N11 G01 X110.000
N12 G01 Y210.000
N13 G01 X-10.000
N14 G01 Y-12.000
(LIFT SPINDLE)
N15 G00 Z10.000 M09
(STOP SPINDLE)
N16 G91 G28 Z0.0 M05
(PROGRAM END)
N18 M30
%
```

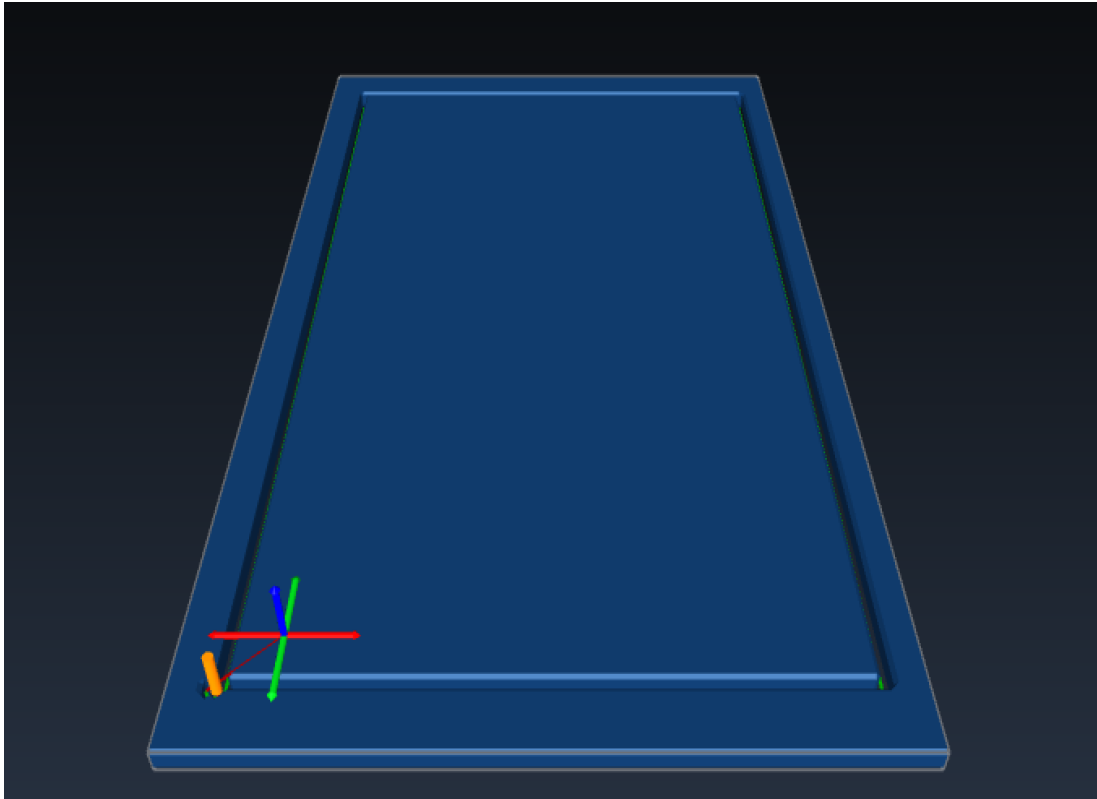


Figure illustrates the cut made by a CNC machine after executing the program.

## Additional Information

A G-Code viewer <https://camotics.org/#quick-start> for debugging.

G-Code:

- Introduction
  - <https://www.autodesk.com/industry/manufacturing/resources/manufacturing-engineer/g-code>
- CNC Machining
  - <http://www03.edu.fi/oppimateriaalit/nctekniikka/NCkoodit.html>
- Instructions
  - <https://en.wikipedia.org/wiki/G-code>