2.008
Design & Manufacturing II

The CAD/CAM Labs

Lab I
Process Planning
G-Code
Mastercam Lathe

Lab II
Mastercam Mill
Check G-Code

Lab III
CNC Mill & Lathe Machining

Labs I, II, III

OBJECTIVE

These lab exercises will introduce you to process planning and the tools you will need to carry through on such a plan. In the end (lab III) you will produce a machined paperweight with your own unique inscription. You will become familiar with a CNC lathe and a CNC mill, as well as Mastercam, a computer-aided manufacturing software package. You will also learn G-code, the alphanumeric programming language used to control CNC machine tools.

BACKGROUND

Process planning is an engineering activity that determines the appropriate procedures for transforming raw materials into a final product as specified by an engineering design. Engineering designs are conventionally documented using detailed diagrams indicating important design characteristics such as dimensions, tolerances, materials, and other pertinent specifications. Even though these diagrams convey a large amount of information about a design, they are incomplete in that they do not describe the manufacturing steps necessary to produce the final part. Effective process plans provide this information. In practice, design, process planning, and manufacturing are interrelated since the capabilities and characteristics of available equipment, manufacturing processes, and personnel can have a significant impact on the final design of a product.

G-code

When the designed product is produced using CNC machine tools, the machine tool controller needs explicit instructions describing the type and order of individual steps required to perform a given task. These instructions are provided to the controller in the form of alphanumeric codes (G-code) and are usually called a part program. A complete listing of G-code commands and a set of example code/part pairs can be found in the appendices.

G-code itself can be written with any number of text editors, including the Windows notepad and, of course, emacs. After the code is verified with a simulation, it is loaded into one of the CNC machines.

Mastercam

Mastercam is a commercial Computer Aided Manufacturing software package. CAM is the process that links CAD (computer-aided design) with machine tools by automatically producing CNC code from a drawing. Mastercam includes a CAD environment, a geometry-to-G-code translator, and a G-code simulator. It is possible to import geometry from other drawing tools like AutoCAD, SolidWorks, or Pro/ENGINEER.

LAB EXERCISES

In the next two labs, you will obtain an aluminum disc 3" in diameter and .5" thick (Fig. 1). Inscribe your own unique pattern on the top with the CNC milling machine (Fig. 2), and turn a custom border/facial profile on the remainder of the disk with the CNC lathe (Fig. 3).

All exercises will be conducted with a partner - each pair must turn in only one part and will receive a common grade.[1] You and your lab mate may need to meet outside of class in order to complete some of these assignments. If you are feeling adventurous, you and your lab mate may produce two different paperweights. They will be graded separately.
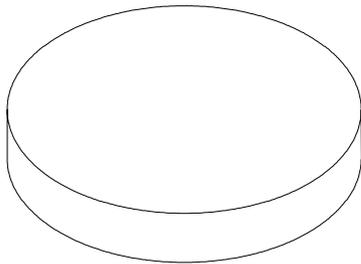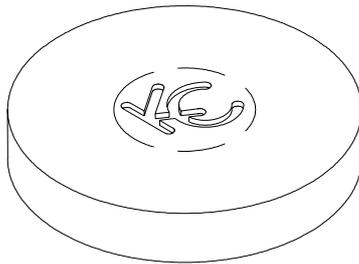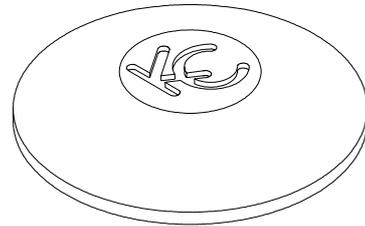
| Fig. 1 | Fig. 2 | Fig. 3 |

During **Lab I**, you and your lab mate will:
- learn how to write G-code so that you can manually create your own part programs
- run a series of tutorials designed to teach you how to use Mastercam Lathe
- see Mastercam create the G-code for a similar profile as shown in Fig. 3
- use Mastercam to draw the lathe profile of your paperweight (the top drawing can be done by hand or any software)
- use Mastercam to run a toolpath on the profile
- prepare for next week's lab

During **Lab II** you and your lab mate will:
- learn how to use Mastercam Mill
- check your pre-prepared handwritten G-code program for the mill portion of your paperweight
- prepare for next week's lab

During **Lab III** you and your lab mate will:
- finalize the drawings (combining top and side views)
- finalize your pre-prepared handwritten G-code program for the mill portion of your paperweight
- run your program on the CNC mill to inscribe your unique symbol into the center of your disc
- use the CNC lathe to run the profile program to turn/face the shape on your paperweight

---

[1] If you feel that your lab mate is not performing his/her fair share of the work please contact the TA to discuss grading arrangements.

DELIVERABLES

This page explains what you must have completed by a week after each lab session. See the "Grading Procedures" section of the webpage for more details.

DELIVERABLE - Lab I

1) A 2-D drawing of the inscription to be machined on the top of your paperweight. This should be either hand-drawn on graph paper or computer-drawn and dimensioned.
2) A handwritten G-code program of your unique inscription. A Mastercam Mill generated program is cheating!
3) A separate detailed cross-sectional sketch of the side view showing the profile shape that will be machined on the lathe. This should also be properly dimensioned and neat.

Things to think about:

- You should have the handwritten program on a 3.5" floppy, or a Zip disk, (100MB, or 250MB) if you work on it at home so that you will be ready to debug and run at the beginning of Lab III.
- The program should be typed in using Windows Notepad. It must have a filename extension ".txt".
- Keep in mind that your inscription should fit within a 2.0" diameter circle to allow for machining of the profile on the perimeter.
- Your milled inscription and profile shape must be unique.
- When dimensioning, be sure to reference everything from the center and top of the disk.
- Don't forget Z depths!
- Your inscription must have at least one arc or circle - all straight lines are not allowed.
- Remember the diameter of the end mill (it must fit between features) before you try anything too fine and detailed.
- Remember to consider how the disk will be held in the chuck on the lathe so that you don't machine into the jaws of the chuck - leave at least .200" of uncut material at the bottom.
- Remember to consider the shape of the bit and the angle from which it will cut.
- An isometric view of your part is purely optional.

DELIVERABLE - Lab II

1) Finalized handwritten G-code for mill.
2) Mastercam generated lathe toolpath for the profile view.

DELIVERABLE - Lab III

1) A completed paperweight, with both the milled inscription and turned profile.
2) A hardcopy of the final handwritten program of the milled inscription if it was changed since Lab II.
3) A Mastercam drawing of the final output of the entire paperweight, with both the top view and the side view on the same drawing, completely dimensioned.

APPENDIX A: Producing the Sample Part

Step I: Milled Inscription
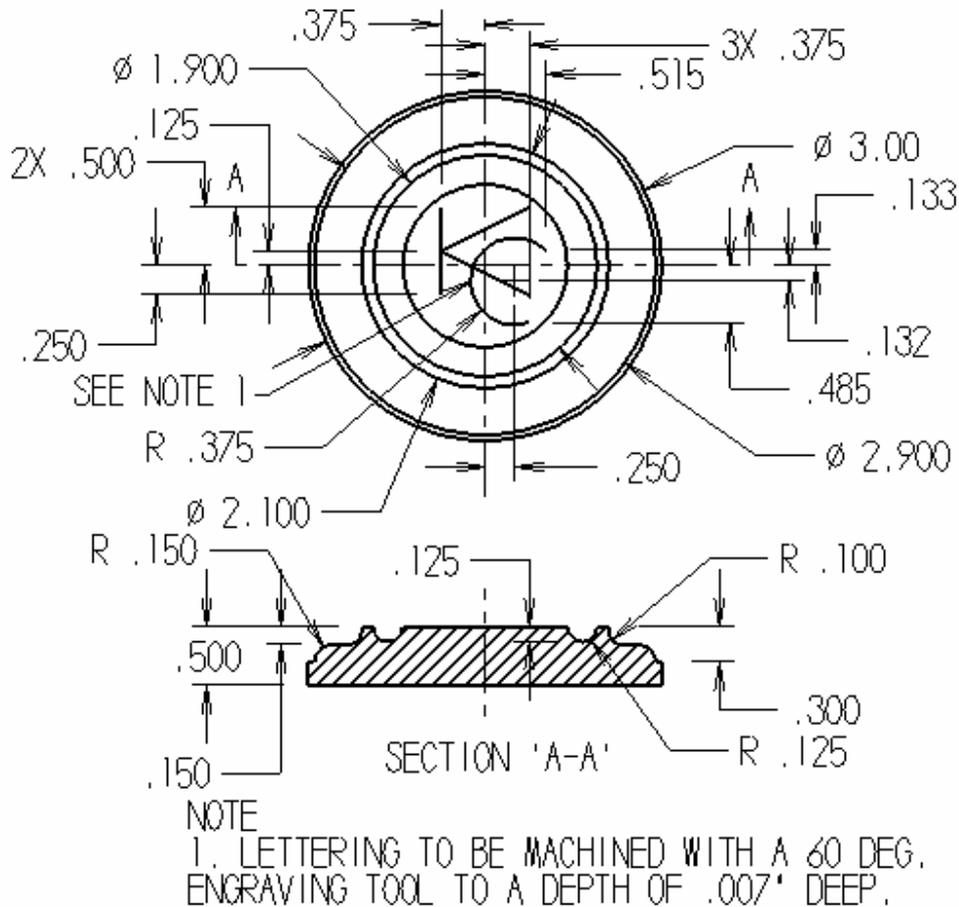
In this example the Z0 has been established below the top of the raw stock by .010". The first tool in the program is a 1.5" diameter end mill (T11) that will face off the top of the raw stock to establish a plane at Z0. It will remove all imperfections from the original rough surface to create a uniform flat surface parallel to the machine's X, and Y planes. It is important to note that the parallelism of the surface to the machine's axes is critical for the engraving to be uniform in depth and width of cut. The engraving tool (T14) is then used to engrave the letters "KC" at a depth of -.007" into the face, followed by a 1/4" ball end mill (T8) to machine a circular groove around the letters "KC".

The following is the program. The 2nd column is only to describe what each line, or "block" is doing and is not used by the control on the machine.

	Return the X, Y, and Z-axes to a predetermined clearance point for changing tools.
N1M26T11S2000	Install tool T11 (1.5" end mill) for facing top surface, spindle speed 2000RPM
N2G0G55G75G90X-2.0Y-1.3	Rapid positioning mode, designate work coordinate system 1, multi-quadrant circle mode, absolute positioning mode, move to the X, and Y values,
N3Z.1	Rapid position Z to the clearance plane, .100" above the part.
N4G1Z0.F5.	Linear feed to Z0 at 5" per minute
N5X2.0F10.	Linear feed to X value at 10" per minute
N6X2.4Y0.	Compound linear feed movement, still at 10" per minute
N7X-2.4	Feed to next point
N8X-2.0Y1.3	Feed to next point
N9X2.0	Feed to next point
N10G0Z.1	Rapid position Z up to clearance plane
	Return the X, Y, and Z-axes to a predetermined clearance point for changing tools.
N11M26T14S4000	Tool change, T14. (Engraving tool), spindle speed 4000RPM
N12G0G55G75G90X-.375Y.5	Rapid positioning mode, designate work coordinate system 1, multi-quadrant circle mode, absolute positioning mode, move to the X, and Y values,
N13Z.1	Rapid position Z down to clearance plane
N14G1Z-.007F5.	Feed Z into part to a depth of -.007" at 5.0" per minute
N15Y-.25F10.	Feed Y to -.25 at 10" per minute
N16G0Z.1	Rapid Z up to clearance plane
N17X.375Y.5	Rapid position to start point of next cut
N18G1Z-.007F5.	Feed into part
N19X-.375Y.125F10.	Feed to next point
N20X.375Y-.25	Feed to next point
N21G0Z.1	Rapid Z up to clearance plane
N22X.515Y.1333	Rapid position to start point of next cut
N23G1Z-.007F5.	Feed into part
N24G3X.25Y.243R.375F10.	Circular interpolate CCW to the first quadrant of the circle. R is the radius of the circle.
N25X-.125Y-.132R.375	Continue to circular interpolate to the next quadrant
N26X.25Y-.507R.375	Continue to circular interpolate to the next quadrant
N27X.375Y-.4855R.375	Continue to circular interpolate to the end of the circle
N28G0Z.1	Rapid Z up to clearance plane
	Return the X, Y, and Z-axes to a predetermined clearance point for changing tools.
N29M26T8S3000	Tool change, T8. (.250 ball end mill)
N30G0G55G75G90X.825Y0	Rapid positioning mode, designate work coordinate system 1, multi-quadrant circle mode, absolute positioning mode, move to the X, and Y values
N31Z.1	Rapid Z down to clearance plane
N32G1Z-.125F5.	Feed Z-axis into part to a depth of -.125" at 5.0" per minute
N33G3X0.Y.825R.825F5.	Circular interpolate CCW to first quadrant of circle, at 5.0" per minute
N34X-.825Y0.R.825	Circular interpolate CCW to next quadrant
N35X0.Y-.825R.825	Circular interpolate CCW to next quadrant
N36X.825Y0.R.825	Circular interpolate CCW to next quadrant
N37G0Z.1	Rapid Z up .100 to clearance plane
N38M26	Return the X, Y, and Z-axes to the clearance point.
N39M30	End program

Step II: Turned Profile

To view the profile: Launch <u>Mastercam Lathe</u>, from the "MAIN MENU", select "File", "<u>G</u>et", go to the course locker: X:\2.008\Lathe\Mc8\, open KCL.Mc8. Mastercam will try to find the associated tools, just click "OK", or "Cancel" to get through the screens.

To view the toolpath: From the "MAIN MENU", select "<u>T</u>oolpaths", "<u>O</u>perations". In the Operations Manager window, click "Select All", then "Backplot", then select "<u>R</u>un" on the Lathe Backplot menu.

To view the G-code: From the "MAIN MENU", select "<u>F</u>ile", "<u>E</u>dit", "<u>N</u>C", open "KCL.NC, this is also in the course locker: X:\2.008\Lathe\NC\

APPENDIX B: Programming Machine Tools

A part program is composed of a number of machine tool instructions. These instructions can be commands to: turn the spindle on, move on an axis, turn the coolant on, select a tool, or execute an entire pattern of movements. A letter followed by digits is called a 'word' and each program line is termed a 'block'. The commands are executed in order, one 'block' at a time.

Machine tool programming began from a common starting point. The current common character set derives from the first standard NC machining data communication standard – one-inch wide paper tape. In this standard, letters are used to define specific program elements:

Address	Meaning	Address	Meaning
O	program number	F	feed rate
N	sequence number	E	thread lead
G	preparatory function	S	spindle speed
X, Y, Z	coordinate axis motion	T	tool number
R	arc/corner radius, or rapid plane	M	misc./machine functions
I	absolute center of arc in x-axis	J	absolute center of arc in y-axis

Due to the power and frequent use of the preparatory functions, CNC control programs are often called G-code.

Logically, different types of machines require different G-code programs. A lathe has different requirements than a mill or a grinder. In addition, as products have become more advanced and manufacturers have wished to differentiate their products from their competitors, custom commands have proliferated and made certain programs incompatible with different machines. It is therefore important to learn which codes are available on which machines.

COMMONLY USED G-CODE FOR MILLING

G00	rapid linear motion
G01	linear motion at preset feedrate F
G02	circular feed motion - CW
G03	circular feed motion – CCW
G04	dwell for time (P seconds)
G40	cutter compensation off
G41	cutter compensation left
G42	cutter compensation right
G70	inch units
G75	multi-quadrant circles (EZ-Trak machine)
G80	canned drill cycle cancel
G81	standard canned drilling cycle (no pecking)
G83	peck drill cycle
G90	absolute values
G91	incremental values
G92	establish zero point

G55   Workshift 1 (WS1) EZ-Trak machine, Paperweight, and Thermoform mold origin
G56   Workshift 2 (WS2) EZ-Trak machine, Core mold origin
G57   Workshift 3 (WS3) EZ-Trak machine, Cavity mold origin

S   spindle speed (RPM)
F   feed rate (IPM)
T   tool number
R   Height for rapid positioning in drilling cycles, or arc radius value when in G02, or G03

M00   program stop
M01   planned optional stop, can be turned on or off at machine control
M02   end program (Cincinnati Milacron 7vc machine)
M03   spindle on (CW)
M04   spindle on (CCW)
M05   spindle off
M06   change tool
M08   coolant on
M09   coolant off
M13   coolant & spindle on (Cincinnati Milacron 7vc machine)
M25   return Z-axis to machine home (EZ-Trak machine)
M26   return X, Y, and Z axes to a predetermined clearance position for tool change (EZ-Trak)
M30   end program
H   tool height offset

Leading zeros are commonly left out to help reduce the size of the programs.

Computer notes:

The computers are networked to the ME server in bldg. 3. Please note that Mastercam writes to C:\temp on the local hard drives. You must Copy your geometry file ".MC8", and part program file ".TXT" for the EZ-Trak mills, or ".NC" for the lathe, to your assigned lab folder on the server, they are safer there and you will be able to access them from any computer in the lab next time you come in, and others in your work group will be able to find them as well. The computers also have 250MB Zip drives and will recognize 100MB Zip disks. **Make sure you save your work over the network! The computers are set to automatically log you off after 15 minutes of inactivity. This time period may change before the labs begin. This automatic logging off will clean the desktop, and the temp folder.** Please keep all files backed up on a floppy disk, or Zip disk. You may create additional folders within your assigned folder to keep your files organized. Such as a "Mill" folder, and a "Lathe" folder, with sub-folders under each such as MC8, NC. Refrain from saving your work in your personal network folder, others in your group will not be able to access the data. Always work out of your assigned lab folder.

There are 2 computers in the shop, Crossshop1, and Crossshop2.

Send the **Lathe programs to** the appropriate folder within the 'send' folder on **Crossshop1**.
The path: My Network Places\\Entire Network\\Microsoft Windows Network\\Mecheng\\Crossshop1\ Daewoo\Send\

Send the **Mill programs to** the appropriate folder within the 'send' folder on **Crossshop2**.
The path: My Network Places\\Entire Network\\Microsoft Windows Network\\Mecheng\\Crossshop2\ EZ-Trak\Send\