# GcodeDeck Manual

# Introduction

## Scope

GcodeDeck is an online conversational tool for the purpose of creating simple tool paths in Gcode for CNC mills or routers. Planning, analysis, and design for this project were completed as coursework for Systems Analysis and Design at UNCO in the fall 2021 semester. Implementation and testing for this project were completed in the Spring 2022 semester as a part of the Senior CIS Project Course, the capstone class for CIS and Software Engineering majors.

We built this system with the idea of making a simple tool that eliminates some of the time it takes to create simple tool paths. Because of this, the project is limited in capabilities to mostly linear movements, drilling, and some other templates that we have implemented. GcodeDeck is not meant to replace existing CAM packages, as the complexity that those allow is much greater than what our system can provide. However, we built GcodeDeck with an open-source mentality, and more functionalities can be added later.

## Who should use GcodeDeck?

Anyone can use GcodeDeck. It is geared towards people that already have a basic understanding of how Gcode works, but it is beginner friendly as well. Our target can range from existing machinists that want to just quickly pump out a simple tool path without having to boot up complex CAM software. Alternatively, beginners can use GcodeDeck to get their foot in the water with creating simple designs.

# Product Documentation

## User Guide

### Account Creation/Login

To use GcodeDeck’s Create a Program tool, our main functionality, you must first create an account and login. If you have an account, there is a login button on the main page that asks you to input your username and password. If you don’t have an account, you must create one first. Creating an account is a simple pop-up form that asks for a few pieces of information. The account registration form asks for a first and last name, a username that is 150 character or fewer which is letters, digits, and @/\_/+/- only. Additionally, it asks for an email, and a password that must be a mix of letters, numbers, and symbols. Password cannot be too similar to other personal information, nor be a commonly used password, or entirely numeric.

Once you have created a valid account, you will be automatically logged in. To log out, click on the account tab on the top right and you will have an option to logout or change your password.

Now that you are a user in our system, you can now access the Gcode generation by clicking on the Create a new program button.

### Creating a program

When you click on the Create a new program button, you will be directed to generation home page, the bread and butter of GcodeDeck. This page has buttons on the left side of the screen that render pop-ups for the user to put in parameters for the various Gcode Scripts that our program outputs. The generation buttons are as follows:

1. Tool Change
2. Spindle Command
3. Rapid Movement (G00)
4. Linear Movement (G01)
5. Drilling (G83 Peck Drilling)
6. Facing Template

#### Tool Change

Tool Change will pop up a form that asks for three parameters:

Tool number (tool pocket number)

Cutter Compensation

* You may choose from None (G40), Cutter Comp Left (G41), and Cutter Comp Right (G42)

Notes about tool (will output notes on the same line in parentheses, which machines ignore when reading Gcode)

#### Spindle Command

Spindle Command parameters:

Direction of Rotation:

* CW (M03)
* CCW (M04)

Spindle RPM

#### Rapid Movement (G00)

Rapid movement is a responsive form. You must first choose the axis of movement, and then set the coordinate for the end location of that movement on the selected axis.

Axis of movement options:

* X
* Y
* Z
* XY (input for both x and y coordinates)

#### Linear Movement (G01)

Linear Movement is the same as rapid movement, but also requires a Feed-rate parameter

#### Drilling

Peck Drilling Requires six parameters:

X – X coordinate of the hole

Y – y coordinate of the hole

Z – bottom of hole location

R – Top of tool part + some clearance (example .1”, this is also how far the drill will retract once the hole is finished. Make sure this value is higher by some margin than the highest point of your part relative to zero)

Q – Depth per Peck

Feed rate – Drilling Feed-rate

#### Facing Template

The facing template takes many parameters, so double-checking things like tool number, feed rate, and cutter diameter is recommended so you don’t ruin any material. The parameters are as follows:

Tool Number – Tool number for facing

Cutter Diameter

Spindle RPM

Feed Rate

Width – width along x axis for facing

Depth – depth along y axis for facing

Clearance – Z clearance for facing (top of part + clearance)

DOC – depth of cut (how much are you taking off the top?)

Plunge rate- Feed Rate for Z moves

Stepover – Amount tool moves over each pass until facing is complete

When any of the above commands are submitted, the website will generate Gcode based on input that is calculated behind the scenes. It is as simple as copying the outputted text and pasting it into whatever file that your machine is going to read. Additionally, if you wish to undo a command, redo it, or copy a command, there are buttons on the right side that will do those things for you. Congratulations, you’ve just created a custom program in minutes!

## Tutorial: Basic Program

This basic entry level tutorial will use the every function of the Gcode Deck application to produce a very simple part, but will demonstrate how each function is used. The part will be a single 2”x2” square with a hole drilled in the center. Obviously this demonstration is very simple, but the goal is to provide a short simple tutorial anyone can follow, and give the user the basic knowledge to make anything they can imagine.

1. Select a tool by using the tool change button
   1. Each time this button is pressed, it will instruct the machine to grab the designated tool
   2. VERY IMPORTANT that the correct tool is used per operation
      1. Make sure every program starts with a tool change and spindle command
   3. Example tool Change:

Graphical user interface, text, application, chat or text message

Description automatically generated

* 1. Example output:

Graphical user interface, text, application, chat or text message

Description automatically generated

1. Now that you’ve selected your tool, you must enable the spindle by designating RPM and Direction
   1. Spindle Example:

Graphical user interface, text, application, chat or text message

Description automatically generated

* 1. Output example:



1. Before you begin milling material, you need to go to your starting position, typically this will be using a rapid movement (G00)
   1. Disclaimer: if using cutter comp, your starting position needs to be at least a radius of tool diameter away from where you intend to cut, so that cutter comp has time on the control side to be enabled and enacted
   2. Rapid movement example:

Graphical user interface, text, application, chat or text message

Description automatically generatedGraphical user interface, text, application, chat or text message

Description automatically generated

* 1. Output example:





1. Next, we will outline a rectangle using G01 Linear Moves
   1. This will be done in four different uses of the Linear Movement Button, starting with a Z move to our starting cut depth of Z-.500”, and subsequent moves to build the outside profile of our part, and then return back to that starting position we used in the previous step.

Graphical user interface, text, application, chat or text message

Description automatically generatedGraphical user interface, text, application

Description automatically generatedGraphical user interface, text, application, chat or text message

Description automatically generatedGraphical user interface, text, application, chat or text message

Description automatically generatedGraphical user interface, text, application, chat or text message

Description automatically generatedGraphical user interface, text, application, chat or text message

Description automatically generatedGraphical user interface, text, application

Description automatically generated

* 1. Output:

Text

Description automatically generated

1. Next we’ll use the facing template to clean the top of our part. The facing template will call the next tool for us as well as enable the spindle, so we don’t have to worry about that this time.

\*\* In order to save space on this document we’re going to skip a screenshot from this form, but provide you with the following table of entered parameters.

|  |  |
| --- | --- |
| Facing Direction | Positive |
| Tool Number | 2 |
| Cutter Diameter | .500 |
| Spindle RPM | 6000 |
| Feed Rate | 45 |
| Width | 2 |
| Depth | 2 |
| Clearance | .1 |
| DOC (depth of cut) | .005 |
| Plunge Rate | 12 |
| Step Over | .4 |

Following the above inputs will produce the following lines of gcode for facing:

G54 G90 G17 G20 G40; (Safe Start)

M06 T2 ; (Switch to correct tool)

M03 S6000 ; (Turn on Spindle)

G00 X2.5 Y0 Z0.1 ; (Rapid Move)

G01 Z-0.005 F12 ; (Linear Move)

G01 X-0.5 F45 ; (Linear Move)

G01 Z.1 F12 ; (Linear Move)

G00 X2.5 Y0.4 ; (Rapid Move)

G01 Z-0.005 F12 ; (Linear Move)

G01 X-0.5 F45 ; (Linear Move)

G01 Z.1 F12 ; (Linear Move)

G00 X2.5 Y0.8 ; (Rapid Move)

G01 Z-0.005 F12 ; (Linear Move)

G01 X-0.5 F45 ; (Linear Move)

G01 Z.1 F12 ; (Linear Move)

G00 X2.5 Y1.2 ; (Rapid Move)

G01 Z-0.005 F12 ; (Linear Move)

G01 X-0.5 F45 ; (Linear Move)

G01 Z.1 F12 ; (Linear Move)

G00 X2.5 Y1.6 ; (Rapid Move)

G01 Z-0.005 F12 ; (Linear Move)

G01 X-0.5 F45 ; (Linear Move)

G01 Z.1 F12 ; (Linear Move)

G00 X2.5 Y2.0 ; (Rapid Move)

G01 Z-0.005 F12 ; (Linear Move)

G01 X-0.5 F45 ; (Linear Move)

G01 Z.1 F12 ; (Linear Move)

1. Lastly, we’ll switch tools like previously to tool 3, turn on the spindle to our wanted RPM (not shown) and fill out the drilling template as follows.

Graphical user interface, text, application

Description automatically generated

Output:

A screenshot of a computer

Description automatically generated with medium confidence

Now, if you’ve followed the above steps you can copy and paste the generated output to a simulation website such as ncviewer.com for verification. Doing so should result in an output that looks like this:

Diagram

Description automatically generated

Generated GCode for this tutorial can be found at the bottom of the document, if you’d like to copy and paste it into ncviewer and see a demonstration without having to go through the forms of this tutorial yourself.

## Other Features

We have implemented some other features that lend to the open-source aspect of our website, accessible through the community link at the top of the page. We have bug reporting, system requests, and a contact form.

### Bug reporting

To report a bug, simply click on the report a bug form and input your username, email, and the message about the nature of the bug.

### System requests

To request a new feature, simply click on the system request form and input your username, email, and the message about the feature you have in mind.

### Contact us

To contact us. simply click on the contact us form and input your username, email, and the message about the feature you have in mind.

TUTORIAL GCODE OUTPUT:

G54 G90 G17 G20 G40; (Safe Start)

M06 T1 ; (Load Tool #1 Notes: .500" Roughing Endmill)

G43 H1 ; (Load Positive Tool Height Offset for tool 1)

G41 D1 ; (Enable Cutter Comp)

M03 S6000 ; (Turn on Spindle CW)

G00 X-0.25 Y-0.25 ; (G00 Rapid Move)

G00 Z0.1 ; (G00 Rapid Move)

G01 Z-0.5 F30.0 ; (G01 Linear Move)

G01 X0.0 Y0.0 F30.0 ; (G01 Linear Move)

G01 Y2.0 F30.0 ; (G01 Linear Move)

G01 X2.0 F30.0 ; (G01 Linear Move)

G01 Y0.0 F30.0 ; (G01 Linear Move)

G01 X0.0 F30.0 ; (G01 Linear Move)

G01 X-0.25 Y-0.25 F30.0 ; (G01 Linear Move)

G54 G90 G17 G20 G40; (Safe Start)

M06 T2 ; (Switch to correct tool)

M03 S6000 ; (Turn on Spindle)

G00 X2.5 Y0 Z0.1 ; (Rapid Move)

G01 Z-0.005 F12 ; (Linear Move)

G01 X-0.5 F45 ; (Linear Move)

G01 Z.1 F12 ; (Linear Move)

G00 X2.5 Y0.4 ; (Rapid Move)

G01 Z-0.005 F12 ; (Linear Move)

G01 X-0.5 F45 ; (Linear Move)

G01 Z.1 F12 ; (Linear Move)

G00 X2.5 Y0.8 ; (Rapid Move)

G01 Z-0.005 F12 ; (Linear Move)

G01 X-0.5 F45 ; (Linear Move)

G01 Z.1 F12 ; (Linear Move)

G00 X2.5 Y1.2 ; (Rapid Move)

G01 Z-0.005 F12 ; (Linear Move)

G01 X-0.5 F45 ; (Linear Move)

G01 Z.1 F12 ; (Linear Move)

G00 X2.5 Y1.6 ; (Rapid Move)

G01 Z-0.005 F12 ; (Linear Move)

G01 X-0.5 F45 ; (Linear Move)

G01 Z.1 F12 ; (Linear Move)

G00 X2.5 Y2.0 ; (Rapid Move)

G01 Z-0.005 F12 ; (Linear Move)

G01 X-0.5 F45 ; (Linear Move)

G01 Z.1 F12 ; (Linear Move)

G54 G90 G17 G20 G40; (Safe Start)

M06 T3 ; (Load Tool #3 Notes: .25 Drill)

G43 H3 ; (Load Positive Tool Height Offset for tool 3)

(Cutter Compensation not enabled.)

M03 S6000 ; (Turn on Spindle CW)

G28 Z ; (Home Z to Prevent Crash)

G00 X1.0 Y1.0 Z-0.625 ; (Rapid to hole location @Z Reference Point)

G83 G99 Z-0.625 R0.1 Q0.05 F12.0 ; (G83 Peck Drill)

G80 ; (Cancel Canned Cycle)