S Y S T E M S
R E S E A R C H
C E N T E R

Control of Machining Accuracy and Numerical Control Machining

by K.R. Kolos
L.Q. Tu
R. Vanegas
K.R. Wardak
Advisor: G.M. Zhang

Control of Machining Accuracy
and Numerical Control Machining

K. R. Kolos
L. Q. Tu
R. Vanegas
K. R. Wardak

Faculty Advisor: G. M. Zhang

May 1991

Abstract

The control of surface quality during machining is an area in manufacturing which has recently received a great deal of attention in the field of research and development. The main objective of this project is to briefly explain and clarify the fundamentals of machining, to teach the basics of numerical control machining, and to give a better physical understanding of the relation between the fundamentals of machining and the actual machining process combined with the use of computers to improve quality and productivity. A basic methodology is proposed in this research for achieving computer-assisted manufacturing processes.

Table of Contents

I. Introduction

The control of surface quality during machining is an area in manufacturing which has recently received a great deal of attention in the field of research and development. The surface quality of a machined part is essential to achieving the desired physical characteristics needed for reliable and durable performance. However, increases in cost and decreases in productivity are two negative factors which result when efforts are made to improve the surface quality of machined parts. In the past, American manufacturing companies have been so intent with saving on costs and increasing productivity that they have sacrificed on quality. As a result, American made products have competitively fallen behind in the world market. Consequently, productivity and quality have become major issues in many U.S. manufacturing industries. The impact of this current state of affairs has triggered a rapid increase in computer applications for production automation to meet the challenges for higher productivity and improved product quality.

An object's surface is the boundary which separates it from another object, substance, or space. The surface boundary of a machined part is formed by the motion of the cutting tool as it removes material from the work piece. Surface texture is the actual surface profile from the nominal surface. Very often, roughness and waviness are observed in the surface texture formed during machining. The deviation in surface peak height distribution determines the surface's functional characteristics. It is very important for the designer to understand the relationship between the texture of a machined surface and its intended or desired function. This requires analytical surface texture measuring systems that define, specify, and control critical surfaces in order to meet important criteria such as lubrication, tolerances, durability, and geometric specifications. One benefit of these systems is the ability to develop a more definitive specification that, if met, will assure that the surface will perform as intended (this improves quality). Another is the ability to analyze and optimize the process (this improves productivity).

The main objective here, is to briefly explain and clarify the fundamentals of machining, to teach the basics of NC and APT programming for computer assisted machining, and to give a better physical understanding of the relation between the fundamentals of machining and the actual machining process combined with the use of computers to improve quality and productivity. The fundamentals of machining are basically used to help the engineer choose the best tools, workpiece material, and machining process necessary to meet the design specifications for a given part. The use of computer simulation through mathematical modeling to predict surface textures or topographies before the actual machining takes place helps the designer optimize the quality and efficiency of the

machining process.   After the optimum process is found, the designer can use NC or APT computer languages to program a computer operated machine to do the actual machining of the workpiece.

II. Fundamentals of machining

Previous work on quality control of a turning machine process has determined that feed, depth of cut and spindle speed are three major cutting parameters related to the machining of a part. The tool kinematics resulting from theses parameters, along with the variable cutting force from tool vibration, effect the surface quality of the part. The machined surface topography, and hence the machine accuracy, can be predicted and controlled through computer simulation that integrates the dynamic and kinematic tool motion during the machining process.

2.1 Selection of cutting parameters
The feed parameter is a measure of the longitudinal speed of the tool, determined by the distance the tool moves along the workpiece in one revolution. The depth of cut is a measure of how deep the tool cuts into the workpiece. The spindle speed is the revolutions per minute of the part being machined. The following figure shows the three cutting parameters in a turning machine process.
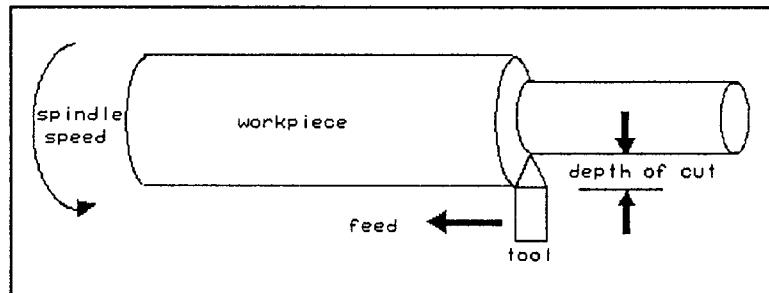
Figure 1. Cutting parameters

The surface quality of the workpiece is dependent upon the proper selection of cutting parameters. The feed parameter effects the surface quality most. A large feed results in a rough surface, whereas as small feed has the effect of polishing. The depth of the cut relates to the chip load on the tool. A smaller chip load results in a weaker forcing function for the tool vibration which gives a smoother surface.

2.2 Modeling the cutting force

To determine the forcing function of the tool vibration for a macroscale analysis, the tool is modeled by the spring-mass-damper system shown in Figure 2. Using this model, the dynamic tool vibration can be determined from the system response.
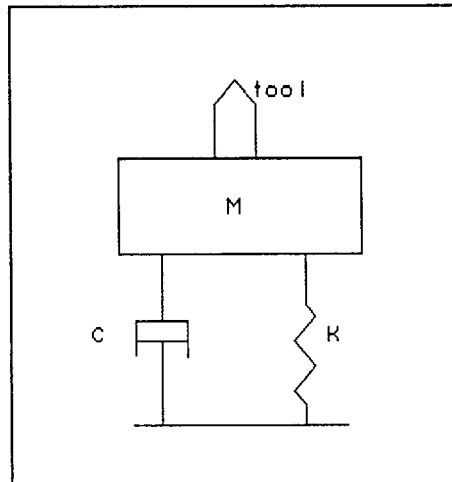

Figure 2. Tool model.

However, for small tolerances the random motion of the tool must be modeled using a microscale analysis, where the cutting force variation is related to the microstructure of the workpiece material. A previous analysis of the cutting dynamics in microscale has been performed and is briefly described here. In this analysis, two assumptions are made. The first assumption is that the main cause of cutting force variation is the non-homogeneous hardness distribution of the workpiece material. The second assumption is that only the magnitude of the cutting force is varied, not the direction of the force.

The cutting force is generated immediately after the cutting tool digs into the workpiece. This nominal cutting force $F_{nominal}$ is assumed to be proportional to the chip load and it is directly related to the material property, tool geometry, and the three cutting parameters. The interaction between the cutting mechanism and machine tool structure result in dynamic variation of the cutting force. These variations are described by $F_{primary}$ and $F_{regenerative}$, which are a consequence of changes in the nominal chip load by the tool vibration and overlapping of the cutting path.

To mathematically describe the random variation of the cutting force due to the microstructure of the material, a statistical approach is used to identify the characteristics of the hardness variation. The random cutting force variation, F_{random} is used to simulate the up and down motion of the tool or changes due to the variation in the material hardness. Therefore, the resultant cutting force consists of four parts, $F_{nominal}= F_{random}+F_{primary}+F_{regenerative}$.

2.3 Computer Simulation

As tolerances decrease, the surface quality of a part must be considered in the machine accuracy. Through simulation of the workpiece surface the surface quality can predicted and controlled. A previous study on quality control used the computer simulation program SIMTOP. This program integrated the tool vibration, the response from the cutting parameters, and the random excitation of the workpiece to simulate the tool motion and surface texture of the workpiece.

The program consists of five modules: user input, tool vibration, tool kinematics, surface texture generation, and surface characterization. In the input module, the tool geometry and machining parameters of feed, depth of cut, and spindle speed are entered. The tool vibration module generates the tool vibratory motion from a mathematical model of the force (in this particular study, the force was modeled by a macroscale analysis). Using the three cutting parameters, the tool kinematics module simulates the ideal surface of the workpiece. Next, the surface texture generation module combines the tool vibratory motions, the kinematic motions, and the material properties of the workpiece to simulate the actual surface texture of the finished part. The final surface characterization module computes the characterization indices of the surface topography and stores the data in a matrix. The columns of the matrix correspond with the incremental cutting location, while the rows correspond with the incremental feed location. For a visual study, the data can be entered into a graphics program to generate a three dimensional representation of the workpiece surface.

The surface texture can be quantified by R_a, an international parameter of roughness; a small value for R_a indicates a smooth surface. The surface characterization module in the computer simulation can be used to find the R_a value. Thus, the R_a value allows for a quantitative study of the surface quality.

Through computer simulation, the surface of the workpiece can be determined before the workpiece is made. The cutting parameters that result in the desired surface quality can be determined by running the computer simulation program with various cutting parameters. By choosing the correct parameters, the surface quality can be controlled, which is essential for controlling the machine accuracy when small tolerances are required.

III. NC Machining

3.1 NC Machining Background

The controlling of a machine tool by the use of a prepared program is known as numerical control, or NC. NC equipment has been termed by the Electronic Industries Association as "A system in which actions are controlled by the direct insertion of numerical data at some point. The system must automatically interpret at least some portions of this data."

In most NC systems, the numerical commands, referred to as G Codes, which are necessary for producing a part are referred to as the "part program" (Koren, p. 3). The part program is arranged in blocks of information where each block contains the codes to produce a certain portion or segment of the workpiece. The block contains all the necessary information required to process a segment of the workpiece: i.e., the segment length, cutting speed, feedrate, etc. All dimensional information is taken from engineering drawings, and dimensions are given separately for each axis of motion (X, Y, etc.). Feedrate, cutting speed, and auxiliary functions (coolant on and off, spindle direction, clamp, gear changes, etc.) are programmed in accordance with the surface finish and tolerance requirements. The part program is maintained on a punched tape for NC machining and the punched tape is moved forward by one block each time the cutting of a segment is completed. CNC machining is an advanced version of NC machining with the only difference being that the programming data can be keyed into a personal computer storage memory through the CNC monitor rather than stored on a tape. In this way, any error that appears on the program can be corrected instantaneously instead of retyping a new tape, thus the CNC system is more efficient than the NC system.

When compared to a conventional machine tool, the NC system replaces the manual actions of the operator. In conventional machining, handwheels are used to move the cutting tool along the edge of the workpiece, which are guided by the operator. Contour cuttings are done by an expert operator by sight. In NC machining, operators of NC machine tools are not required to be expert machinists. The NC operators need only monitor the operation of the machine, operate the tape reader, and usually replace the workpiece. All thinking operations previously performed by expert machinists are now contained in the part program.

The preparation of the part program for a NC machine requires a part programmer. It is necessary that the part programmer have knowledge and experience in mechanical engineering fields. The knowledge of tools, cutting fluids, fixture design techniques, use of machinability data, and process engineering are all of important

factors. Part programmers must have familiarity with the function of NC machine tools and machining processes and have to decide on the optimal sequence of operations. They write the part program manually through G Code or by the use of a computer language, such as APT, to generate the G Code.

3.2 CNC Programming

In this section, some CNC command features and program procedures are explained through example programs in providing sufficient knowledge for all beginning operators to perform machining.

1. The positioning system.
_The *fixed zero point* is permanently located at the left hand front corner of the worktable. At this origin, the x and y axes are running along the front and side edges of the worktable respectively. This is a one quadrant operating area and it can be used as the absolute zero point for the workpiece.
_The *offset zero point* is a zero point which is shifted away from the fixed zero point. This permits a convenient zero location to be established on the work piece as long as all machining points fall within the operating range of the machine tool. This offset zero point can be manually made and then it is registered into the storage memory.
_The *full floating zero point* is a location which enables a machine operator to perform all machining process in all four quadrants rather than one.

2. The x, y, z coordinates.
_In fixed zero or offset zero location, +x and +y are pointed to the right and to the rear of the CNC machine respectively.
_The z axis is used to control the vertical motion of the machine tool. Normally, the surface of the workpiece is designated as zero with positive z upward away from workpiece.

3. Designated coordination command for machining movement.
_The absolute command designation (G90) relates each machining point to the designated zero point.
_The incremental command designation (G91) relates each machining point to the location of the preceding machining point. For example, a "present" machining point to the right or the rear preceding machining point is described as +x and +y increment from that point respectively, while a location to the left or the front is described as -x or -y increment respectively. When summed, all the x and y increments should algebraically total to zero.

4. The x, y, z axes command range.
The command range for all three axes is identical and it is programmable in both inch and millimeter.
_Inch range from   :   000.0001 to 999.9999
_millimeter range from: 0000.001 to 9999.999

5. Important points in writing CNC programs for the system in the Advanced Design and Manufacturing Laboratory of the mechanical engineering department at the University of Maryland.

_A CNC program must be written in <u>WordStar</u> programming language.

_A CNC program must stored in 5 1/4" floppy disk.

_A 4 digit program number must be registered in the beginning of the program starting with letter "O".

_All letter characters must be capitalized.

_Leading zero, trailing zero and positive sign may be omitted from a command or a number. For example: G09=G9; X12.0=X12;

_A z movement command should be placed on a separate block if it is performed a cutting function.

_In beginning of each CNC program, it is recommended to turn on the coolant and the spindle speed so these two machine operations can reach to the steady state prior cutting operations.

_Some CNC command addresses are explained as follows:

N Sequence number. it is used to identify a block of program instruction.
G Preparatory function.
X X axis command.
Y Y axis command.
Z Z axis command.
I X axis offset from arc center.
J Y axis offset from arc center.
K Z axis offset from arc center.
F Feedrate (in/sec).
T Tool select.
M Miscellaneous function.

3.3 CNC program examples.

Four examples of CNC programs are used to demonstrate how machining process are operated on four different workpieces (see figures 3-6).

In each example, interpretations are located at the right side of block statements to illustrate the G codes and other CNC commands (these interpretations are not used in actual programs).

All dimensions are measured in inches.

In each example, each N number path in the figure is corresponded to that N number block statement.

The cutting paths travel in clockwise direction in all examples. Left compensations are used to keep the cutting area in the left side of the traveling paths so that dimensions of the figure are not distorted.

The followings are abbreviations for CNC program interpretation statements:

abs = absolute.
CW & CCW = clockwise & counter clockwise rotation.
comm = command.
coor = coordinate.

```
dia  = diameter.
dir  = direction.
dis  = distance.
pt   = point.
R    = radius.
ref  = reference.
s    = spindle speed (rev/min).
```

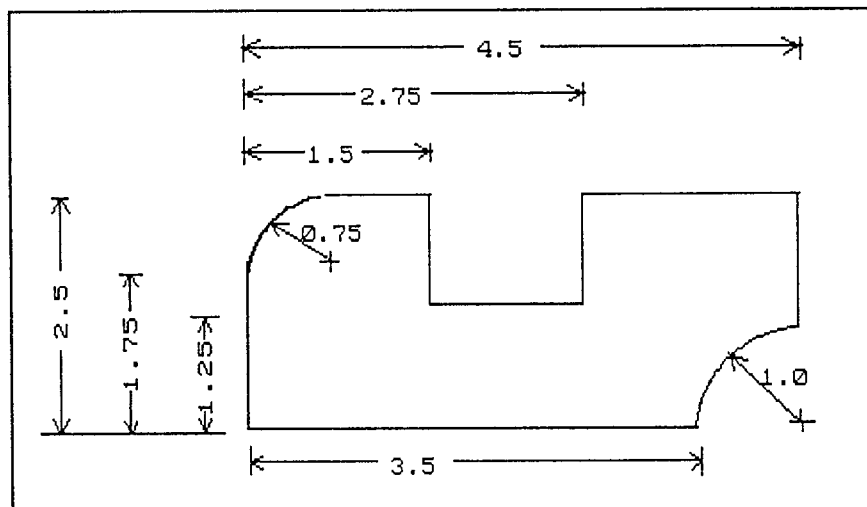Example 1
CNC Program for Figure 3 with Left Compensation

Figure 3. CNC example 1.

```
Notes:
Material property of workpiece:
     transparent plastic block,  4.25"(y)x7"(x)x.25"
T1= 1/4 in diameter face mill.
Starting cutting point= (1,0)
offset zero origin.
Depth of cut: .05"
H1=H2=H3=H4=  .1" above workpiece surface (These  H  commands  are
     stored in memory).
Using absolute command designation (G90) for cutting.
```

```
%                      Starting program.
O6818                  Program number: 6818
N5T1M06S3500           Change to tool 1 (T1M06); S=3500.
N10G91G43Z0H1M3         Using  tool  length  compensation  (G43),  move
                            down (z0) to H1 position; Spin the tool in
                            CW motion (M3).
N15G92Z.1M08           (G92) Set current z position into z=.1 above
                            workpiece; coolant off (M08).
N20G90G1X0Y0G41D25F25   At  (G90)  abs  comm,  move  tool  linearly
```

	(G1) to left radius compensation position (G41) of pt (0,0) with tool dia= .25"; F=25.
N25Z-.05F2	Move down cutting .05" into the workpiece surface; F=2.
N30Y1.75F25	Still at G90 & G1, move to Y=1.75; F=25.
N35G02X.75Y2.5R.75	Move in a CW path (G02) to end pt (.75, 2.5) at R= .75
N40G01X1.5	Change G02 to G1, move linearly to x=1.5
N45Y1.25	Move to y=1.25.
N50X2.75	
N55Y2.5	
N60X4.5	
N65Y1	
N70G03X3.5Y0R1	Move in CCW path (G03); end pt (3.5, 0); R=1.
N75G1X0	
N76Y0.2	Complete the cutting shape in passing the starting pt (y=0) until y=.2
N80G00Z.1M9	Traverse movement back to z=.1; coolant off.
N83G40	Left compensation cancel (G40).
N85G91G28Y0Z0M19	Move incrementally (G91) back to machine tool ref pt (G28); spindle speed stop (M9).
N90M30	End of tape (M30).
%	End of program.

Comment:

1. In N35, an alternate CW path of G2 can be established in the following form of N35G02X.75Y2.5I.75J0 (from the preceding pt of (0, 1.75), arc center is made with x distance (I)=.75 and y dis (J)=0).
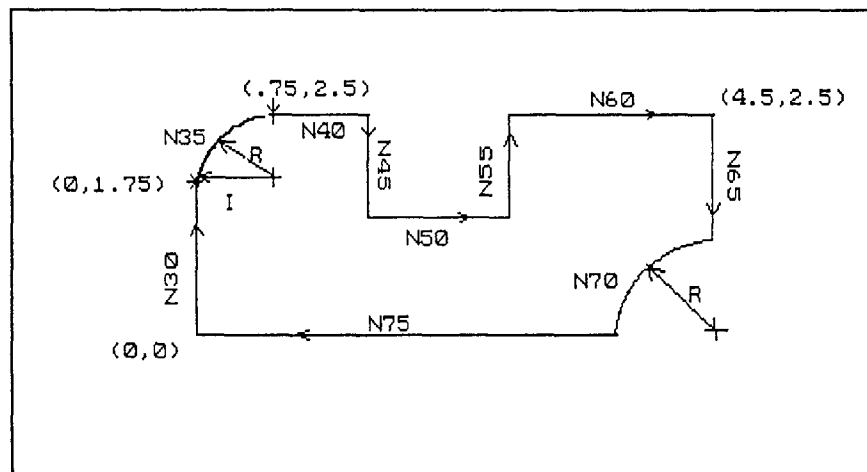

Figure 3A. CNC example with program numbers.

Example 2
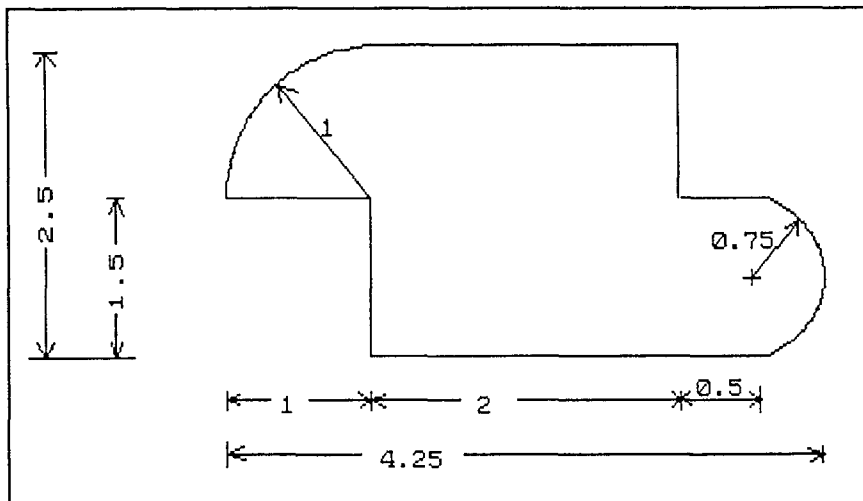CNC Program for Figure 4 with Left Compensation

Figure 4. CNC example 2.

Notes:
Identical notes as example 1.
Starting cutting point= (1, 0).
Using incremental command designation (G91) for cutting.

%	Starting program.
O0124	Program number: 0124
N2T1M06S3500	Change (M06) to tool 1 (T1); set s=3500.
N4G00G90X1.0Y0	Rapid traverse movement (G00) in abs coor to starting position (1,0).
N6G91G43Z0H1M3	Increment z movement (G91) down to memory storage H1 position with tool compensation (G43); tool rotates in CW motion (M3).
N8M8	Turn coolant on (M8).
N10G90G41X1Y0	Abs comm movement (G90) at (1,0) with left compensation (G41).
N12G91Z-.15F2.0	Move down incrementally & linearly (G1) at -.15 of z axis; F=2; (The tool now is cutting the surface .05" deep).
N14Y1.5F25	Move y= 1.5" incrementally at F=25.
N30G01X-1.0	
N35G02X1Y1R1	CW circular cutting path (G02); end pt (1,1); R=1.
N41G01X2	Change G02 to G01 command.
N45Y-1	
N50X.5	
N55G02X0Y-1.5R.75	Same movement as N35; end pt (0,-1); R=.75
N60G01X-2.5	

N62Y.2	Complete the cutting path by passing (1,0).
N65G00G90Z.15M9	Rapid move (G00) Z up .15 (Z coor=.1); coolant off (M9).
N68G40	Cancel left compensation (G40).
N70G91G28Y0Z0M19	Move incrementally & automatically back to machine tool ref position (G28); Spindle speed stop (M19).
N75M30	End of tape (M30).
%	End of program.
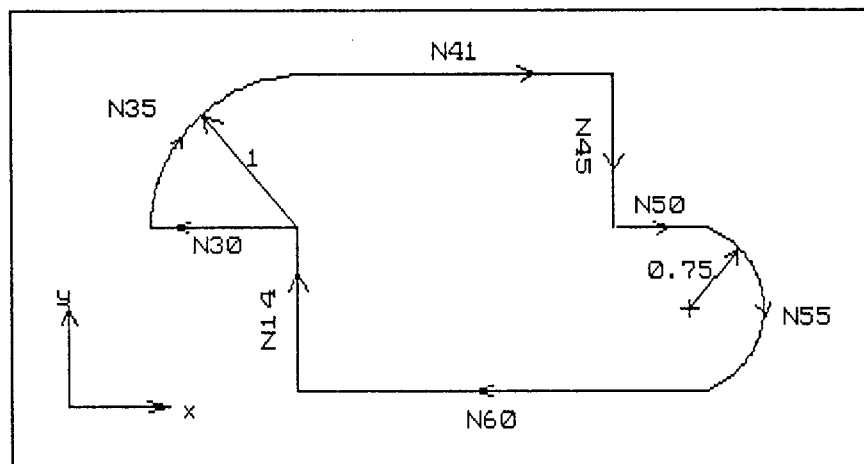
Figure 4A. CNC example 2 with program numbers

Example 3
CNC Program for Fig.5 with Left Compensation

Notes:
Identical material property as example 1.
T1= .25 (1/4) in diameter face mill.
T2= .125 (1/8) in diameter face mill.
Depth of cut= .1 in
Offset zero origin.
Starting cutting point= (0,0.5)
Using absolute command designation (G90) for cutting.
H1=H2=H3=H4= .1" above workpiece surface (These H commands are stored in memory).

%	Starting program.
O6819	Program number: 6819
N5T1M06S3500	Change to tool 1 (T1M06); S=3500.
N10G00G90X0Y.5	Rapid traverse move (G00) in abs coor (G90) to starting pt (0,.5).
N15G91G43Z0H1M03	Using tool length compensation (G43); Move down incrementally (z0) into stored memory H1; spin tool in CW rotation

```
                                (M03).
N20G92Z.1M08              (G92) Set current z position into z=.1 above
                                workpiece; coolant on (M08).
N25G90G01G41D25F25        (G90)    abs    comm;    move    tool    to    left
                                compensation (G41) position with radius
                                .25"; F=25.
N30Z-.1F2                 Cutting down .1" into the workpiece; F=2.
N35Y3.3                   Move from (0, 0.5) to (0, 3.3).
N40G03X.5Y3.8R.5          Move in a CCW path (G03) to end pt (.5, 3.8)
                                at R= .5
N45G01X1.3                Change G03 to G1, move linearly to x=1.3
N50G3X2.3Y3.8R.5          Move in a ccw path (G3) to end pt (2.3, 3.8)
                                at R= .5
N55G1X3.2                 Linearly (G01) move to x= 3.2.
N60G03X4.2Y3.8R.5         CCW path to end pt (4.2, 3.8).
N65G01X5
N70G3X5.5Y3.3R0.5
N75G1Y0.5
N80G03X5Y0R.5
N85G01X4.2
N90G03X3.2Y0R.5
N95G01X2.3
N100G03X1.3Y0R.5
N105G01X0.5
N110G03X0Y0.5R0.5
N113G01Y0.6
N115G00Z.1
N120G40                   Tool radius compensation cancel.
N125G00X.8Y1.9Z.03        Traverse movement(G00) to (.8, 1.9, .03)
N130G01Z-.1F2
N135G12I0.23Q0.23K.4F25D30        (G12) circle cutting through spiral
                                cw path at starting pt at I=.23 (x
                                dis) from center pt (.8, 1.9, -.1); spiral
                                width increment at .23 (Q.23) to radius of
                                finished circle (k.4).
N140G00Z.1M9              Using G00, move to z=0.1; (M9) coolant off.
N145G91G28Y0Z0M19        Incrementally (G91) return to tool ref pt
                                (G28) through intermediate pt (x not change,
                                0, 0); spindle speed stop (M19).
N150T2M6S1500            Change to tool 2 (T2M6); s=1500.
N155G00G90X1.9Y1.9       Using (G00) in abs coor (G90), move to (1.9,
                                1.9).
N158G91G43Z0H2M03        Similar to N15 (H2 position).
N159G92Z.1M08            Similar to N20.
N160G90G98G81Z-.1R.03F10        Establish drilling operation (G81)
                                for next N blocks.  After each
                                drilling to z=-.1, move tool in z dir in
                                passing pt R (z= 0.03) to initial position
                                (z=.1 in N159), then move to next drilling
                                (this is canned cycle function (G98)).
N165G70X2.3Y1.9I.4J180L8        (G70) bolt hole circle path; starting
                                pt (2.3, 1.9) at angle (J) 180°; radius
```

```
                                  (I)= .4; 8 repetitions (L8).
N170G72X3.3Y2.4I.25J0L7        At  G98  &  G81,  line  path  drilling
                               (G72)  at  starting  pt  (3.3,  2.4)  with  0°  angle
                               (J0);      interval      increment      (I)=.25;      7
                               repetitions(L7).
N175G72X4.8Y1.9I.25J180L7      See N170; starting pt (4.8,1.9)
N180G72X3.3Y1.4I.25J0L7        See N170; starting pt (3.3,1.4)
N185G80M09                     drilling operation cancel (G80).
N190G91G28Y0Z0M19              Similar to N145.
N195M30                        End of tape (M30).
%
```
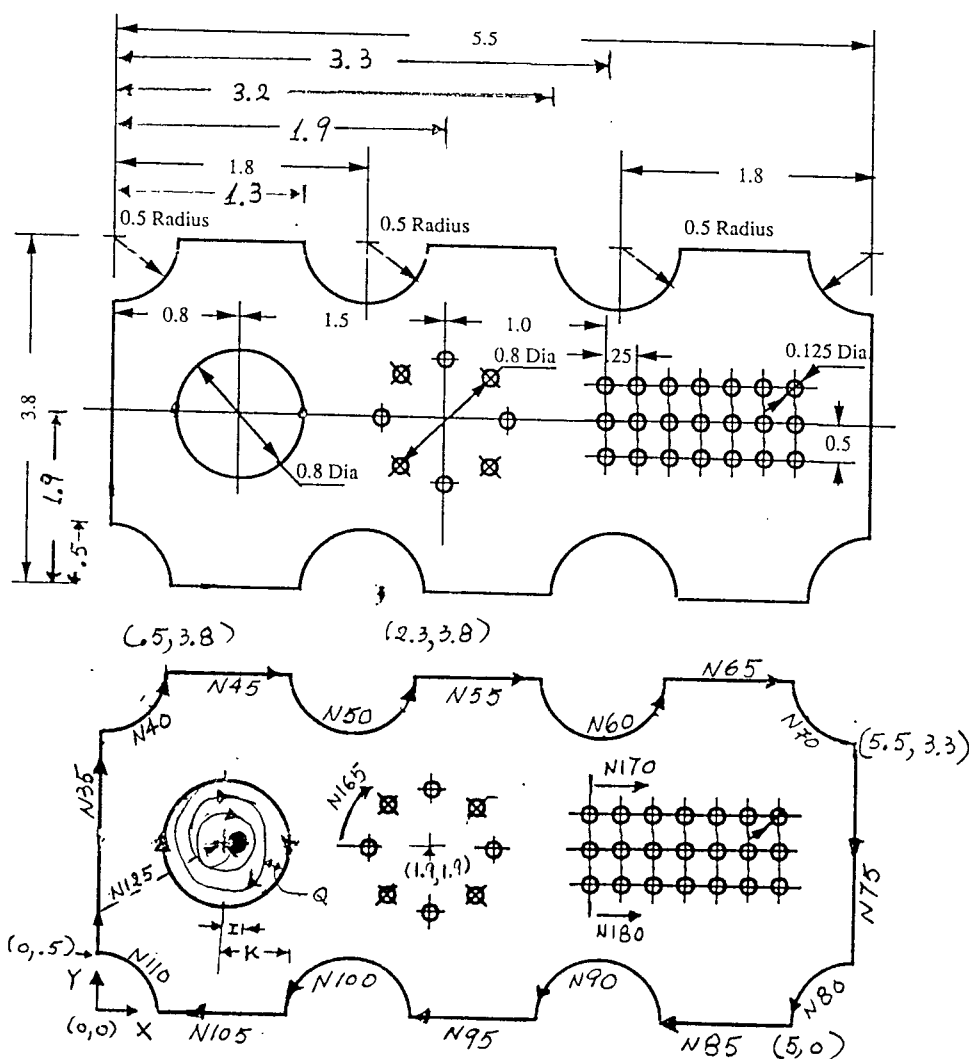

Figure 5. CNC example 3.

<u>Example 4</u>
<u>CNC Program for Fig.6 with Left Compensation</u>
Notes:
Material property of workpiece: aluminum block, 6(y)"x7(x)"x1".
T1= 3.0 in diameter face mill.
T2= .75 in diameter face mill.
T3= # 3 center drill
T4= .1875 (3/16) in diameter end mill; .15 minimum drill length.
Machining procedures:
    1. Milling the top surface by T1 (depth of cut= .02")
    2. Milling the outline shape by T2 (depth of cut = .02")
    3. Drilling holes along the left surface edge and continue
        to make a circle pattern with T3 (depth of cut= .02).
    4. Milling a circle, a pocket, an arc at various depth of
        cut.
Full-floating-zero origin.
Starting cutting point= (2.7, -4.5)
Using absolute command designation (G90) for cutting.
H1=H2=H3=H4= .1" above workpiece surface (These H commands are
    stored in memory).

%	Starting program.
O4816	Program number: 4816
N1T1M06S4000	Change to tool 1 (T1M06); S=4000.
N2G00G90X2.7Y-4.5	Rapid traverse move (G00) in abs coor (G90); starting pt (2.7, -4.5).
N3G91G43Z0H1M03	Using tool length compensation (G43); Move down incrementally (z0) into stored memory H1; spin tool in CW rotation (M03).
N4G92Z.1	(G92) Set current z position into z=.1.
N5G90Z-.02	At (G90) abs comm; move tool to z= -.02.
N6G1X-8.4F50	Brush the workpiece surface at F=50.
N7G00Y-1	At G00, move to y=-1.
N8G1X2.7	Change G00 to G1, move to x= 2.7.
N9G00Y-2.5	
N10G1X-8.4	
N11G00Z.1	
N12G91G28Z0M19	Incrementally move (G91) to tool ref position (G28); stop spindle speed (M19).
N13T2M6S3000	Change to T2 (see N1).
N14G00G90X0Y.6	Move in abs coor (G90) to (0, .6).
N15G91G43Z0H2M3	Similar to N3.
N16G92Z.1M8	See N4; coolant on (M8).
N17G90Z-0.05	At G90, move down to z=-.05.
N18G1G41X-.5D75F25	Left tool radius compensation (G41) at (-.5, .6); diameter= .75"; F=25.
N19G1X0Y-2	
N20Y-5	
N21X-5	
N22G2X-5.6402Y-3.3218R1	CCW circular path (G2), end pt (-5.6, -3.2); R=1.
N23G1X-3.1401Y-.2318	

```
N24G2X-2.5Y0R1          See N22.
N25G1X.5
N26G00G40Y.6Z.1M9       Cancel left compensation (G40); move to
                             (.5, .6, .1); coolant off (M9).
N28G91G28Y0Z0M19        See N12; passing intermediate pt (.5, 0, 0).
N29T3M6S1500            Change to T3.
N30G00G90X-5Y-4.7       At G00 & G90, move to (-5, -4.7).
N31G91G43Z0H3M3         See N3.
N32G92Z.1M8             See N4, M8 coolant on (M8).
N33G90G99G81Z0R0.03F10     At G90 & drilling operation (G81),
                             set  R(z  dir=.03).   After  each
                             drilling at z= -.02 in next N blocks, move to
                             to R and then to the next drilling (this is
                             canned cycle function (G99)).
N34G71X-5Y-4Z-.02I.7J-120K-10L11  Arc  path  drilling  (G71)  at
                             center  (-5,  -4);  depth  z=  -.02;  radius
          (I)=.7; J= -120 is angular position of the 1st hole).
N35G72X-5.5362Y-3.55I.15J50.2L20   Line path of drilling (G72) at
                             angle of 50.2° with starting pt (-5.5, -3.5);
                             hole interval (I)= .15; 20 holes (L20).
N36G70X-3.212Y-1.3604I.5J180L20     Bolt  hole  circle  path  (G70);
                             center  pt  (-3.2,  -1.3);  radius  (I)=.5;
                             angular starting pt at 180°; 20 holes (L20).
N37G80M9                G80 cancel G81; coolant off (M9).
N38G91G28Z0M19          See N12.
N39T4M6S5000            Change to tool 4.
N40G00G90X-1.5Y-4.3 Move to (-1.5, -4.3).
N41G91G43Z0H4M3         Set tool compensation (G43).
N42G92Z.1M8
N43G90Z.03
N44G1Z-.02F2
N45G3I.5J0F15           CCW path (G3); from center pt at N40 , create
                        starting pt by x dis (I)= .5 & angle (J)= 0°.
N46G00Z.1
N47X-3.5Y-3.8
N48G00Z.03
N49G1Z-.025F2
N50G13I.18Q.18K1F15        Circle cutting through ccw spiral path
                        (G13);  starting  pt  at  x  dis  (I)=  .18  from
                        center  pt  of  N47;  Q=  .18  (spiral  path
               interval increment; k= radius of final circle.
N51G00Z.1
N52X-.6Y-2.5
N53Z.03
N54G1Z0F2
N55G3X-1.6Y-1.5Z-.15R1  CCW  path  (G3)  move  to  (-1.6,  -1.5)  at
                             radius=1; z=-1.
N56G3X-2.1Y-2Z0R.5      Continue cutting to the end pt
                             (-2.1, -2) at z=0.
N57G00Z.1M9             Stop coolant (M9).
N58G91G28Y0Z0M19        Return  to  tool  ref  pt  (G28)  through  pt  (x
                        unchange, y=0, z=0); Spindle speed stop(M19).
```
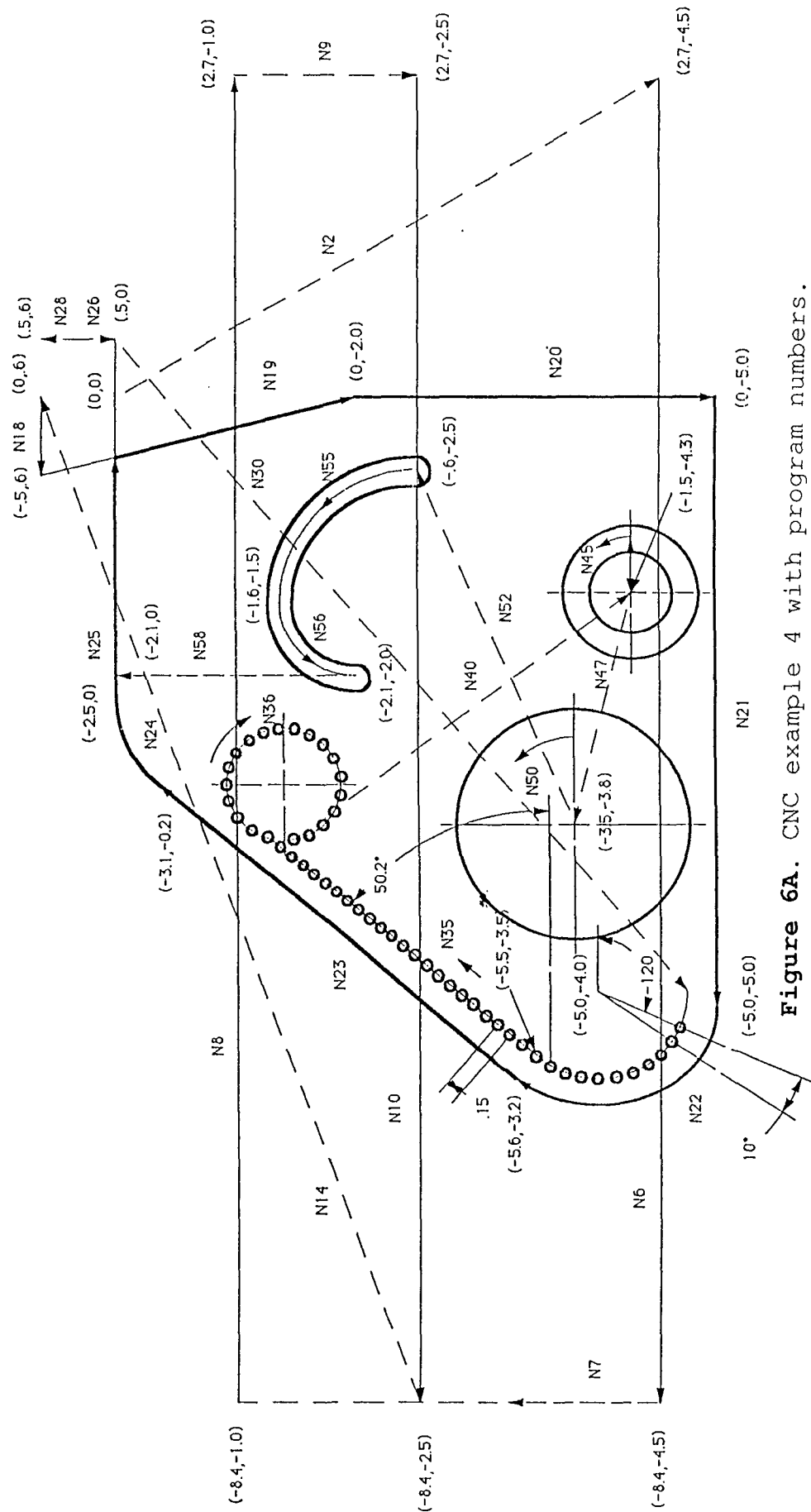
N59M30   End of tape.

%

Figure 6. CNC example 4.

Figure 6A. CNC example 4 with program numbers.

19

3.4 APT Programming

APT is the abbreviation for Automatically Programmed Tool. It was initially developed at M.I.T. in 1956, and it is the most popular programming language in the United States. The Illinois Institute of Technology Research Institute (IITRI) has continued the development and administrative responsibility for the fourth version of APT which was developed in 1970. APT is the most powerful universal-purpose part-programming system, against which other systems are usually compared and evaluated. The following statements summarize characteristics of the APT system according to Niebel, Draper, and Wisk:

1. Three-dimensional unbounded surfaces and points are defined to represent the part to be made.
2. Surfaces are defined in an X-Y-Z coordinate system chosen by the part programmer.
3. In programming, the tool does all the moving; the part is stationary.
4. The tool path is controlled by pairs of three-dimensional surfaces; other motions, not controlled by surfaces, are also responsible.
5. A series of short straight-line motions are calculated to represent curved tool paths (linear interpolation).
6. The tool path is calculated so as to be within specified tolerances of the controlling surfaces.
7. The X,Y,Z coordinates of successive tool-end positions along the desired tool path are recorded as the general solution to the programming problem.
8. Additional processing (post-processing) of the tool end-point coordinates generates the exact tape codes and format for a particular machine.

There are four types of statements in the APT language:
1. Geometry statements. These define a scaler or geometric quantity.
2. Motion statements. These describe a cutter path.
3. Postprocessor statements. These define machining parameters like feed, speed, coolant on/off, etc.
4. Auxiliary statements. These describe auxiliary machine tool functions to identify tool, part, tolerances, etc.

There are two additional important features provided in the APT program:
1. Macros. Individual macros similar to FORTRAN subroutines are created for adding to the APT program routines. A library of frequently used routines and definitions can be created as special macros.
2. Loops. An individual section of an APT program can repeat itself until a specified result is obtained.

3.5 APT examples

The following example is a typical APT program which is used to machine fig. 7:

A vertical NC milling machine is in the home position in fig. 7. The machine is commanded to scribe 5 olympic crest type circles of 1 inch diameters. The circles are to be machined to a depth of .5 inches.

program:

```
PARTINO, OLYMPIC CREST          ;defines part name
MACHIN/P & W HORIZON V          ;defines machine type
SETPT=POINT/0,.5,.5             ;defines the home tool  position
C1=CIRCLE/1.5,3.0,.0.5          ;defines location of circle 1
C2=CIRCLE/2.75,3.0,0.5          ;defines location of circle 2
C3=CIRCLE/4.0,3.0,0.5           ;defines location of circle 3
C4=CIRCLE/2.125,2.25,0.5        ;defines location of circle 4
C5=CIRCLE/3.75,2.25,0.5         ;defines location of circle 5
L1=LINE/LEFT,TANTO,C1.
LEFT,TANTO,C2                   ;defines location of line 1
L2=LINE/RIGHT,TANTO,C4
RIGHT,TANTO,C5                  ;defines location of line 2
$$ PART DESCRIPTION HAS NOW
BEEN COMPLETED
CUTTER/0.250                    ;cutter diameter
INTOL/.001                      ;inner tolerance
OUTTOL/.001                     ;outer tolerance
SPINDL/1740,CLW                 ;spindle RPM
FEDRAT/2500                     ;feedrate
FROM/SETPT                       ;defines the initial location of
outer
GO/TO,L1                        ;moves tool to line 1
GORGT/L1,TANTO,C1                ;go right on line 1, tangent to
circle
GODLTA/0,0,-0.75                 ;go down
GOFWD/C1,TANTO,L1              ;go forward on circle 1, tangent to
line 1
GODLTA/0,0,+0.75                 ;go up
GORGT/L1,TANTO,C2                ;go right on line 1, tangent to
circle 2
GODLTA/0/0,-0.75                 ;go down
GOFWD/C3,TANTO,L1              ;go forward on circle 2, tangent to
line 1
GODLTA,0,0,0.75
GORGT/L1,TANTO,C3
CODLTA/0,0,-0.75
GOFWD/C3,TANTO,L1
GODLTA,0,0,+0.75
GOTO/L2
GOLFT/L2,TANTO,C4
```

```
GODLTA/0,0,0.75
GOFWD/C4,TANTO,L2
GODLTA/0,0,+0.75
GORGT/L2,TANTO,C5
GODLTA/0,0,-0.75
GOFWD/C5,TANTO,L2
GODLTA/0,0,+0.75
RAPID                           ;change feedrate to rapid
GOTO/SETPT                      ;go back to home position of tool
COOLANT/OFF                     ;turn coolant off
FINI                            ;termination of part program
```
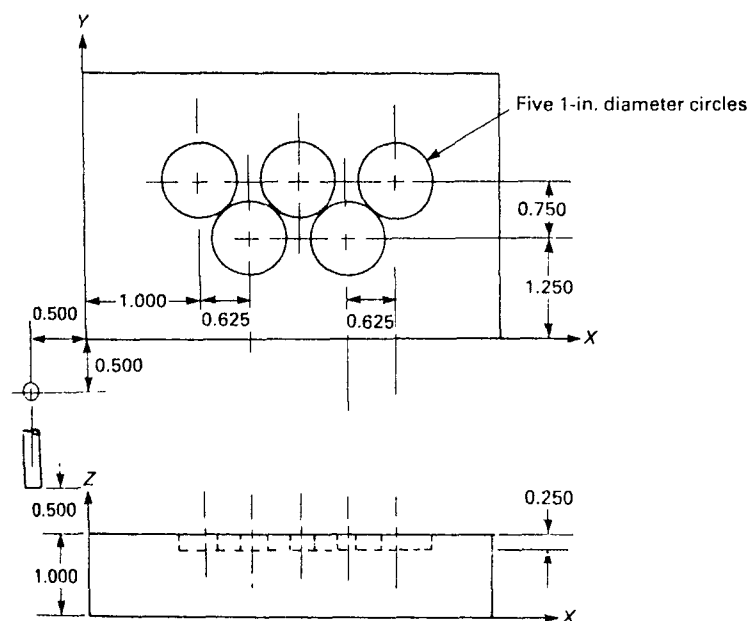
Figure 7. APT example for program.

CONCLUSION

In order to compete successfully in a world market, U.S. manufacturing companies are integrating multiple parameter surface texture evaluation into their design, manufacturing, and quality control operations. Quality is improved because the computer helps the designer choose the optimum cutting parameters with greater accuracy. The mathematical modeling of the random cutting force vibrations which depend on the hardness variation of the workpiece

material also helps the designer to choose the material which yields the best results for a high quality finish. Efficiency is increased because computer surface simulation eliminates the traditional trial and error methods used to find the optimum process. One of the great advantages of CNC machines is that they help to serve as a direct link between the designer and the actual machining process; this helps the engineer to come up with designs that are practical for commercial mass production. According to manufacturing experts, the challenge for U.S. manufacturing companies is to invest in analytic surface texture measuring equipment, to perform the empirical testing that will clarify the relationship between surface texture and function. Furthermore, to develop more meaningful specifications by involving design engineers in the process, and to use this new knowledge and equipment to improve performance and reduce cost.

4.1 Why undergraduates should take a course on NC machining

There are nationwide opportunities for engineers with NC programming experience. Recruiting specialists in metalworking have a constant need for degreed engineers experienced in process, design, automation, and tooling engineering. Companies like ITM which manufacture precision machine parts for aerospace, defense, aircraft, and automotive industries are always looking for manufacturing engineers; however, they usually require that the engineer have experience as a machine parts planner, or NC/CNC programmer. Many universities have recognized the current demand for manufacturing engineers, and are responding by hiring professors who specialize in areas such as computer control of machines and processes, and modeling and simulation for computer integrated manufacturing systems. Companies like Mastercam, one of industries leaders in CAM (Computer Aided Manufacturing) technology are constantly developing new CNC programming systems that offer more NC programming power with new features such as automatic surface roughing, three-dimensional swept surface and enhanced pocketing routines for better efficiency. This proves that NC programming is a field with a bright future since it is constantly growing and expanding; it will not be replaced because the new technology currently being developed is designed to work using NC programming. For example, Sandia National Laboratories has recently introduced a software system that simplifies NC programming. Other new technologies, such as "beam" cutting processes, can be easily integrated into CNC machines and need only a programming change and process adjustments to accommodate new jobs.

4.2 Recommendations

It would be helpful if students could initially see the different types of products that are manufactured by machines which are operated by NC programming. Furthermore, a few examples of actual cases where surface quality is very important, such as for

lubrication surfaces, would help clarify the importance and significance of this research. Learning to do NC programming is very helpful in terms of giving one a better feel for what computer assisted machining is about because the student gets to actually see how to operate the machine. However, it would be even better if the students could write a program, and then machine the same configuration three of four different times while varying the cutting parameters in order to see what physically happens to the surface quality for each trial. The two reports on controlling surface were very helpful, but the student needs to either write or learn how to use a previously written for finding the optimum cutting process and parameters as a project in order to get a better feel for what is explained in the two reports (Control of the Surface Quality During Machining, and Analysis of the Cutting Dynamics in Microscale). A good project for the above would be to assign students to machine an actual part according to a give set of specifications. They could first use or write a program to find the optimum cutting parameters, and then write the NC program using these optimum parameters. Finally, a brief look at the traditional procedures used for machining parts while comparing and contrasting them with the improvements achieved by CNC machining together with computer surface quality simulation which show the significance of this new technology.

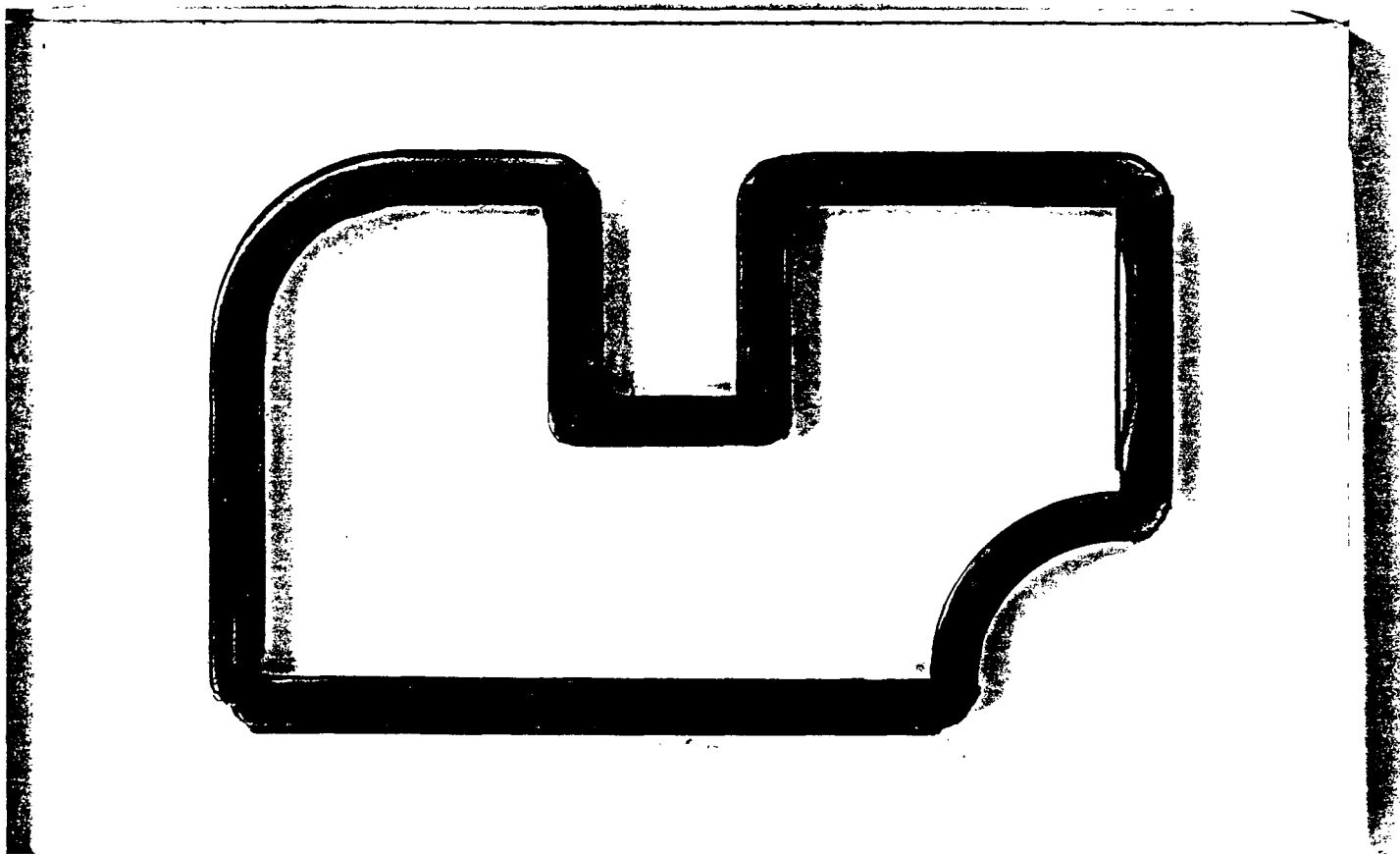## Acknowledgements

References:

1. Control of the Surface Quality During Machining, Engineering Experimentation, 1990.
2. Draper, Niebel, and Wisk, Modern Manufacturing Process Engineering, New York: McGraw Hill Publishing Company, 1989, pp.870-873.
3. Koren, Computer Control of Manufacturing Systems, New York: McGraw-Hill Book Company, 1983, p.3.
4. Lavoie, R. A., "Understanding the Limits of R_a," Manufacturing Engineering. vol 106, no. 1, p.6.
5. Zhang, G.M., and Hwang, T.W., Analysis of the Cutting Dynamics and Microscale, University of Maryland, Systems Research Center
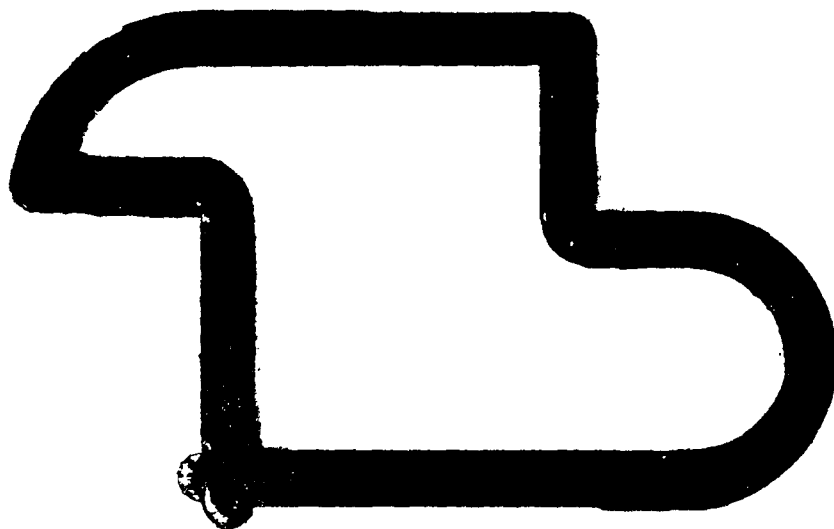
V. APPENDIX

EXPERIMENTAL VERIFICATION

The first three CNC programs in this paper were tested on plexiglass. The following photographs are the results of the tests. The machined cuts were outlined in black to increase their visibility.

Example 1.

Example 2.

Example 3.