Lab 05: CNC Lathe Programming

I. Purpose
    A.)   Provide an introduction to manual CNC programming
    B.)   Provide experience designing and programming parts for production on the laboratory's CNC equipment

II. Practice
Your lab team is to produce a CNC turning program file for your rotational project parts in preparation for automated production. The CNC lathe is not currently programmable by using the MasterCAM program, since the dimensions of our parts are so small. You are to document the process in your engineering notes.

The stock material for your parts will be either red oak or ash wood, up to 4.750" long and 0.500" in diameter. At least one end of the stock will need to be faced. 0.625" of the stock will be held within the chuck, and the parting operation will consume approximately 0.0875" of stock You will be doing the parting operation several times – following the plunge speed requirement will allow you to avoid an extra facing operation and further loss of stock material! You will only be able to get a few parts from each piece of stock. See the illustration, below.
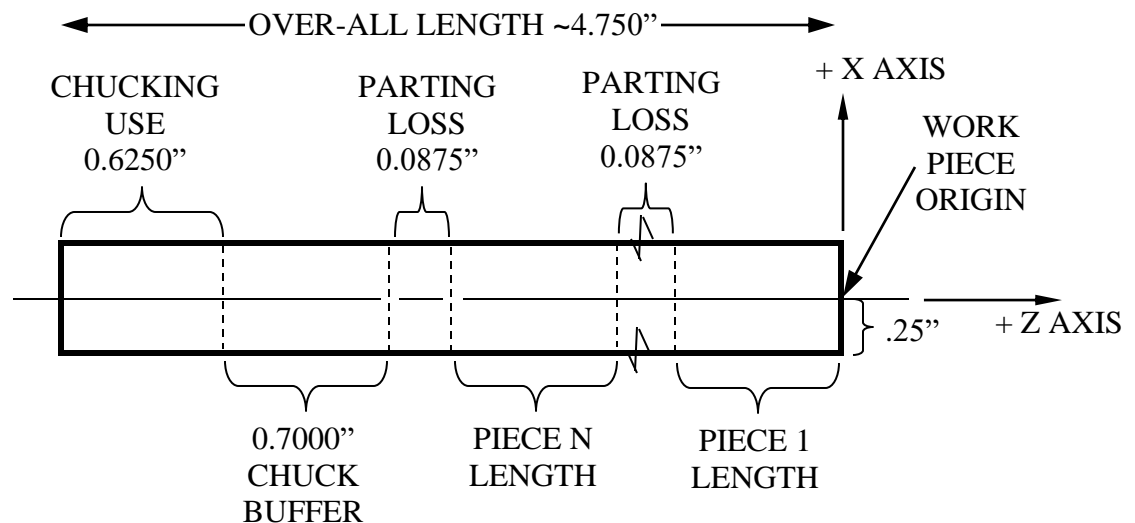

Figure 1. Allocation of Stock Material and Workpiece Coordinate System.

Depending upon your design, you may need to utilize some of the following tools. The tools and their tool numbers are listed in the table below. Tool one (T01) is used for the touch-off operation. **All tool changes must take place 0.5" above the top edge of the workpiece and 0.75" from the current end of the workpiece in the + Z direction**. Watch for tool holder clearance, plunge and feed rate limitations, and avoid rapid travel (G00) within the workpiece boundaries. You may use absolute and/or incremental coordinates as you wish.

Table 1. Tooling Magazine for Lab

Tool	Description
T01	Right hand profiling tool – 55º Diamond Insert (limit feed rate to 4 ipm)
T03	Right hand parting tool - .0875 wide (limit plunge feed rate to 1 ipm)
T05	Left hand profiling tool – 55º Diamond Insert (limit feed rate to 4 ipm)
T07	Grooving tool – Square Insert (90º, limit plunge depth to .02" at feed rate of 1 ipm)

Your first few cuts will be a partial facing cut to obtain a smooth piece end of diameter .500" and at least two cuts to produce part end marks that are .05" deep (to mark the visual ends of the parts and avoid tooling cuts into the chuck buffer zone). You will change from T01 to T03 to make these cuts. The next cuts will be roughing cuts to obtain the maximum diameter of 0.400". These roughing cuts, and the finishing profile cut will use T01 – cutting from right to left, and/or T05 – cutting from left to right. T07 may be used to cut v-shaped grooves, and T03 will be used again to part the pieces from the remaining work stock. The lab instructor will provide you with the initial and final CNC code sections for your part(s) on the Materials page of the course website. Each person on the team is responsible for helping to create the CNC program for their pieces. A member from your team may have to write code for a set of pieces that will be part of the team demo. Prepare your code manually, and your final program as an MS-DOS text file, then rename it with the extension **.NC** in your team's jump drive folder. The program file must be verified before production.

Program Requirements
1.) Your program must create at least two parts – the parts do not have to be the same.
2.) Your program must have at least one circular interpolation ≥ 45º of arc on each part.
3.) Your program must have at least one groove on each piece, using T07 or T03.
4.) All tooling changes must take place **safely** (see bold note on p. 1).
5.) All programs must be verified by the TA / Instructor on the specific machine before cutting.

When your individual parts are cut, tape them together with masking tape and put your initials on the tape, then place the pieces into your team box. Save your NC file in your team folder on the jump drive, with the name ZXXX.nc (where the Z is your first initial and XXX is your last name). There is no lab report required for this lab.
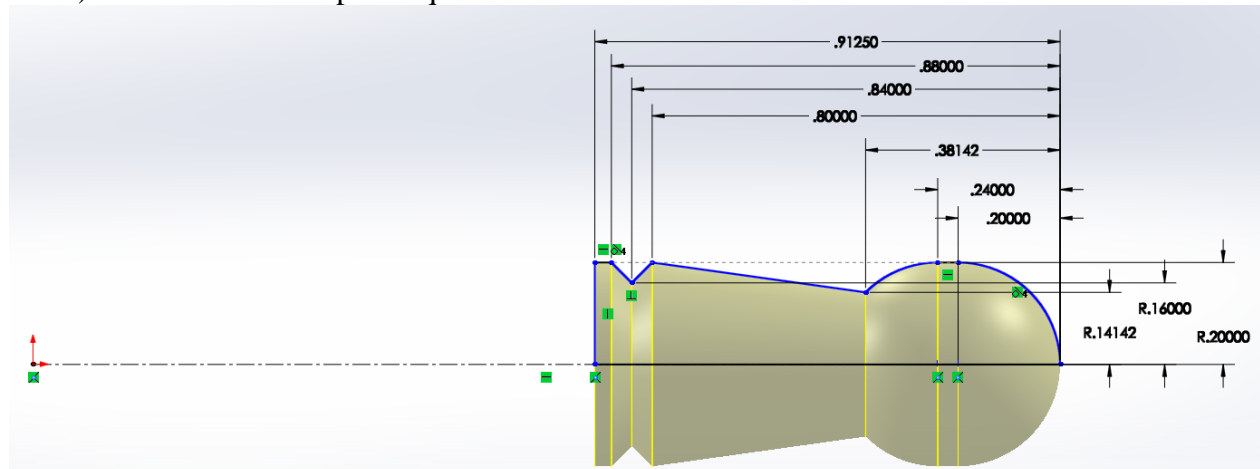
Example Program for a Pawn.

The code below will make a pawn piece (see above figure). Note that the workpiece *origin* is on the centerline of the workpiece (NOT at the top, right corner of the part – that's the touch-off!).

```
%
(SET UP CODE )
N10 G90 G80 G40 F4 S2000 T01 M06
N20 G00 X.5 Z.75 M03

(PIECE ONE STARTS AT Z0.0 / PAWN CODE GIVEN / MAY REPLACE CODE STARTING HERE)
(FACING CUT)
N30 T03 M06
N40 G00 X.3 Z0 F1   (After tooling change, move close to, but clear of, the workpiece to adjust coords)
N50 G01 X-.1
N60 G00 X.3
(FIRST GROOVING CUT – 1 INCH WARNING MARK)
N70 G00 Z-1.3375
N80 G01 X.2
N90 G00 X.3
(SECOND GROOVING CUT – CHUCK BUFFER WARNING MARK)
N100 G00 Z-3.45
N110 G01 X.2
N120 G00 X.75
N130 G00 Z.75
(FIRST ROUGHING CUT - HORIZONTAL)
N140 T01 M06 F4
N150 G00 X.2 Z.05   (After tooling change, move close to, but clear of, the workpiece to adjust coords)
N160 G01 Z-1.3375
N170 G00 X.75 Z.75
(SECOND  ROUGHING  CUT – ARC RELIEF)
N180 T07 M06 F1
N190 G00 X.3 Z-.02   (After tooling change, move close to, but clear of, the workpiece to adjust coords)
N200 G01 X.14
N210 G00 X.3
(GROOVE AT BASE)
N220 G00 Z-1.00
N230 G01 X.16
N240 G00 X.75
N250 G00 Z.75
(LEFT HAND LINEAR CUT / 45 DEGREE CUT)
N260 T05 M06 F4
N270 G00 X.3 Z-.8   (After tooling change, move close to, but clear of, the workpiece to adjust coords)
N280 G01 X.2
N290 G01 X.14142 Z-.38142
N300 G01 X.2 Z-.24
N310 G00 X.75 Z.75
(RIGHT HAND 90 DEGREE ARC)
N320 T01 M06
```

N330 G00 X.25 Z.15   (After tooling change, move close to, but clear of, the workpiece to adjust coords)
N340 G01 X0 Z0
N350 G02 X.2 Z-.2 I0 K-.2
N360 G00 X.75 Z.75
(PARTING CUT / PIECE 1)
N370 T03 M06 F1
N380 G00 X.3 Z-1.3375 (After tooling change, move close to, but clear of, workpiece to adjust coords)
N390 G01 X-.1
N400 G00 X.75
N410 G00 Z-.25
(END OF PIECE ONE CODE)

(PIECE TWO STARTS AT Z-1.3375 / INSERT CODE HERE)

Add code for Roughing passes, leaving some material for finish (as per the diagram, again). Then get set for the finish profile cut.
Add code for the Finishing profile pass. Cut to the start of any edges or cut to the start of an arc.   Add the code for any circular interpolation; then make any other cuts to the end of the first piece. Then move to a safe location for a tool change to T07 or T03.
Make the tool change, then line up for the grooving cuts. Remember the grooving plunge cuts at slower feed rate.   Make the groove cut(s), then move to a safe location (again) for a tool change to T03 (for parting).
Write the tool change code, and then make the final parting cut(s) (at a slow feed rate) right at the workpiece edge. Plunge the cutter to an X value of -.2 to make the smoothest end possible. Repeat as necessary for other part(s).

(ENDING CODE / KEEP THIS CODE TO 5S THE LATHE)
N1400 G00 X.75
N1410 G00 Z.75 M05
N1420 T01 M06
N1430 G28
N1440 M02 M30
%

Save your file to your team folder/drive and verify it with the Instructor before cutting material!