CNC Machining

Intro to CNC Machining

- CNC stands for ***computer numeric controlled***. It refers to any machine tool (i.e. mill, lathe, drill press, etc.) which uses a computer to electronically control the motion of one or more axes on the machine.

- The development of NC machine tools started from a task supported by the US Air Force in the early 1950's, involving MIT and several machine-tool manufacturing companies. The need was recognized for machines to be able to manufacture complex jet aircraft parts.

- As computer technology evolved, computers replaced the more inflexible controllers found on the NC machines; hence the dawn of the CNC era.

- CNC machine tools use software programs to provide the instructions necessary to control the axis motions, spindle speeds, tool changes and so on.

- CNC machine tools allow multiple axes of motion simultaneously, resulting in 2D and 3D contouring ability.

- CNC technology also increases productivity and quality control by allowing multiple parts to be produced using the same program and tooling.

Basics of CNC Programming

There are two ways to program modern CNC machine tools.

1. **Conversational Programming.** This is the simpler of the two methods. In effect, this is a macro programming language used to instruct the machine to perform pre-programmed cycles (i.e. facing, drilling holes in arrays, etc.). When writing a conversational program, you simply enter the appropriate parameters associated with each machining cycle. *This is analogous to using the polar array function in SolidWorks or Pro/E; you don't have to do the layout or trig to find the location of the features; you just specify the essential parameters and the software does the rest for you.*

2. **CAM Programming.** This is the more powerful of the two methods. Using this method, you import your part model into a CAM (computer aided manufacturing) program and define the parameters associated with each and every machined feature on the part. These parameters include tool diameter and length, depth of cut, tool path geometry, etc.

Conversational CNC Programming

The following cycles are typical of the machining operations available when programming a 3-axis CNC milling machine.

Position. Used to move the XYZ coordinates at rapid feedrate.

Drill_one. Used to position the tool at a specific XYZ coordinate position in order to automatically drill a hole. The automatic drill cycles allow for simple drilling, peck drilling, spot-facing and bore cycles.

Drill_pattern. Used to define polar or rectangular hole arrays for automatic drilling.

Line. Used to cut straight lines along an axis or a diagonal at the desired feedrate.

Arc. Used to cut a circle or partial circle that is part of a series of cuts that usually includes lines as well.

Face. Used to define a rectangular zig-zag pattern used to clean off a part surface.

Pocket. Used to clear the material out of a rectangle, circle or polygon.

Frame. Used to cut the inside or outside outline of a rectangle, circle or polygon.

Tool. Used to enter tool parameters, machine function parameters and program pause/stop codes.

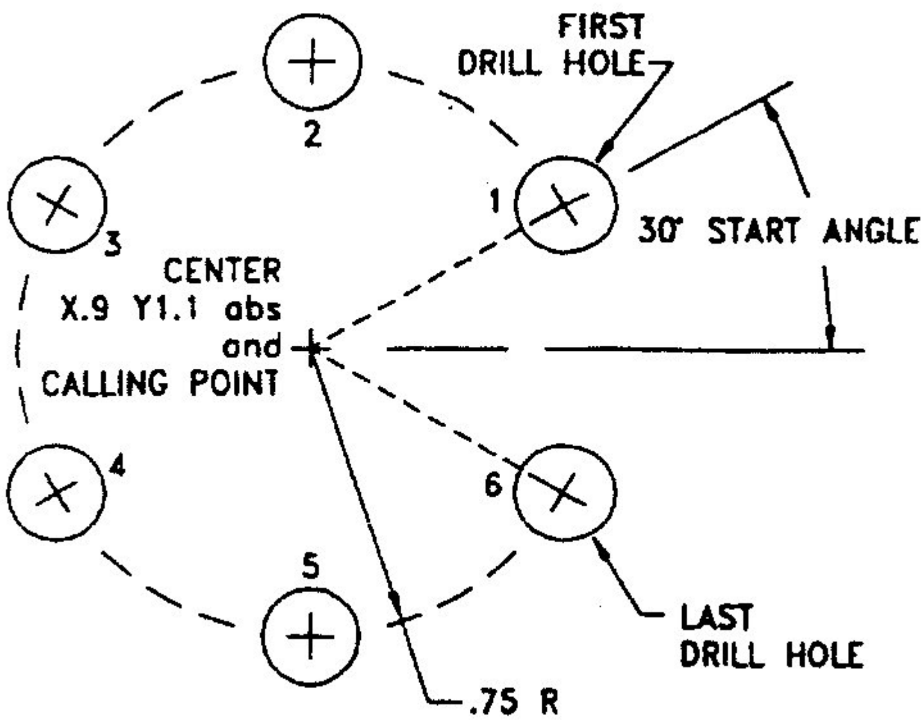
Scale/mirror. Used to scale and/or mirror other part features.

Rotate. Used to repeat other part features around a specific center of rotation.

Conversational CNC Programming Example #1

Drill Pattern Bolt Circle Variables (G121):

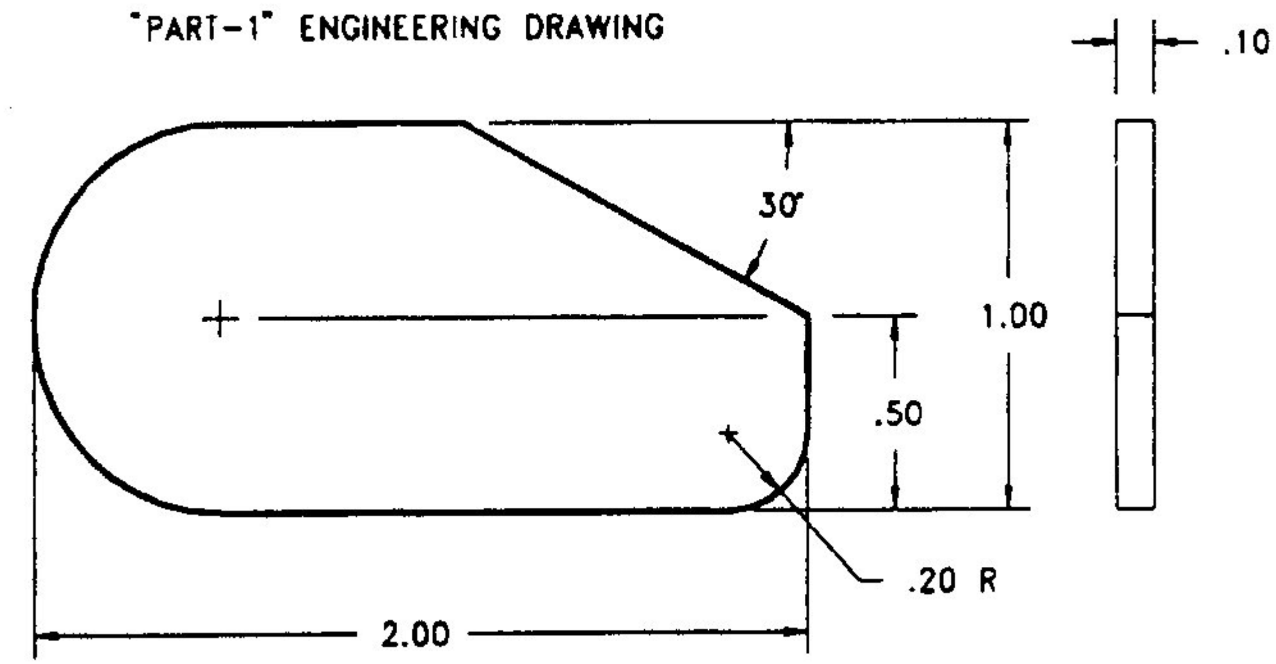
X = X center
Y = Y center
R = Radius
A = Start angle (absolute)
N = # of holes
H = # of holes to drill

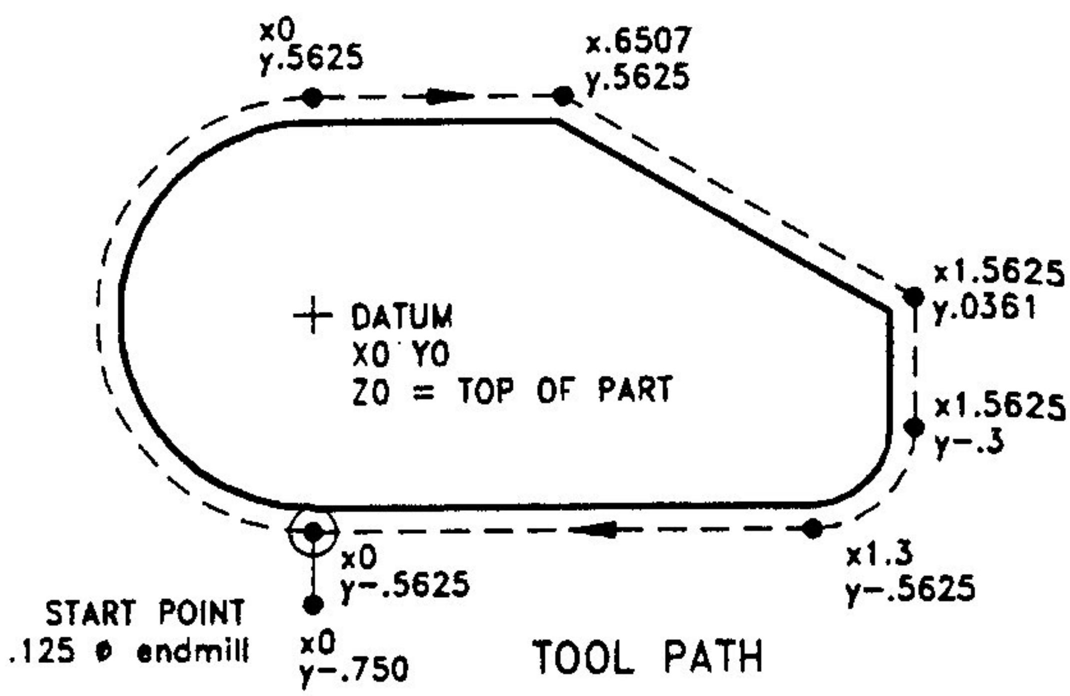



```
N10 G90 G0 X0 Y0
N15 X.9 Y1.1
N20 G121 X.9 Y1.1 R.75 A30 N6 H6 M0
```

Conversational CNC Programming Example #2

Arcs and Lines (dashed line is tool path for 1/8” diameter endmill)





Conversational CNC Programming Example #2 (con't)

Below is the actual tool path code for the previous example. After the user enters the basic parameters, this is the program that is generated by the conversational interface to run on the CNC.

An analogy to software programming is that conversational programming is similar to programming using a compiler (ie C, Fortran, VB, etc.) and the actual tool path code generated is equivalent to the final compiled machine code or instructions.

```
G90 G0 X0 Y-0.75 Z1 F5          [G90=absolute; G0=rapid; F=XY feed]
Z0 M3                           [M3=spindle on, CW]
G1 Z-0.1 E2                     [G1=linear motion; E=Z feedrate]
Y-0.5625
G2 J0.5625 X0 Y0.5625           [G2=CW circular motion]
G1 X0.6507                      [G1=linear motion]
X1.5625 Y0.03608
Y-0.3
G2 I-0.2625 X1.3 Y-0.5625       [G2=CW circular motion]
G1 X0                           [G1=linear motion]
G0 Y-0.75 Z1                    [G0=rapid]
M30                             [M30=end of program/rewind]
```

CNC CAM Programming

Once the part has been designed using conventional mechanical design methods (structural analysis, FEA, fatigue study, etc.), the part is manufactured using the following method.

1. Create a solid 3D model of the part to be produced. Any standard CAD format is acceptable.

2. Import the solid model into the CAM (***computer aided manufacturing***) software. (this demonstration uses MasterCAM)

3. Input the raw material stock size and set the part's coordinate origin.

4. Input the necessary information for each tool used in machining the part features. Typically, a tool library will exist, which is simply a database of tools and their related parameters.

5. For each part feature, select the appropriate tool from the library and set the parameters necessary for machining that feature. Typical parameters include spindle speed, depth of cut, feedrate, number of passes, tool path pattern, etc.

6. Verify the programmed tool path(s) by running the CAM software's virtual machining cycle.

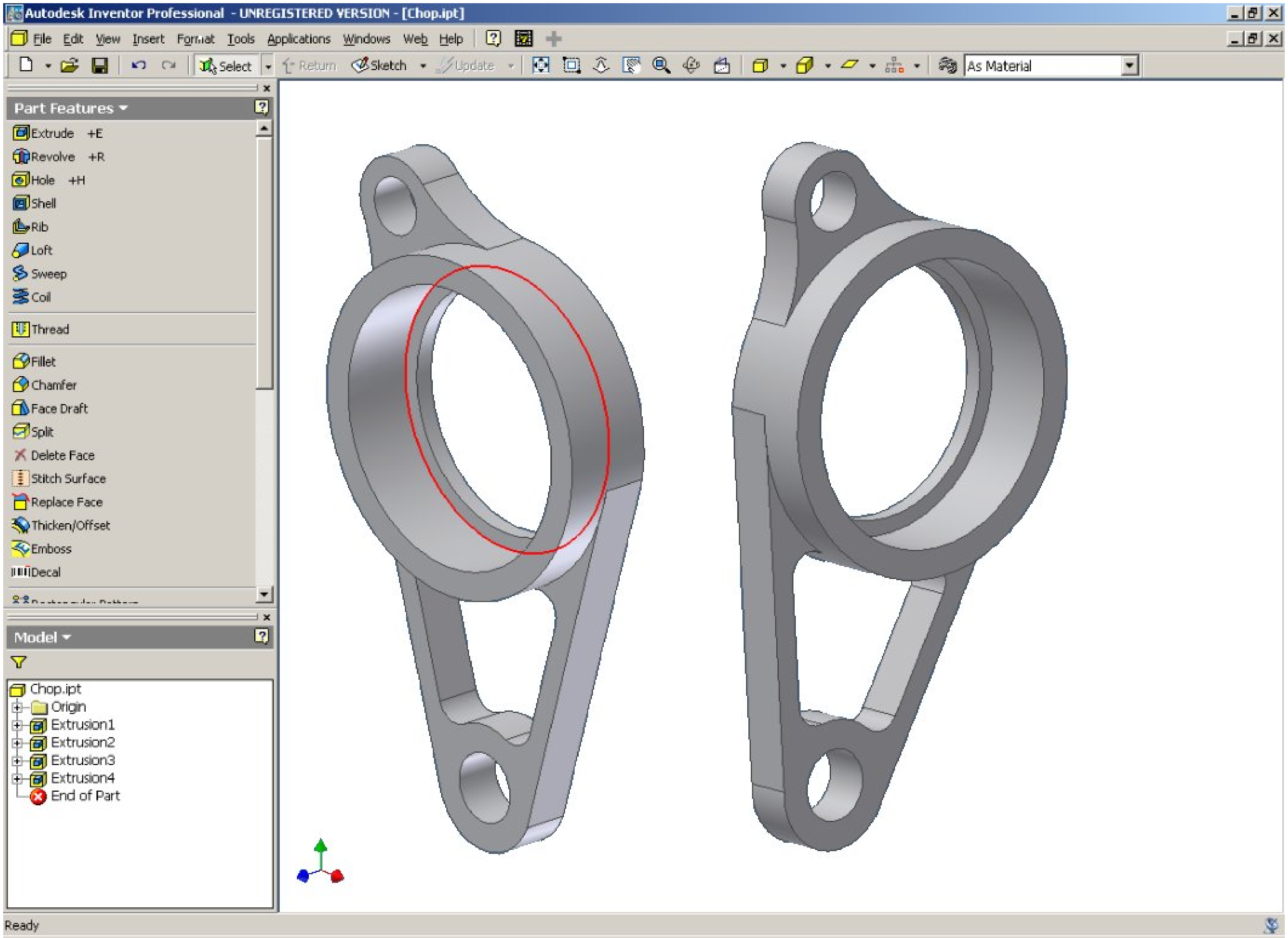


Figure 1. Inventor CAD model of example part (mirrored for clarity).

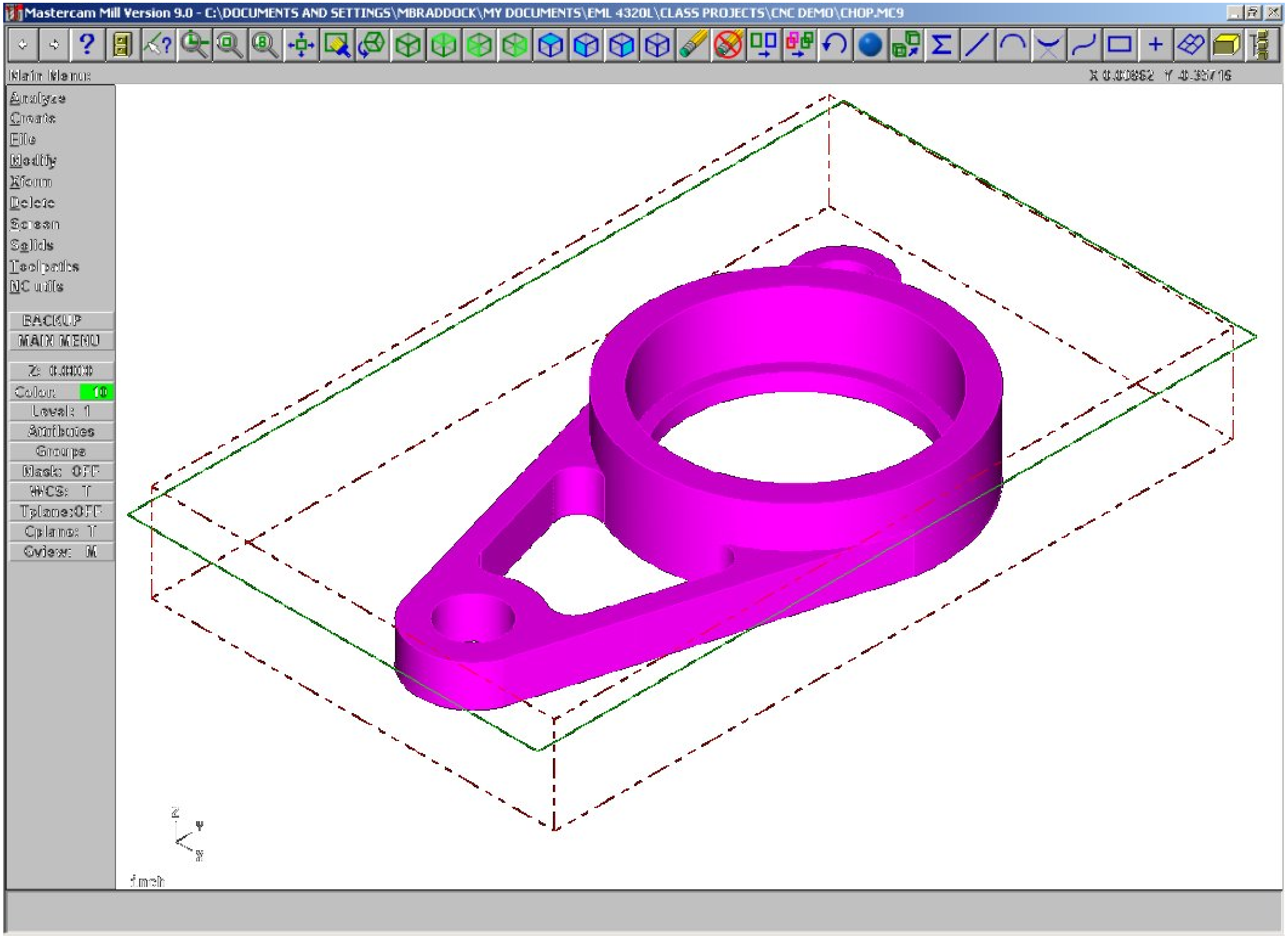


Figure 2. CAM part setup and coordinate zeroing.

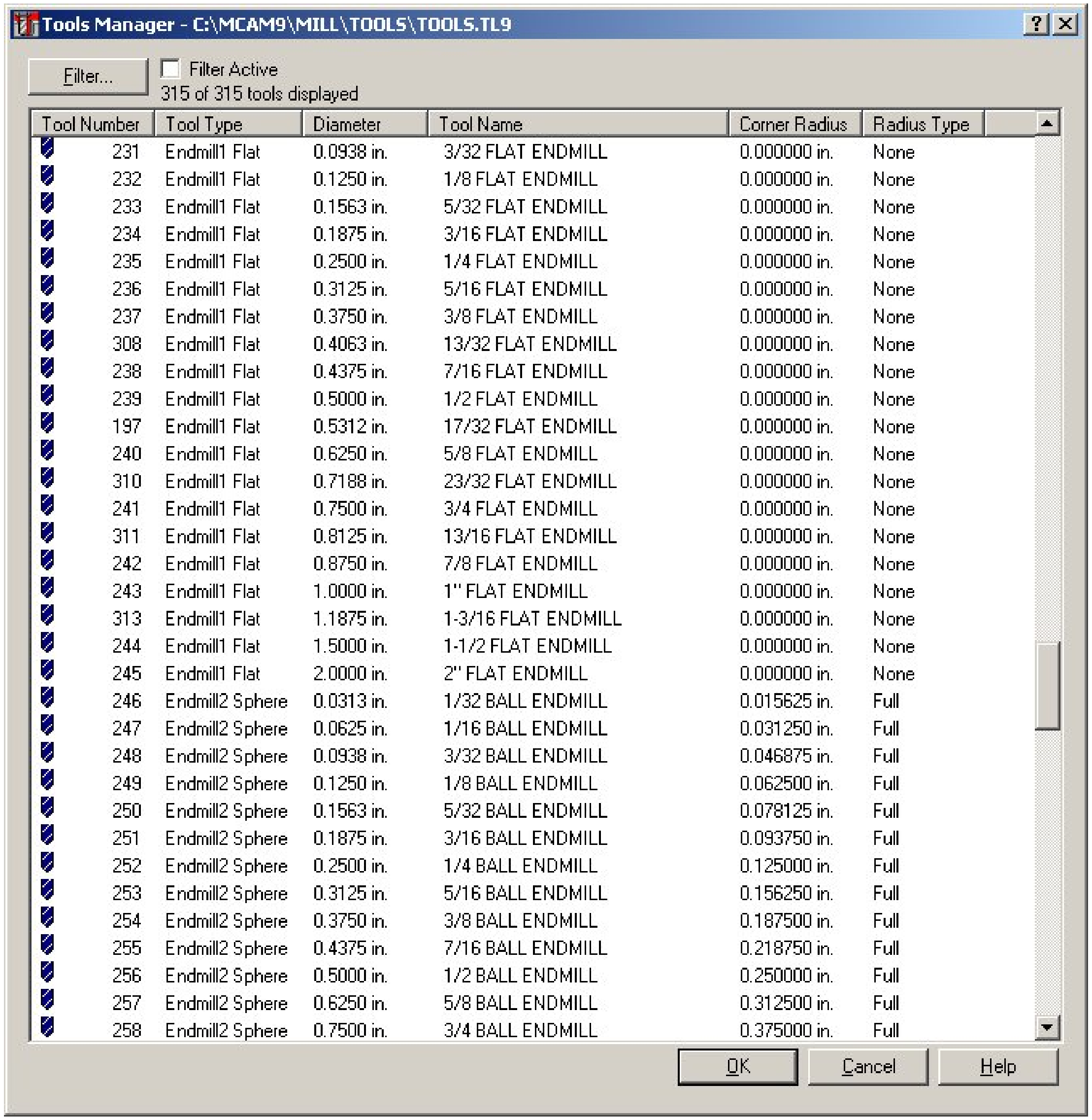


Figure 3. Tool library showing database of previously used tools.

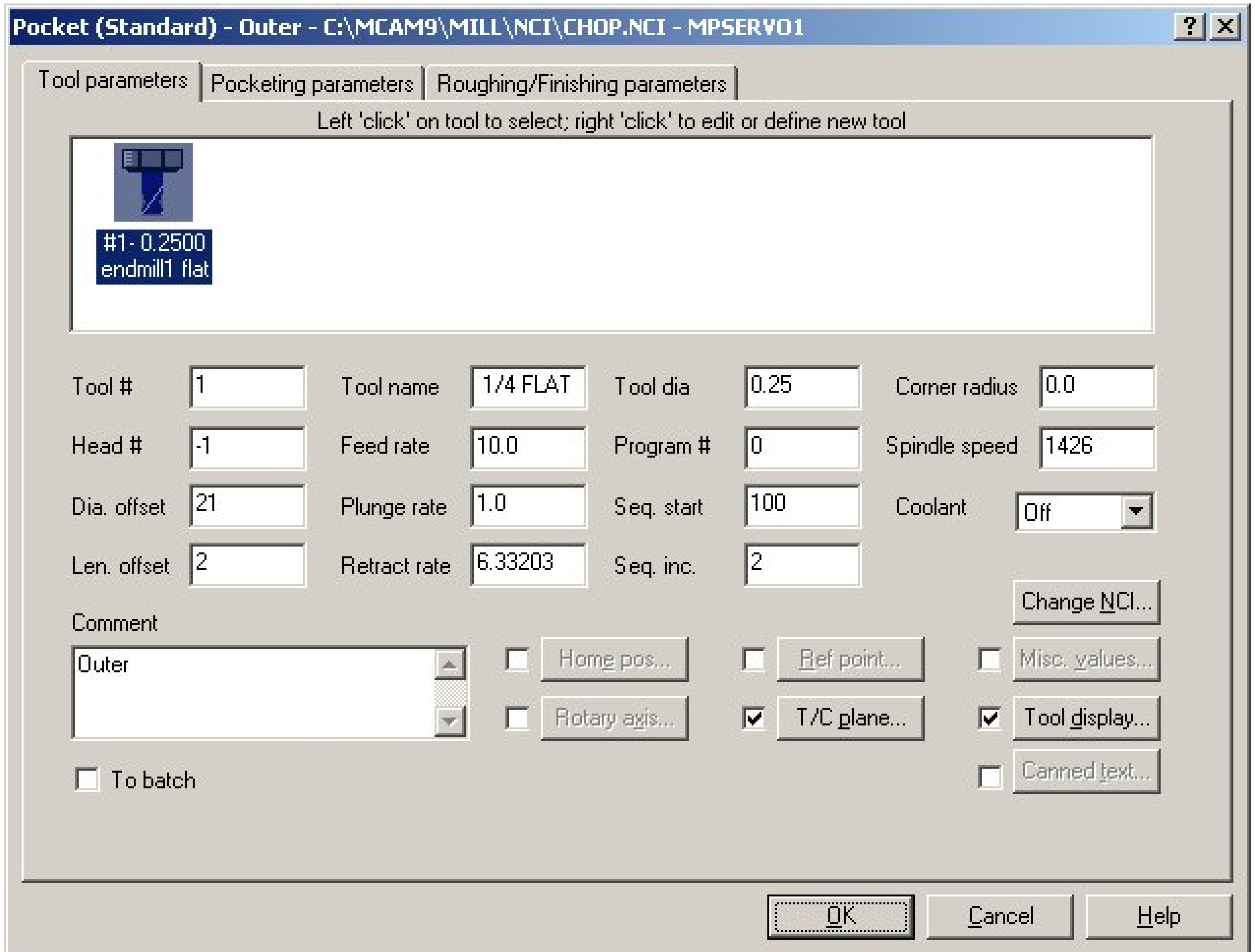


Figure 4. Tool parameters stored for each cutting tool used.

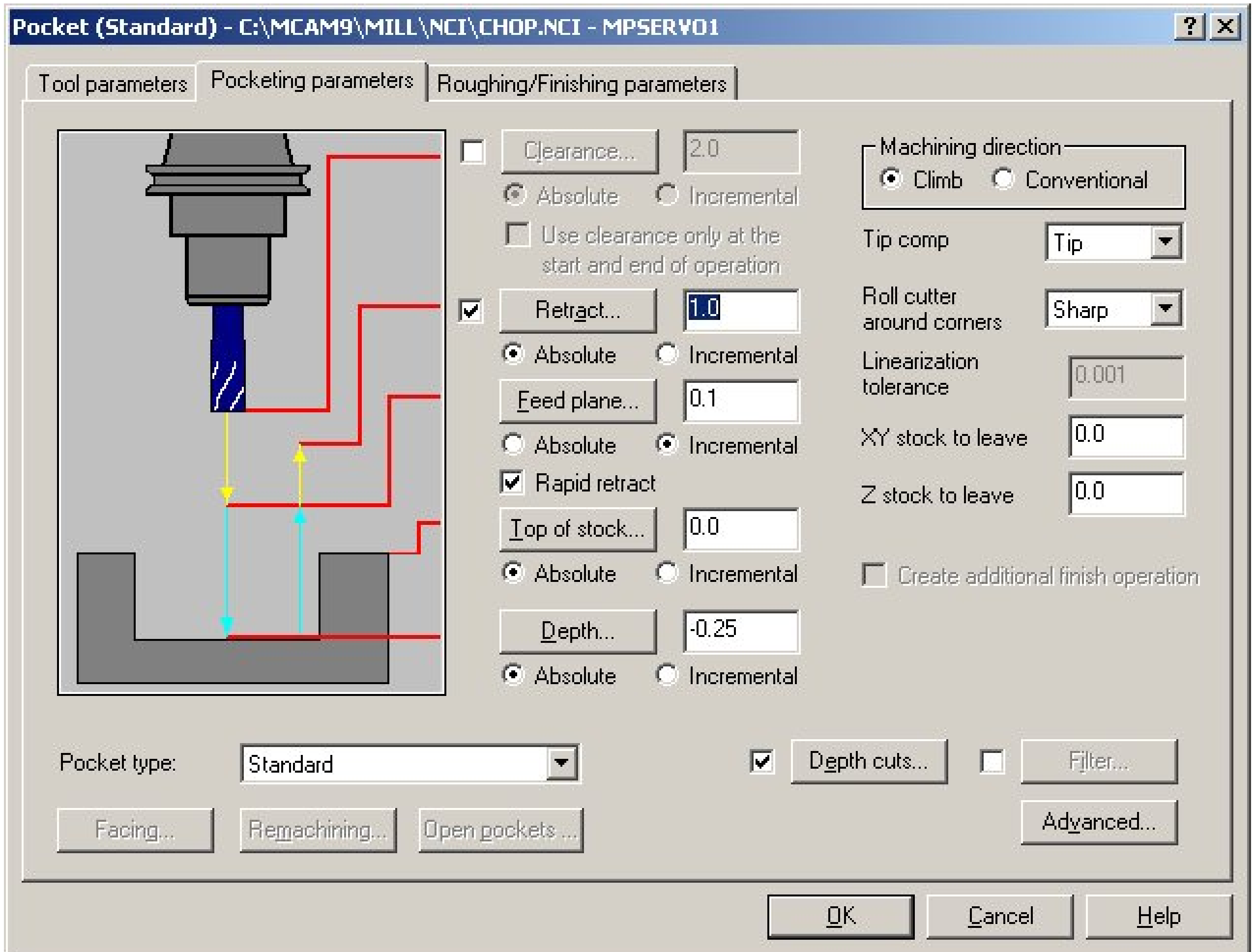


Figure 5. CAM parameters for cutting one feature (pocket) in the part.

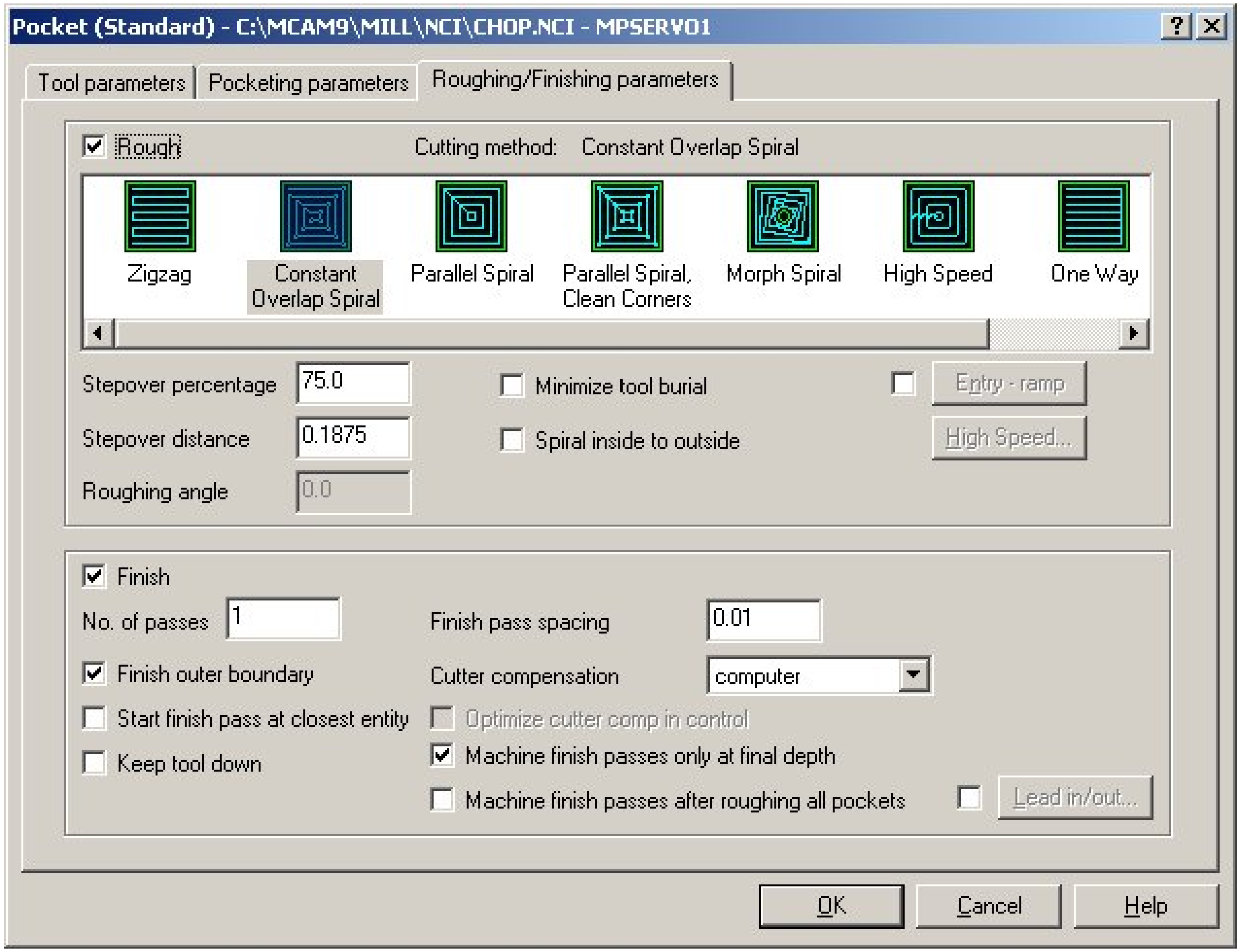


Figure 6. CAM roughing and finishing parameters.

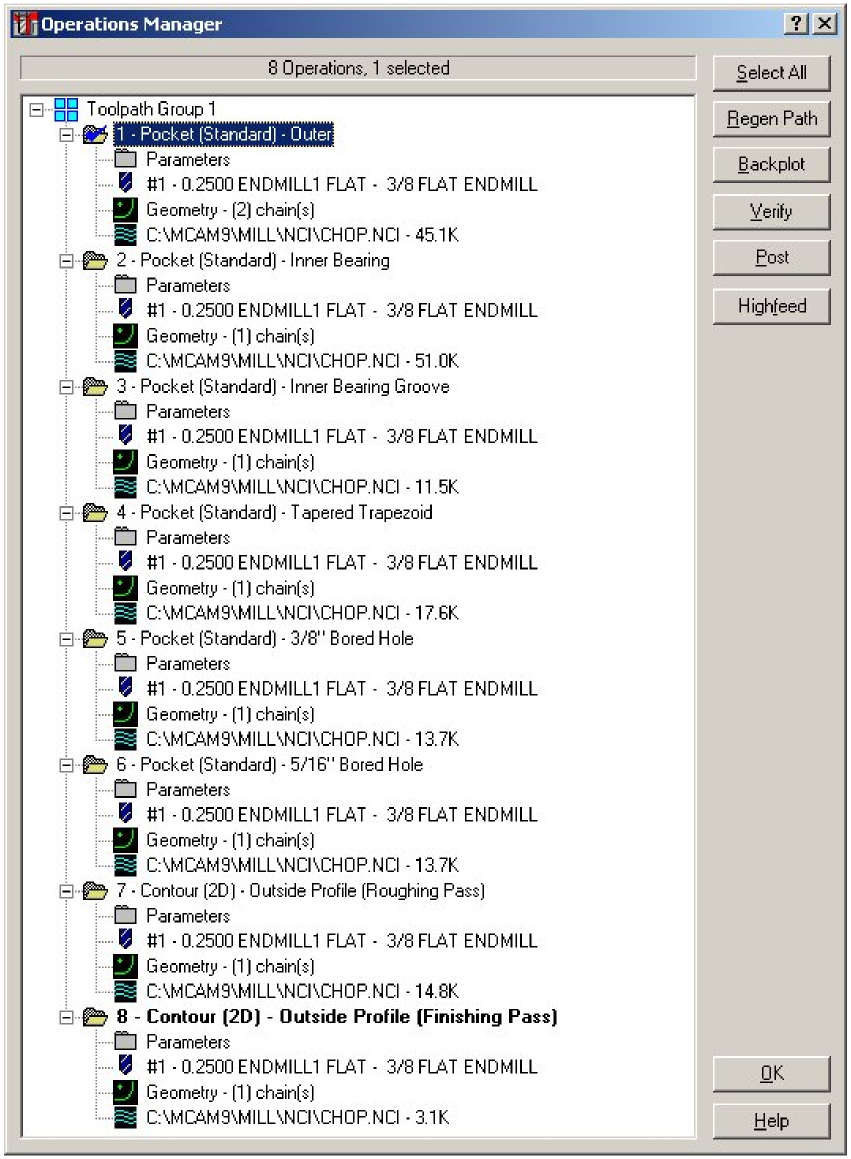


Figure 7. CAM operations list showing all cutting operations & tools.

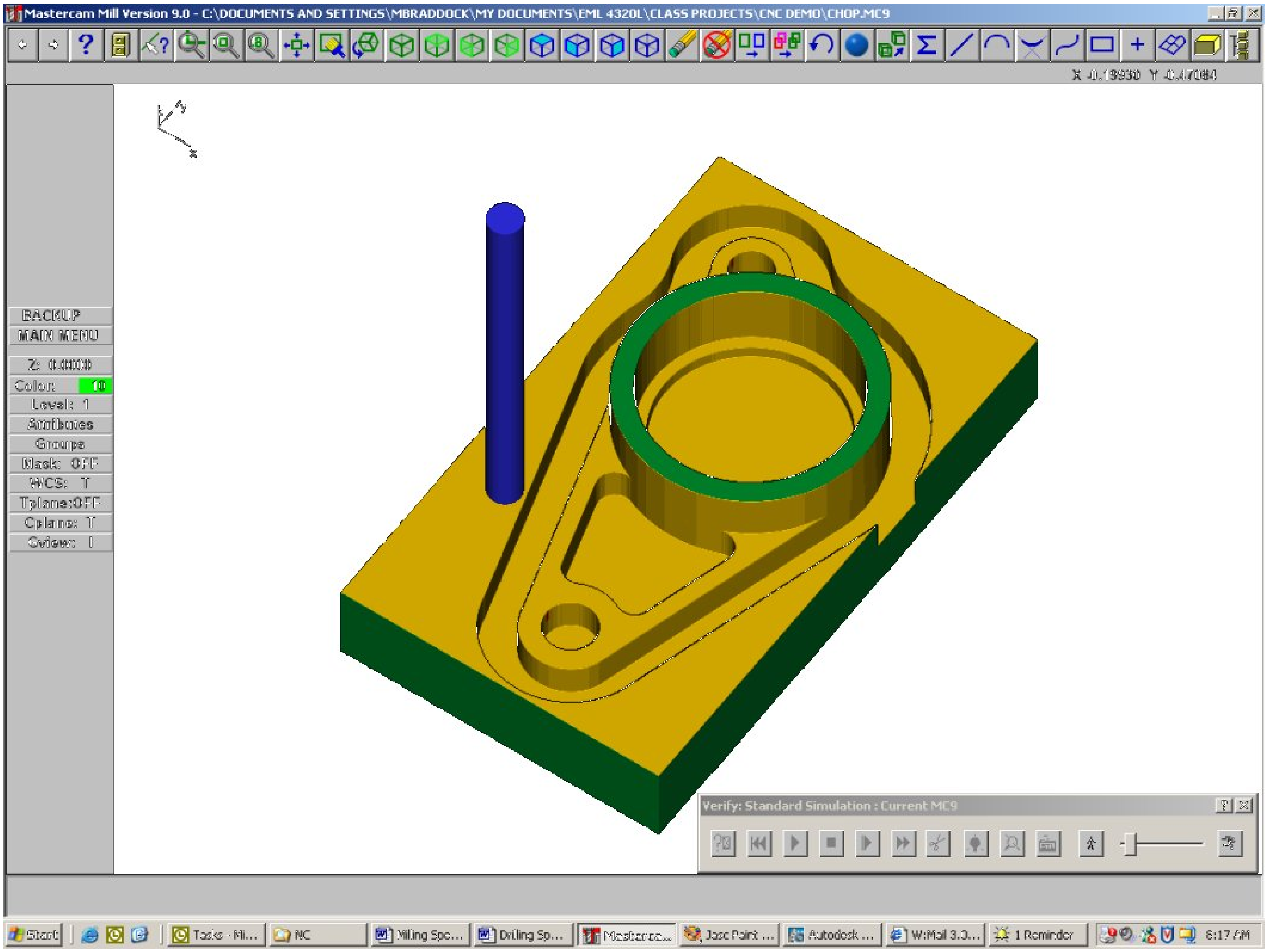


Figure 8. Virtual verification cycle used to catch errors before cutting.

CHOP.NC

```
( CHOP NC    DATE=DD-MM-YY - 20-10-03  TIME=HH:MM - 20:57 )
(  3/8 FLAT ENDMILL   TOOL - 1 DIA. OFF. - 21 LEN. - 2 DIA. - .25 )
N100 G70
N102 G00 G40 G80 G90
/ N104 G91 Z0.
/ N106 X0. Y0.
/ N108 G92 X0. Y0. Z0.
N110 T1
N112 G00 G90 X-1.2427 Y1.5415 S1426 M03
N114 Z1.
N116 Z.1
N118 G01 Z-.05 E1.
N120 X1.0315 Y1.5548 F10.
N122 Y.1194
N124 G03 X-1.0384 Y0. I-1.0315 J-.1194
N126 X1.0315 Y-.1194 I1.0384
N128 G01 Y-2.6286
N130 X-1.2427
N132 Y1.5415
N134 X-1.0802 Y1.3839
N136 X.869 Y1.3964
N138 X.8565 Y1.2964
N140 Y.8839
N142 X.8815 Y.7464
N144 X.8307 Y.8839
N146 X.6547 Y1.0214
N148 X.4578 Y1.1214
N150 X.2345 Y1.1839
N152 X.139 Y1.1964
N154 X-.139
N156 X-.3455 Y1.1589
N158 X-.5679 Y1.0714
N160 X-.7593 Y.9464
N162 X-.9365 Y.7714
N164 X-1.0702 Y.5714
N166 X-1.0983 Y.4964
N168 X-1.1285 Y.2339
N170 X-1.0677 Y.6339
N172 Y1.3339
N174 X-.8927 Y1.2214
N176 X-.494 Y1.2464
N178 X-.6171 Y1.2339
N180 X-.7383 Y1.1839
N182 X-.8861 Y1.0839
N184 X-.9052 Y.9964
N186 X-.8802 Y1.1589
```

Figure 9. Final program ready to be processed by the CNC machine.

Final Facts about CNC Machining

- CNC manufacturing offers advantages on two types of parts: (1) simple parts that are mass produced and/or (2) complex parts with features requiring multiple axes of simultaneous motion. For simple parts in low quantity, it is often quicker to produce the parts on manual machines (as in lab).

- CNC does not inherently imply increased part accuracy. An old CNC with a lot of hours of use will produce less accurate features than a new quality manual machine and vise-versa; so don't automatically associate higher accuracy with CNC machines. (Accuracy has more to do with machine design, component selection and mechanical wear.)

- Modern CNC machines offer increased productivity due to stiffer machine and spindle designs, more powerful motors, high pressure coolant (up to 1000 psi) that floods the cutting zone, automatic tool changers, digital workpiece and tool probing, and/or horizontally mounted spindles.

- Downsides to CNC machines are higher initial cost, larger space and electrical requirements, increased maintenance cost, required programming skillset and their inherent complexity means there's a higher probably of component failure during the useful lifespan.