AutoCAD plot to G-code Converter

Abstract

Numeric Control is the automated control of machining tools by means of a computer.This study aims to generate G-code from AutoCAD plot .An AutoCAD plot can directly be converted into G-Code .The plot is divided into smaller indivisible parts .We can have a detailed view of the plot with the help of DATAEXTRACTION command .We can export the file and can be processed to G-code without human interface.

Introduction

The primary function of CNC machine tool is to execute a sequence of multi-axis motions following part geometry.Before the actual machining process takes place, it is necessary to employ computer software packages that are capable of representing the physical and geometry of the part product[1].It is hardly possible to find open source software's used to simulate and generate G-codes for CNC milling.This paper aims to generate g-code from AutoCAD plot .

Methods and Data

Figure 1 shows the flow chart of generating g-code from AutoCAD plot. There are various steps to be followed for generating the g-code which are specified below.

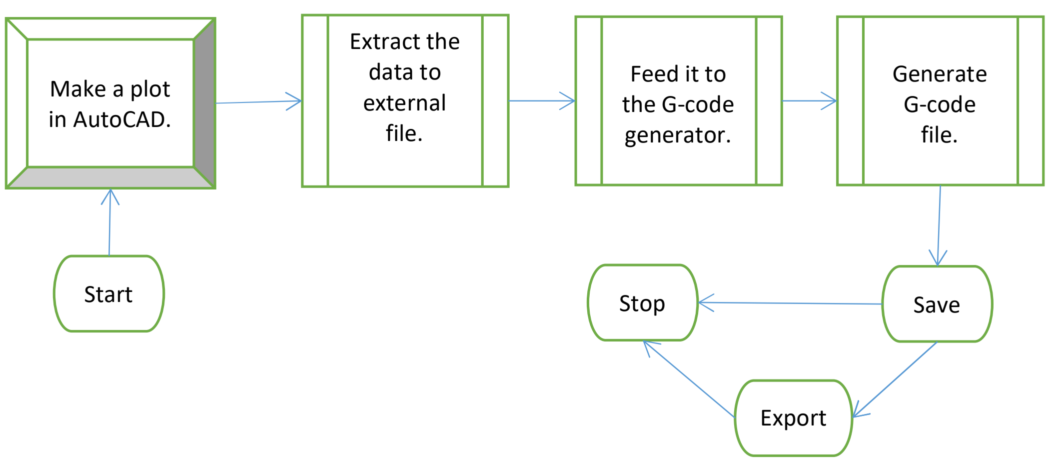


Figure 1: Flowchart of G-code generate process from AutoCAD plot.

STEP 1: An AutoCAD plot is made by using different commands provided in there such as "**polyline**","**spline**","**arc**","**circle**","**line**","**ellipse**".In these provided drawing commands polyline,spline and ellipse are divisible into even more smaller segments .Polyline is breakdown into smaller particles using "**EXPLODE**" command.

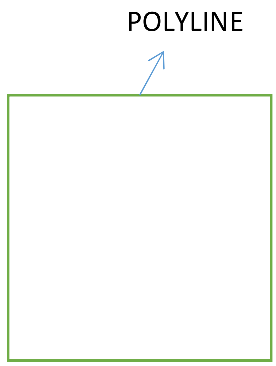


Figure 2: Before Explode

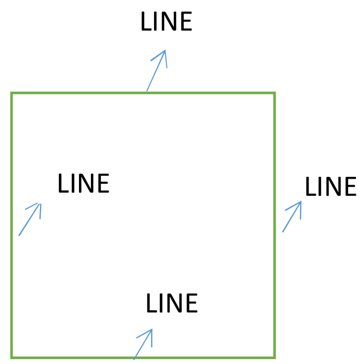


Figure 3: After Explode

To divide a spline into small indivisible parts firstly it must be changed into a polyline using "**PEDIT**" command .
After drawing a spline in the workspace and by entering a command PEDIT in the command line .

You can find the below .

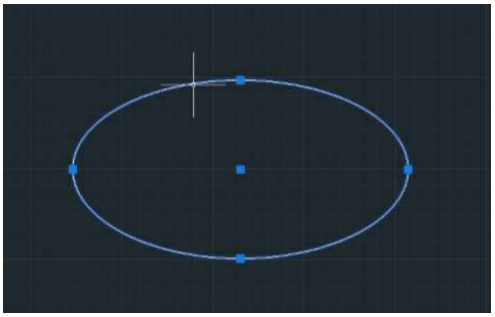
PEDIT Select polyline or[Multiple]: You need to select the spline which you want to convert .

PEDIT Do you want to turn it into one? <Y>: If yes "**Y**" or else "**N**".

PEDIT Specify a precision <10>:The precision value determines how accurately the resulting polyline is fit to the source spline. Enter an integer between 0 and 99. Note: A high precision value might cause slower performance.

Even Ellipse can be eventually divided into smaller particles by using the command **"PELLIPSE".**
Before drawing an ellipse in the workspace enter the command PELLIPSE which makes an ellipse in the form of arcs and then by using EXPLODE command you can divide an ellipse into arcs of different radius.

You can find the below after entering the command PELLIPSE in the command line.
PELLIPSE Enter a new value for PELLIPSE <0>:0 (as Figure 4) or 1 (as Figure 5).



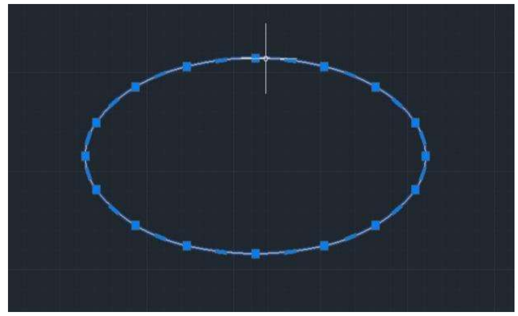


Figure 4: PELLIPSE= 0

Figure 5: PELLIPSE= 1

STEP 2: After making the AutoCAD plot, the file is saved and then the data is extracted by entering the command DATAEXTRACTION and hit enter then you will go through a series of steps.

The dataextraction wizard will open on the screen as shown in the figure 6.
Choose the Create a new data extraction option in the window, and then click on "Next >".

In the next window specify the name of the data extraction file and specify a location where you want to save it and click the Save button.

The next page i.e. page 2 of Data Extraction wizard has the Panel on top called "Data source" which involves choosing whether we want to extract data from the whole drawing, or only a specific part as shown in the figure 7.

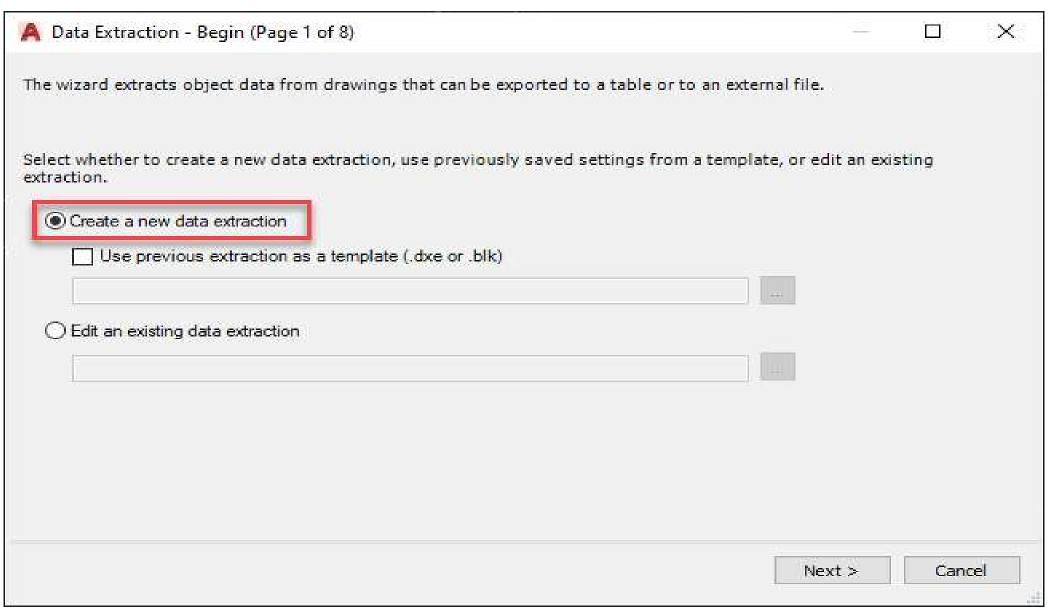


Figure 6: Data extraction begin page

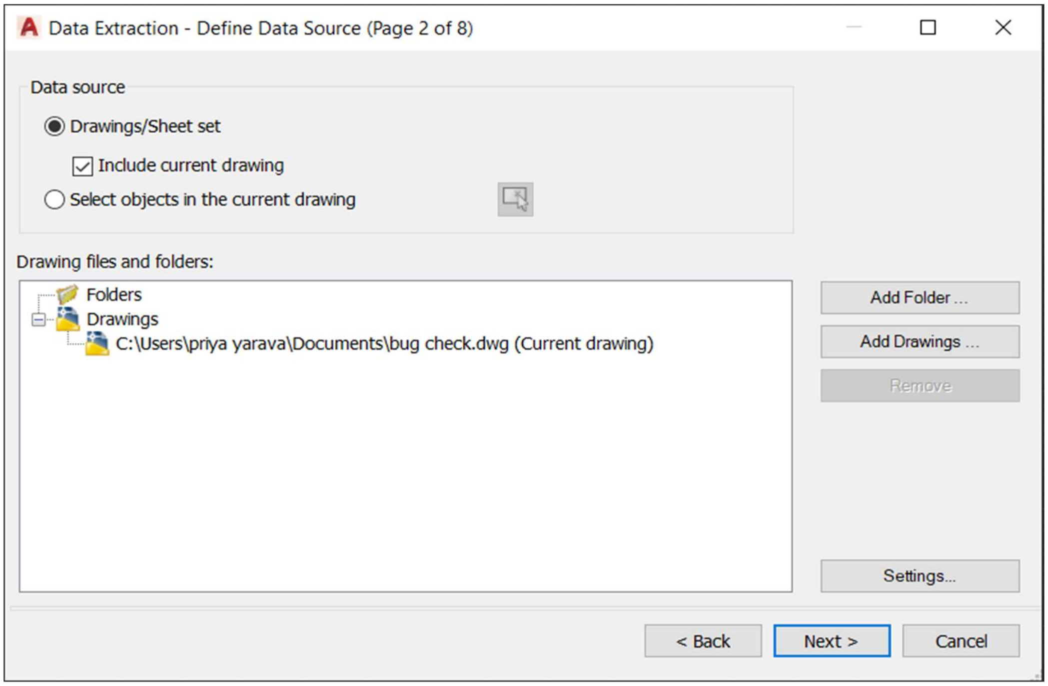


Figure 7: Defining data source

In this choose the same as shown in the figure 7 and click on “Next >”.

Then the Window will go through a loading process to a new window called Data Extraction – Select Objects (Pages 3 of 8 figure 8) as shown in the image below.

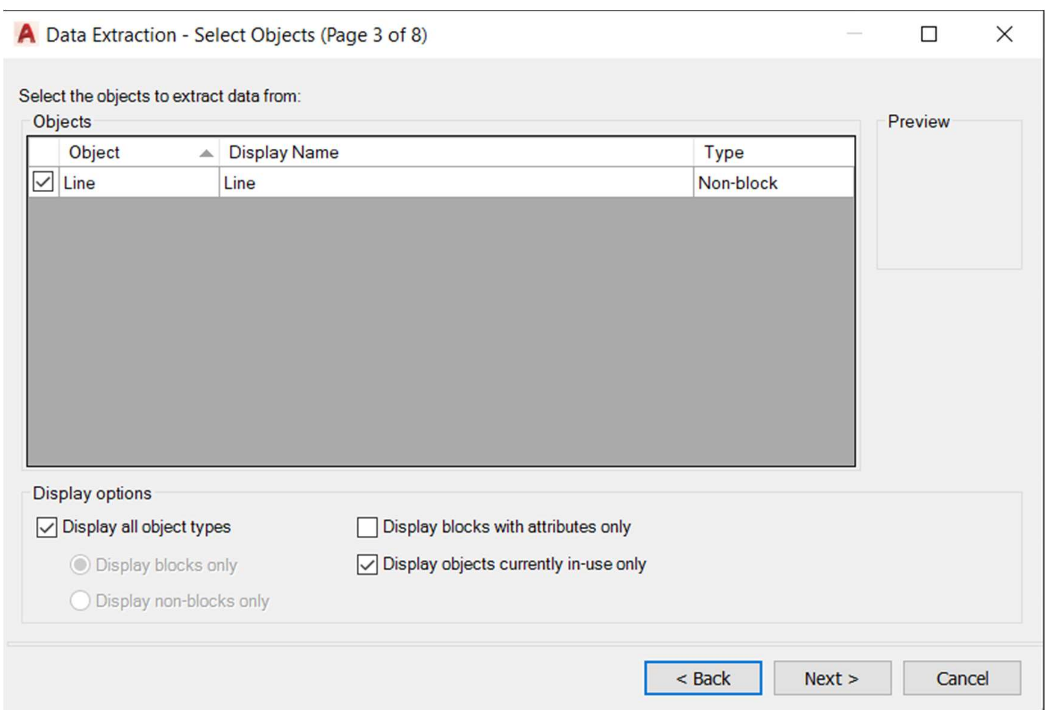


Figure 8: Data Extraction – Select Objects

Choose the objects that are needed and leave the rest and click “Next >”.

Now you can see a list of properties of the selected objects as shown in the figure-9 .By unchecking in the properties those properties will not appear in the final data file.

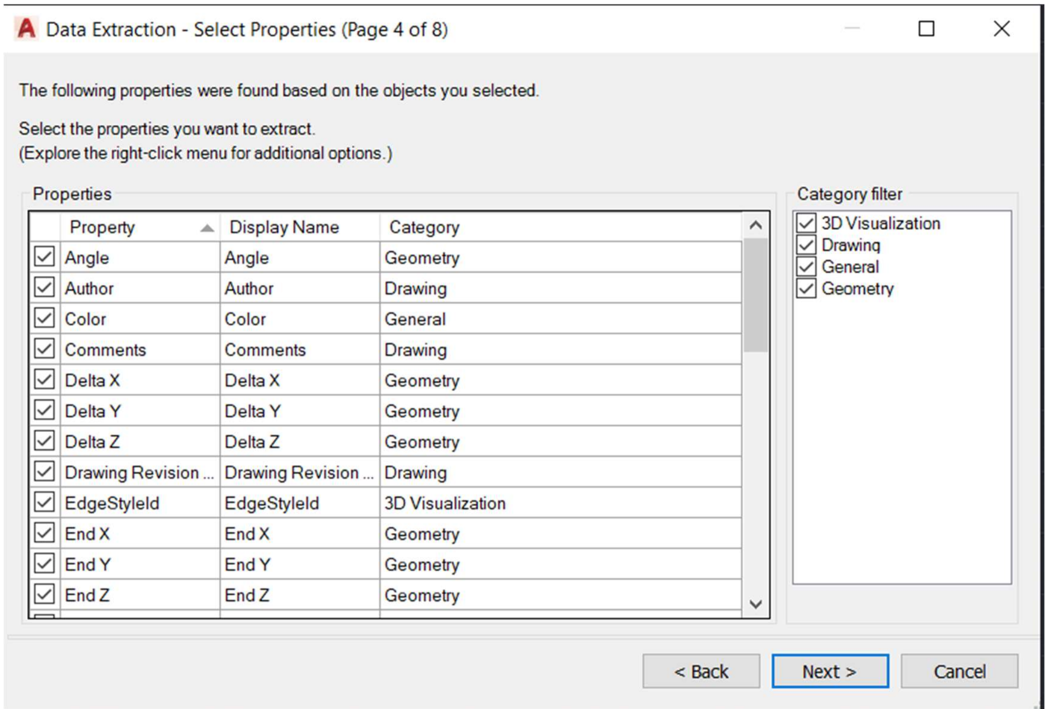


Figure 9: List of properties of selected objects

After clicking “Next >” you will move to the page which will show you the final table as shown in the figure 10.

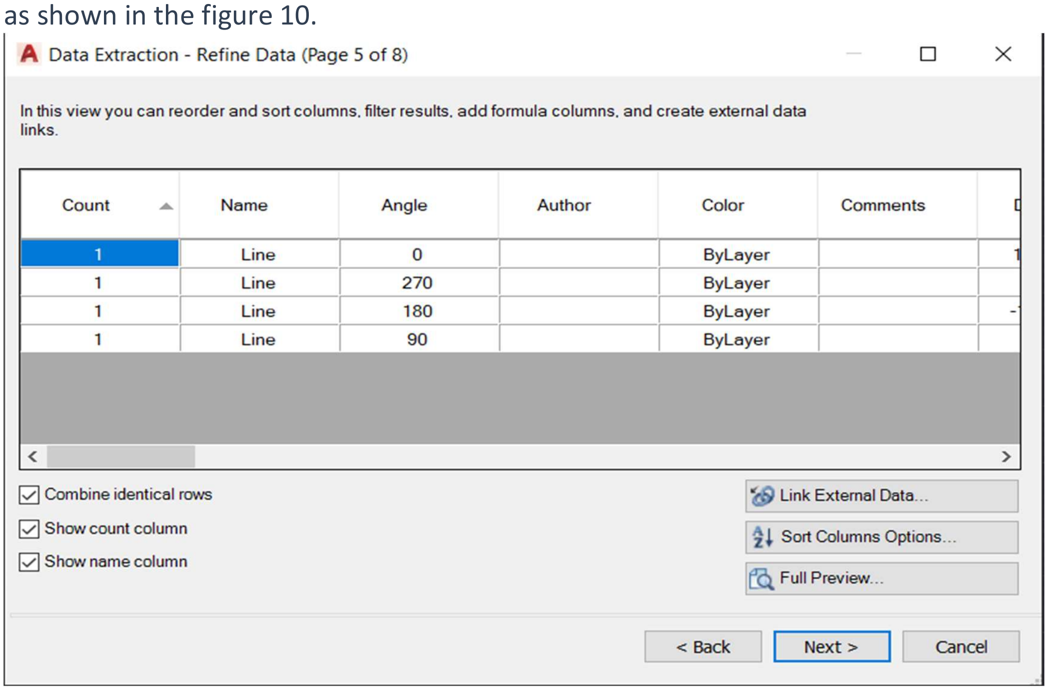


Figure 10: Final list of Data table

After verifying the final list of data click enter we will reach the page 6 of dataextraction wizard.On page 6 as shown in the figure 11, you will only see two checkboxes.
The first checkbox "Insert data extraction table into drawing" will let you make an AutoCAD table right inside the drawing.
The second option "Output data to external file" will let you export the data extraction data into an external file like XLS(excel file).

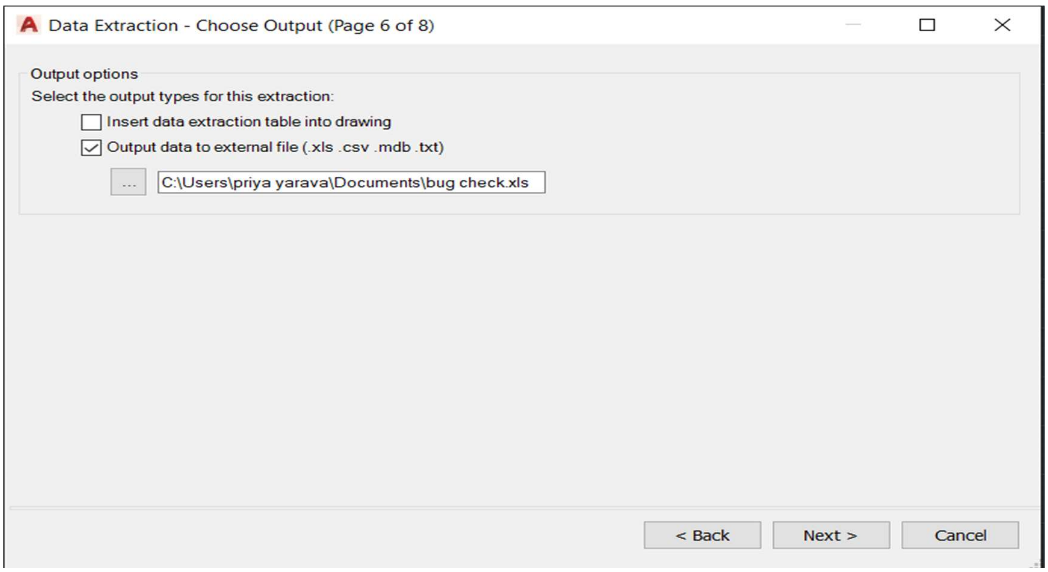


Figure 11: Output selection page

After selecting the location of saving the excel data file then click "Next >" then you will be directed to a confirmation page where you are asked to whether finishing the dataextraction or end and go back, click on finish and an excel file will be generated at the location you have provided.

After making the plot in workspace and then extracting the data with steps discussed before(Step 1 and Step 2).
Step:3 Now it is time to feed the data to G-code generator .The G-code generator is implemented using python version 3 and runs on a computer using Windows7 and above, as operating system.G-code generator is actually designed to take the input that is extracted by following above discussed steps and can process into g-code without any human interface.

This G-code converter mainly takes in the parameters on which CNC milling depends such Speed of spindle ,Rotation of spindle . G-code generator actually comprises of a set of code which can check the type of plot such as line ,arc or circle and gives the output in the form of g-code.

Let us consider a sample case of making a square of length 100, the data after extraction from AutoCAD is as shown in below figure 12.

	A	B	C	D	E	F	G	H	I	J
1	Count	Name	Angle	End X	End Y	End Z	Length	Start X	Start Y	Start Z
2	1	Line	180	0.0000	100.0000	0.0000	100.0000	100.0000	100.0000	0.0000
3	1	Line	270	0.0000	0.0000	0.0000	100.0000	0.0000	100.0000	0.0000
4	1	Line	0	100.0000	0.0000	0.0000	100.0000	0.0000	0.0000	0.0000
5	1	Line	90	100.0000	100.0000	0.0000	100.0000	100.0000	0.0000	0.0000
6										
7										

Figure :12: Output data file

G-code generator looks for the cell which is equal to “Name” and stores the entire data underneath it in the list and in the same way it searches for required name in the first row and store the rest of the data.

The program is as follows

```
name=[]  # initializing a list with name as “name”
for i in row1:  #iterating over the whole row1
   if i==”Name”:  #if the data which we are looking is found then it allows to store the
data underneath it in the list
      for j in range(len(columnB)): #iterating over coloumnB
        name.append(columnB[j]) #stores the data at each iteration
```

After storing all the data the G-code is generated as follows:
Initially it asks the human some parameters by which it selects the set of common code to be in the output file.

```
if unit == "mm":
   if feedunit == "mm":
      print("G21 G94")
   elif feedunit == "inches":
      print('G21 G93')
elif unit == "inches":
   if feedunit == "mm":
      print('G20 G94')
   elif feedunit== "inches":
      print('G20 G90')
print("G91 G28 X0 Y0 Z0")
print("M06 T0", Toolnumber)
if directionofspindle == "clockwise":
   print("M03 S", Speedofspindle)
elif directionofspindle == "counterclockwise":
   print("M04 S", Speedofspindle)
```

After the common set of programming lines.The G-code for the actual plot is made by iterating over the list “name”.

```
for i in name:
  if i=="Line":#Searching in the list for line
     print("G00 X{0} Y{1}".format(start[i],star[i]))#getting the corresponding value of
        starting points of line
     print("G01 Z{0}".format(z22))#Dropping the tool to required depth
     print("G01 X{0} Y{1}".format(x2[i], y2[i]))#getting the corresponding value of
        ending points of line
  if i == "Arc":
    if directionofarc=="Clockwise":
      print("G02 X{0} Y{1} R{2}".format(startx[i], starty[i], radiusarc1[i]))
      print("G02 X{0} Y{1} R{2}".format(endx[i], endy[i], radiusarc1[i]))
    if directionofarc=="CounterClockwise":
      print("G03 X{0} Y{1} R{2}".format(startx[i], starty[i], radiusarc1[i]))
      print("G03 X{0} Y{1} R{2}".format(endx[i], endy[i], radiusarc1[i]))
```

‘’’ For an arc we get the center point , radius ,starting point and the ending point so we must check whether a clockwise arc to be drawn or an anticlockwise arc to be drawn these set of code converts the information of starting angle and ending angle into starting points of arc into ending points of arc by using the formula x=rcos(θ) y=rsin(θ) in which r is the radius and θ is the angle made at the point.’’’

```
  if  i == "Circle":
    x = cx1[i] + r1[i]
    y = cy1[i]
    X = cx1[i] - r1[i]
    print("G03 X{0} Y{1} R{2}".format(X, y, rad[i]))
    print("G03 X{0} Y{1} R{2}".format(x, y, rad[I]))
```

‘’’ For a circle we get the information of radius , center . Here the given information is converted to 2 extreme points of the line joining the center and the circle .Here the ordinate would be the same only the abscissa gets changed the first abscissa is obtained by x=cx+r, X=cx-r where cx is the abscissa of center point .x is abscissa of one end and X is abscissa of other end.’’’

RESULT

Before we feed the data to G-code generator it asks us the basic information regarding the tools ,direction of spindle,Speed of spindle ,depth of cut,feed unit and the location of data file.

Firstly it asks for the number of tools as shown in the figure

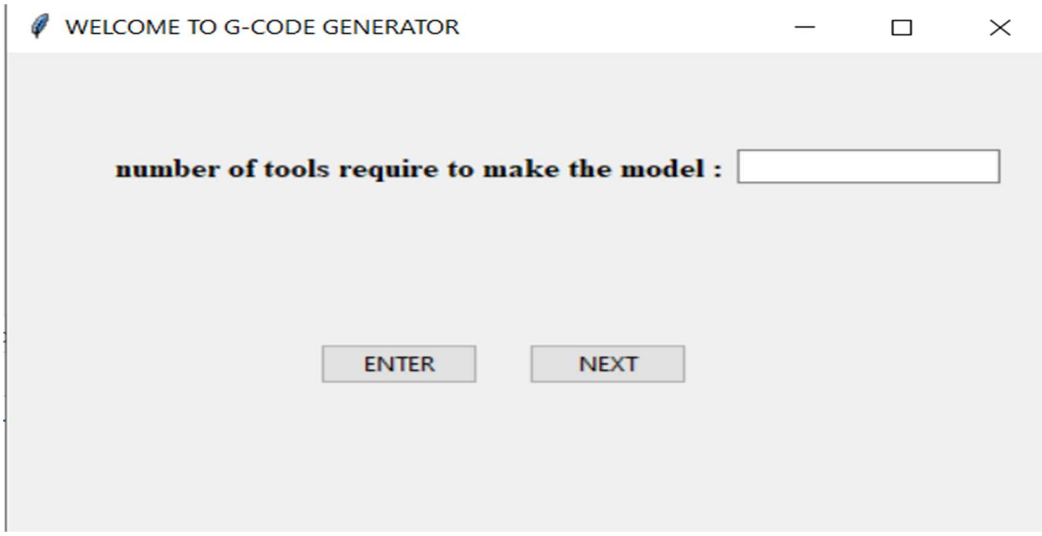


Figure 13 : Window to input number of tools

After entering the number of tools a window shown below pops up.

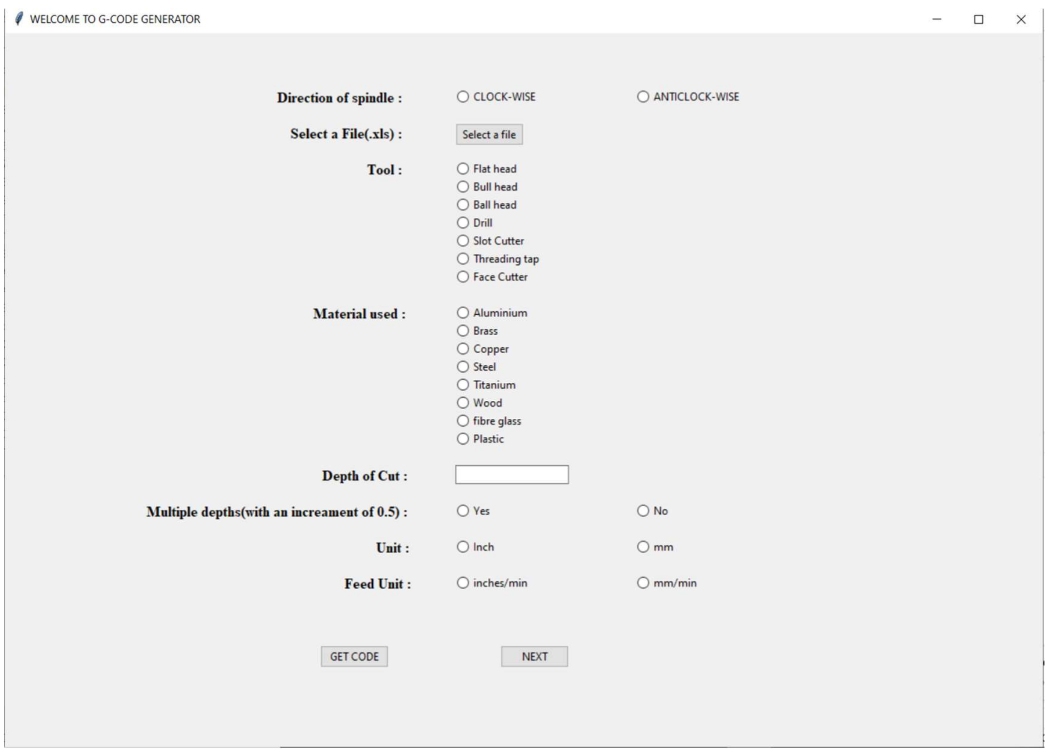


Figure 14 : Window for inputting various parameters regarding the tool

It(figure14) pops up for the number of times you have entered in the entry box shown in figure13 because each tool has the different path to be followed and different parameters when compared with other tools.

After entering and uploading the output excel file,we can obtain the G-code for it as shown below.

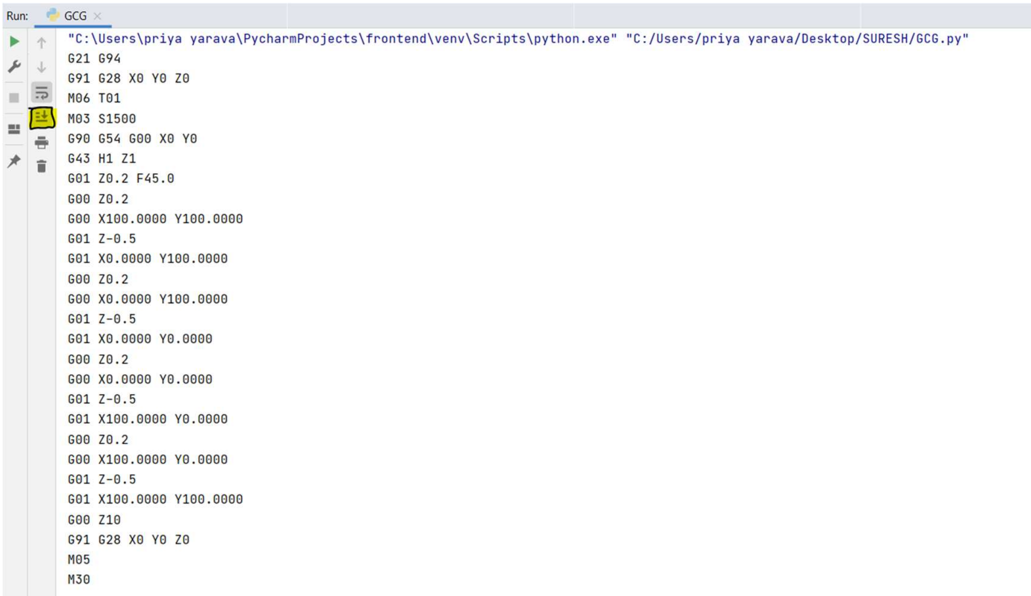


Figure 15: Output G-code file

You can download and save the Output obtained for further references and also to feed into the machine.

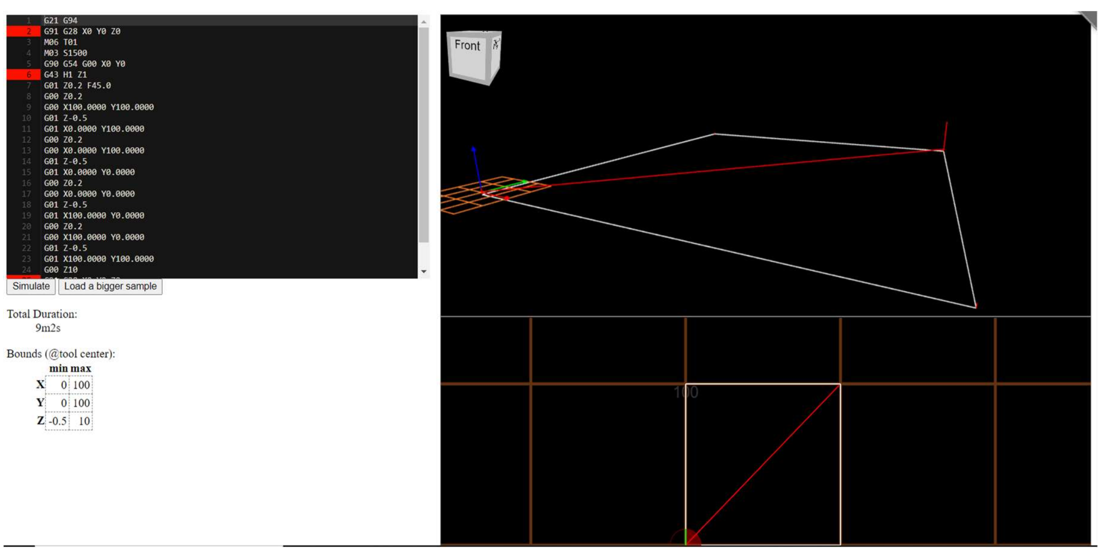


Figure 16: Path view of output G-code