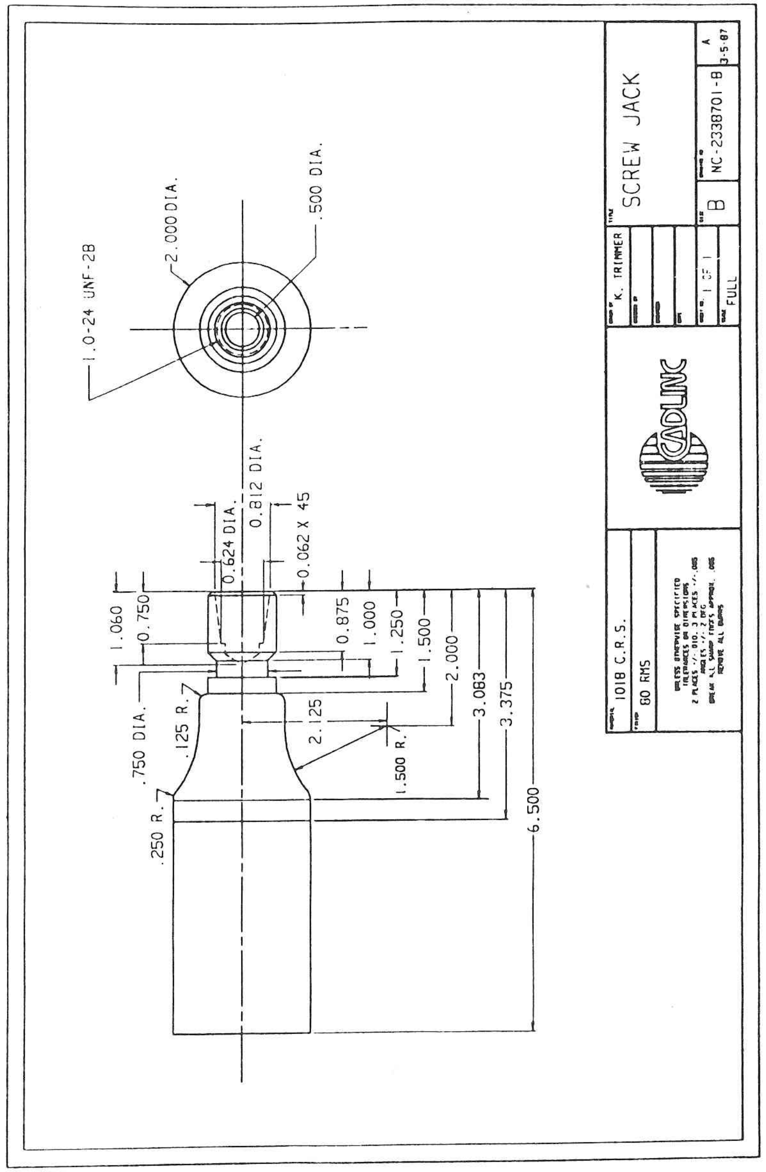


Figure 10.39 *CAD screen displaying headline, graphic icons, and part print in graphics area.*

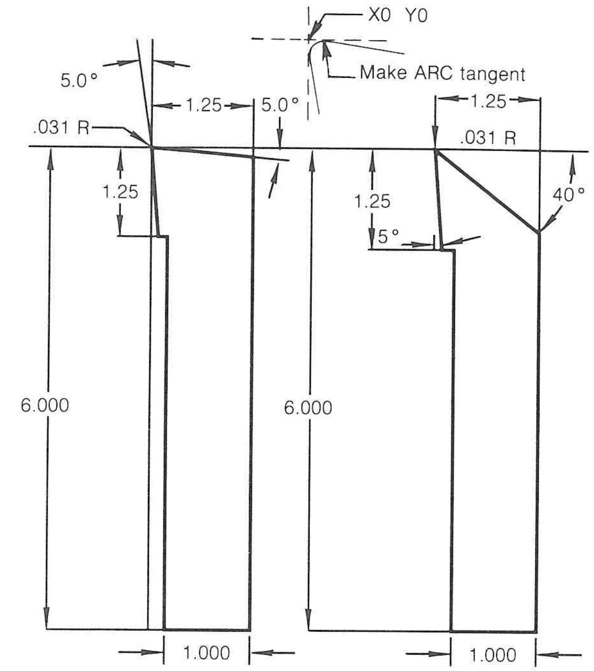


Figure 10.40 *Cutting tool geometry creation for tool library.*

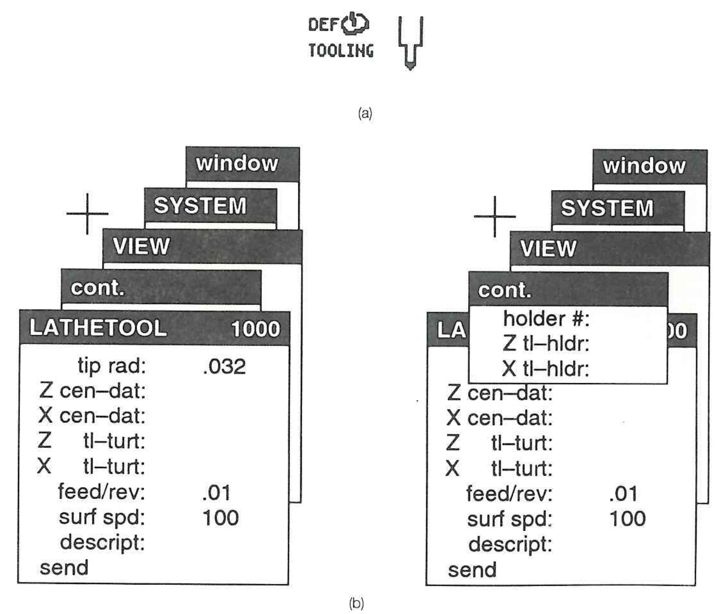


Figure 10.41 *CAM (a) tool definition icons and (b) pop up menus.*

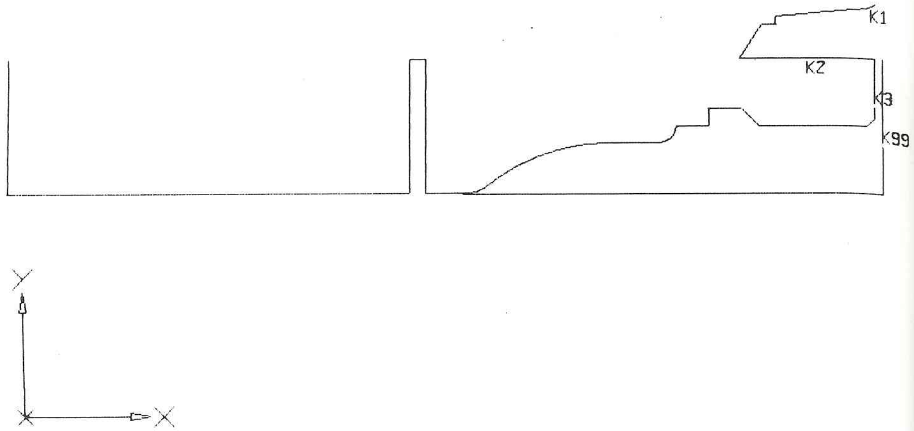


Figure 10.42 *CAD part drawing reduced to required CAM profile curves.*

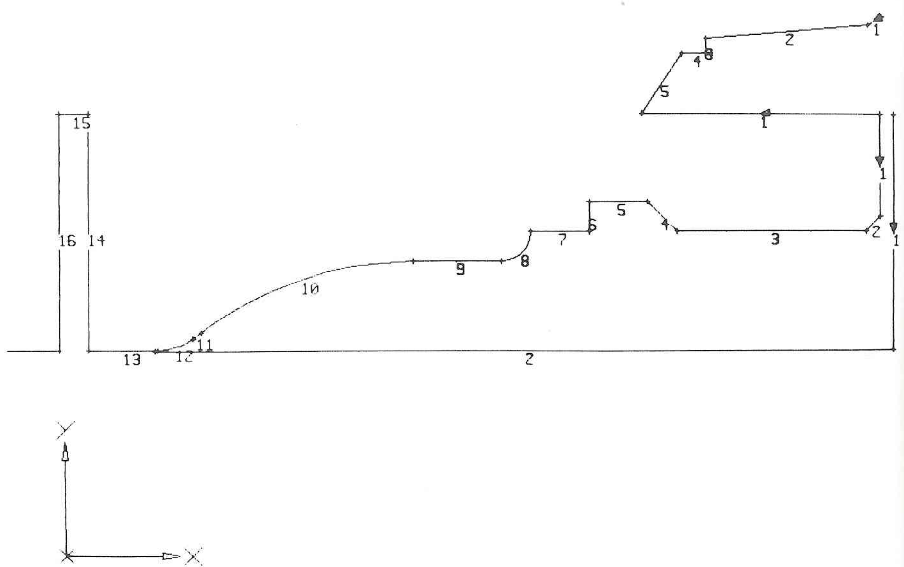


Figure 10.43 *CAM profile curves displayed in spans or segments form.*

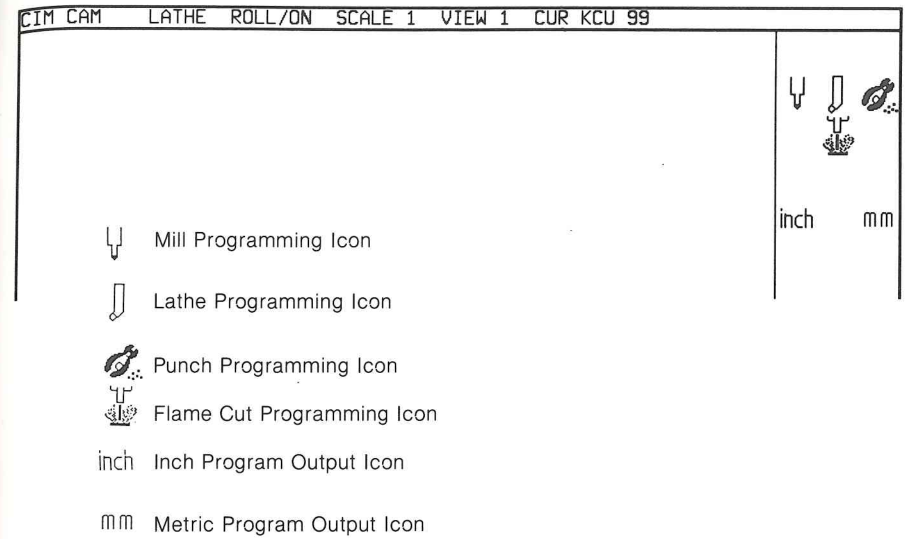


Figure 10.44 *CRT display entering CAM for machining option and inch/metric output. a—mill programming icon; b—lathe programming icon; c—punch programming icon; d—flame cut programming icon; e—inch program output icon; f—metric program output icon.*

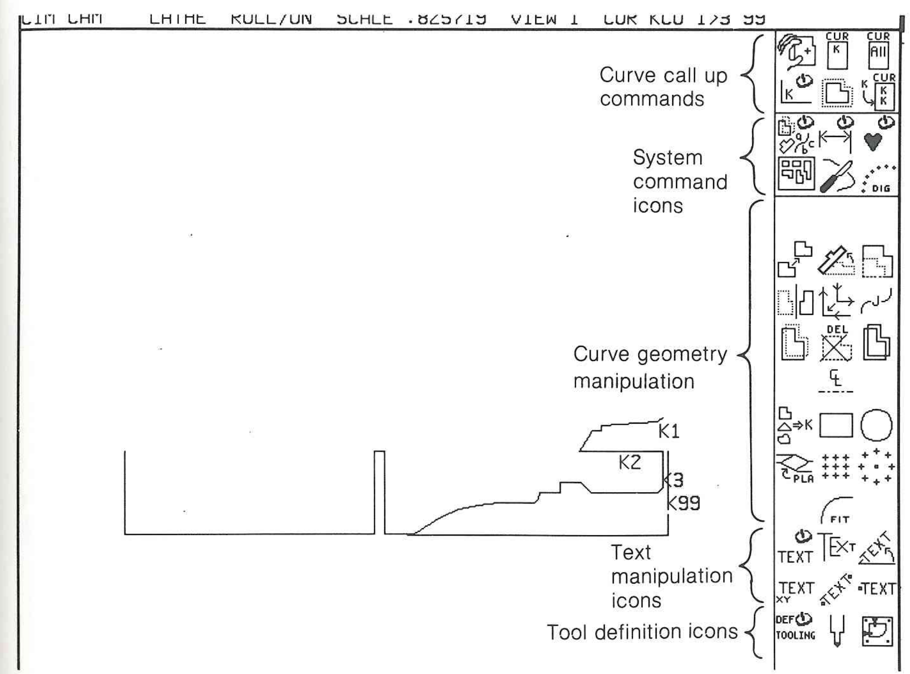


Figure 10.45 *Part curve display in CIMCAM.*

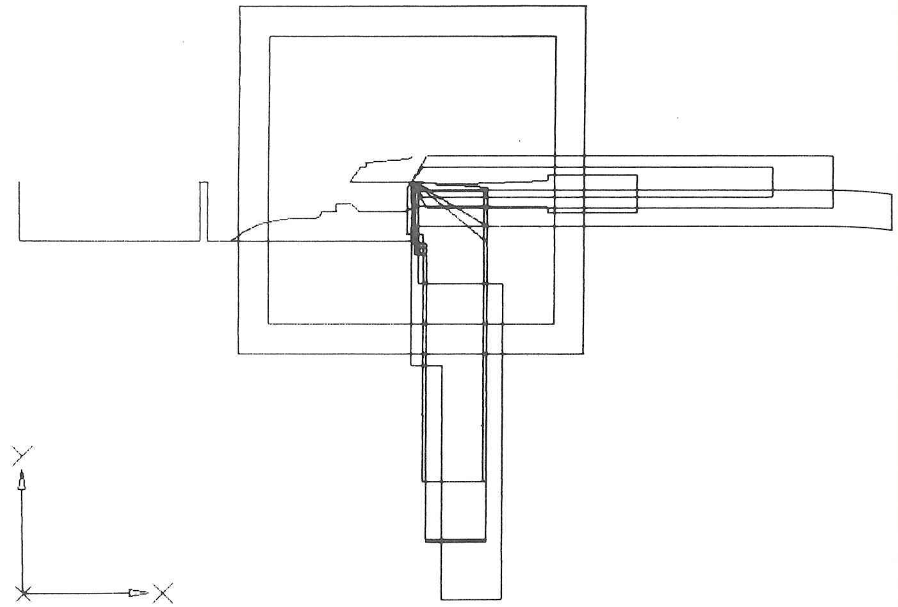


Figure 10.46 *CRT display of part, turret, and tool data indicating common datum point.*

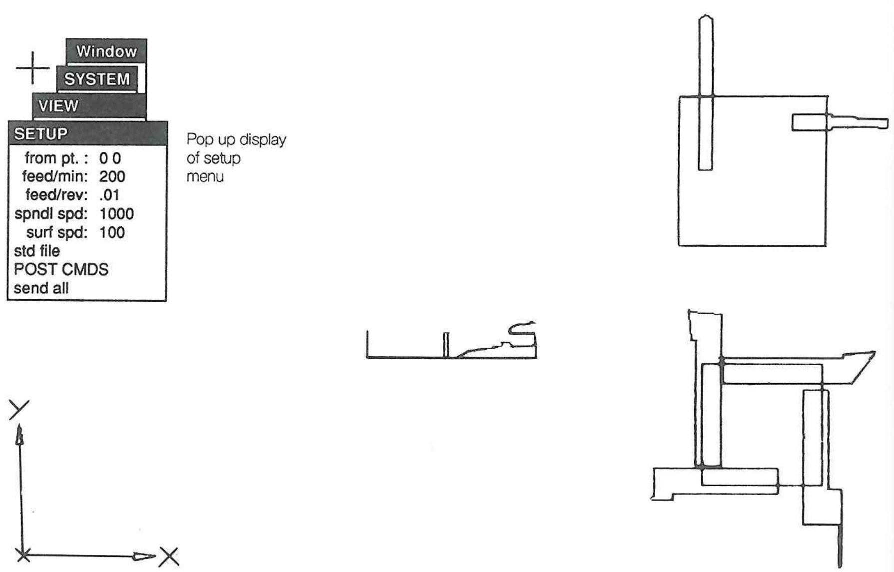


Figure 10.47 *CRT display of part, turret, and tool data after completion of computer setup statement. The setup statement is completed by using the indicated pop-up menu.*

to machine all spans or any percentage of a given span. This kind of freedom allows for any type of machining pass to be created.

Now that the CAD data are generated, the programmer can enter the CAM system to start the creation of machining paths. The machining system is entered by typing in the command CIMCAM. The computer system responses by asking for part data and tool library input files and a name for the output file. The CRT will then display the screen in Figure 10.44 so the type of machining to be performed (milling, lathe work, punching, or flame cutting), and inch or metric output for the machine tool can be selected. The majority of CAD/CAM systems today will allow options as to inch or metric output, and at least milling and lathe work selections. The selections on this system are completed by selecting icons with the mouse or typing the command.

Once CIMCAM is entered, the programmer can then display the machining curves or the machine's tool curves or both as in Figures 10.45 and 10.46. Using the top six icons the programmer can show and close-up on just one curve or show all curves at once. Figure 10.47 indicates the programmer's next task of properly orienting the part and machine tool turrets of tools. Figure 10.47 shows this operation completed. The programmer could then use the system to dimension Figure 10.48 and give it to the machine operator as setup instructions.

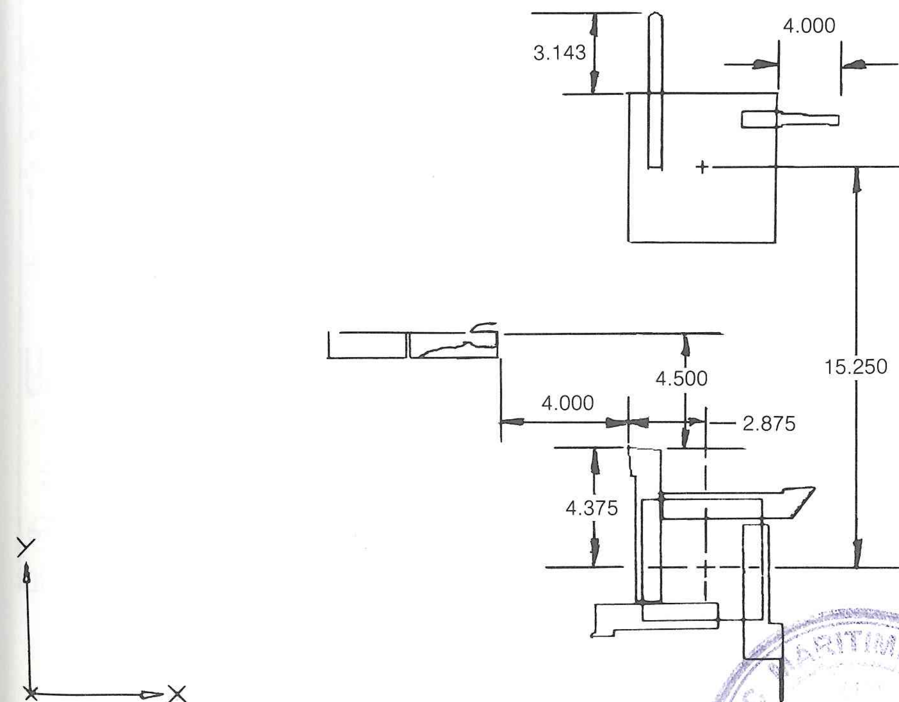


Figure 10.48 *CRT display of part, turret, and tool setup dimensions.*

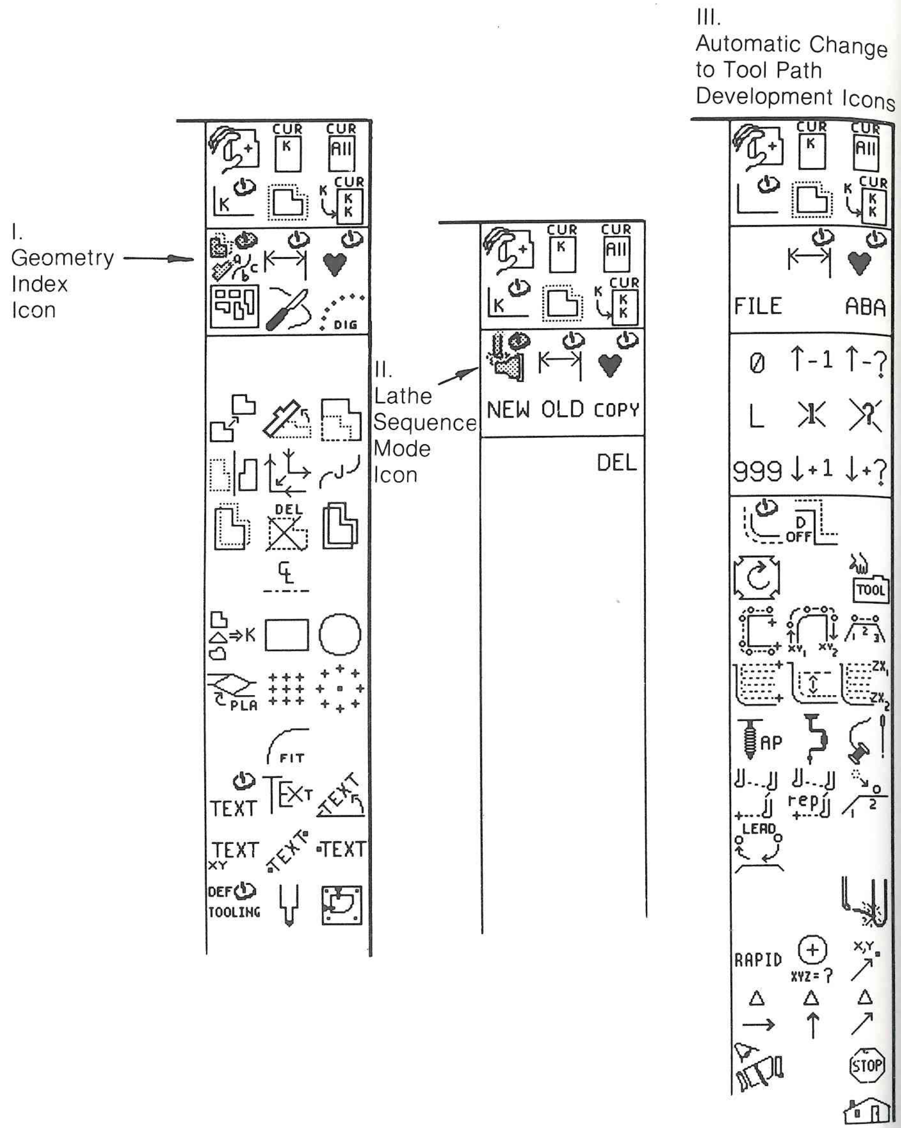


Figure 10.49 *Icon commands for entering tool path generation mode.*

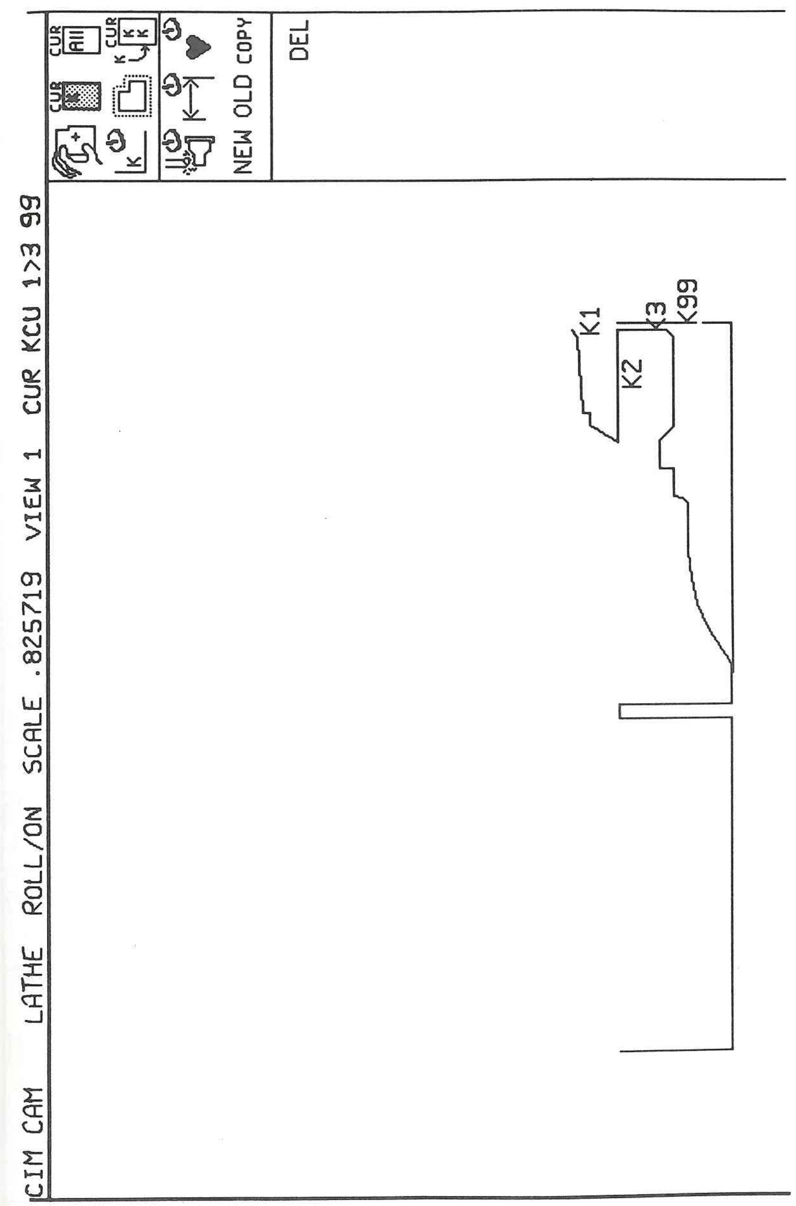


Figure 10.50 *Screen display showing part curves activated by operator.*

The task to perform now is to create the machining paths that you would like the computer to create for you in the machine tools language. In order to do this, the program must enter the machining sequence mode by activating the geometry index icon to switch to the sequence or operation mode index icon display, Figure 10.49. With the sequence mode icons active, the programmer selects a new operation and enters a operation number, automatically going into the machining mode.

With tool path generation icons active, the programmer then goes step by step through the process allowing the computer system to generate the needed tool path. Figures 10.50–10.61 and computer program (Fig. 10.62) show the computer commands and program statements generated and the graphics commands used to do so.

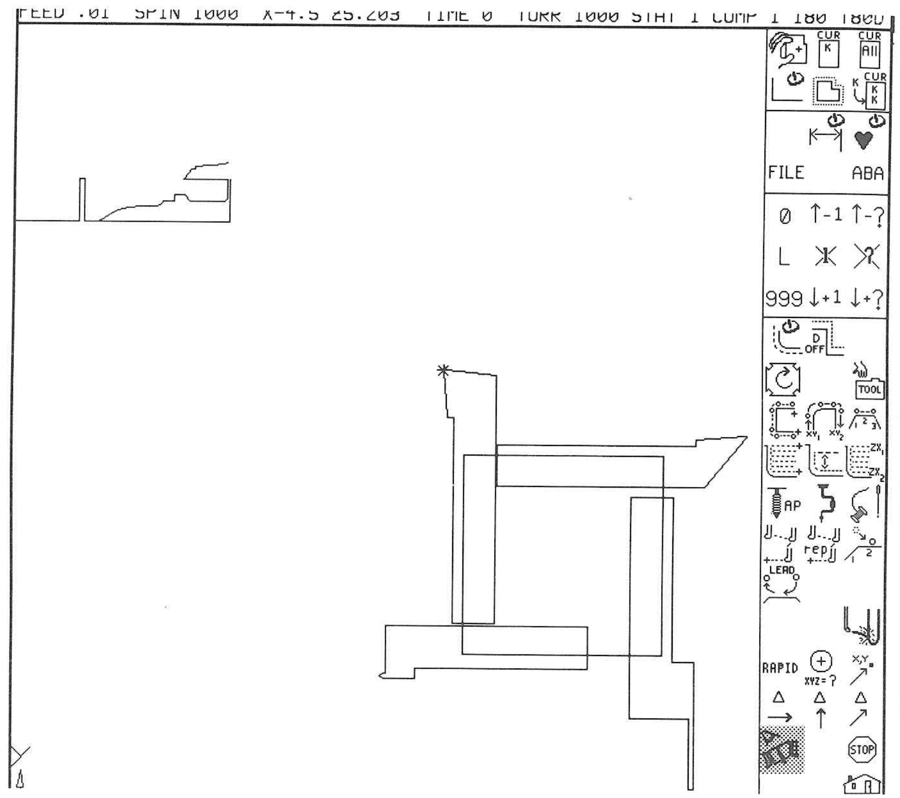


Figure 10.51 *Screen display after the setup statement (Fig. 10.47) in pop-up menu has been completed. Turret display icon has been activated.*

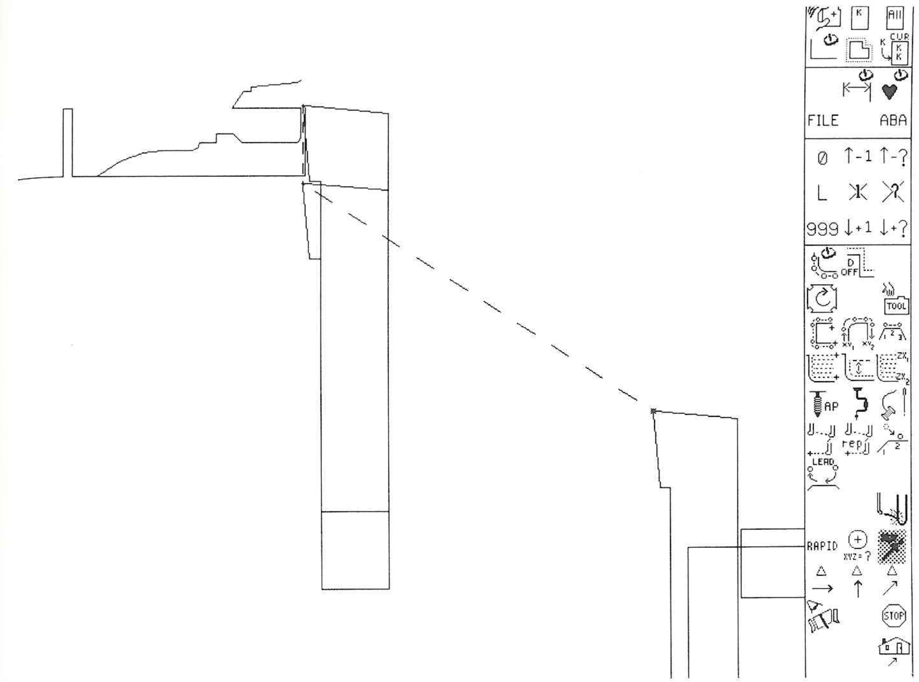
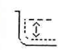
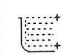


Figure 10.52 *Interactive CRT display of facing operation and computer icons. (See Figure 10.62 for computer statements generated, GOTO commands—sequence 10, lines 11 and 13.).*

 Icon used to set parameters for rough cutting part.

 Single GO CLEAR icon command used to produce the cut paths shown.

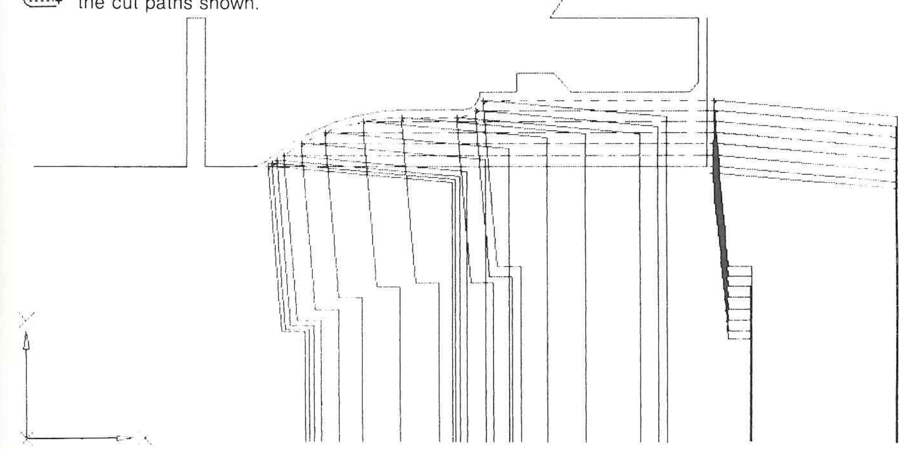


Figure 10.53 *Interactive CRT display of computer-generated roughing passes. (See Figure 10.62 for computer statements generated by GO CLEAR command—sequence 10, line 23.).*

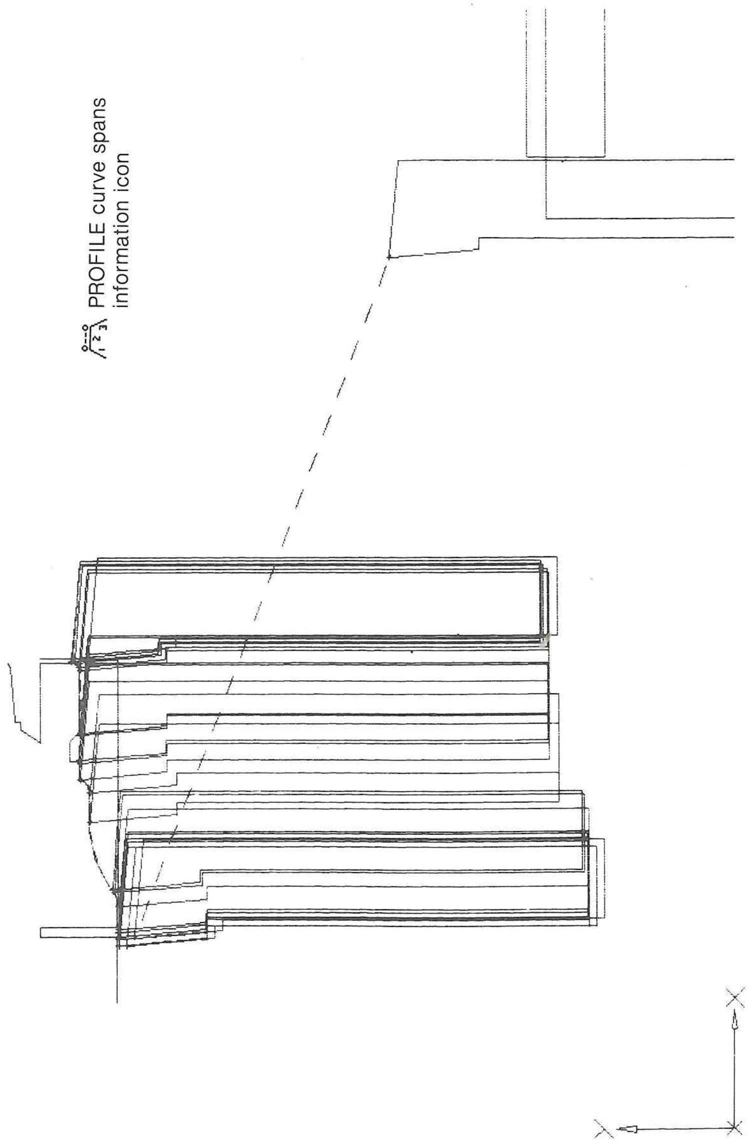


Figure 10.54 *Interactive CRT display of computer-generated semifinish profile pass. (See Figure 10.62 for computer statements generated by profile command—sequence 10, lines 26, 27, and 28.).*

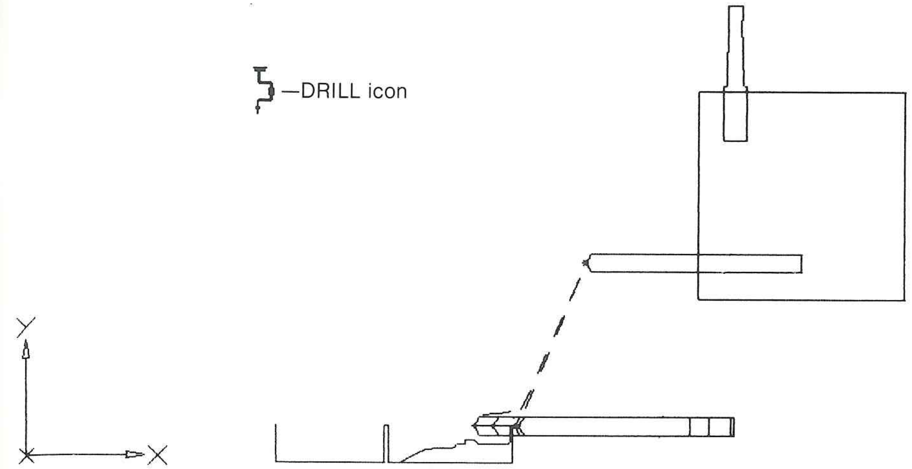


Figure 10.55 *Interactive CRT display of computer-generated drilling cycle. (See Figure 10.62 for computer statements generated by the drilling command—sequence 20, line 11.).*

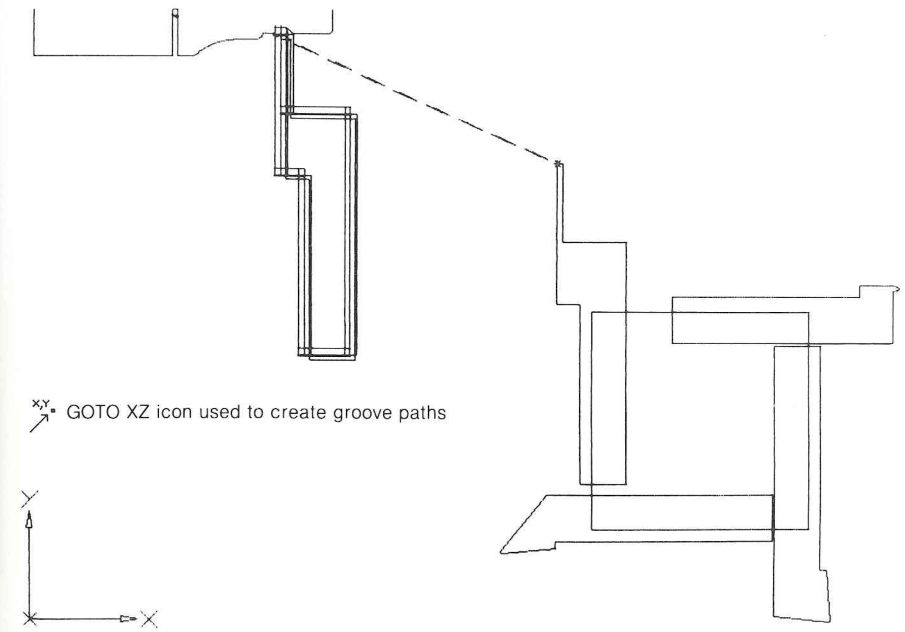


Figure 10.56 *Interactive CRT display of computer-generated grooving operation. (See Figure 10.62 for computer statements generated by the GOTO command—sequence 30, lines 12, 17, and 22.).*

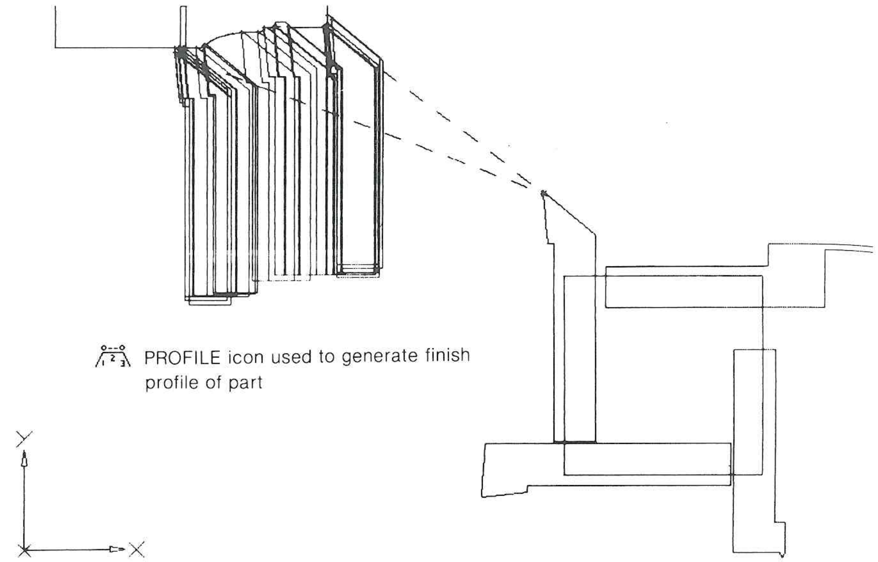


Figure 10.57 *Interactive display of computer-generated finish profile pass on part. (See Figure 10.62 for computer statements generated by the PROFILE command—sequence 40, lines 15, 16, and 17.).*

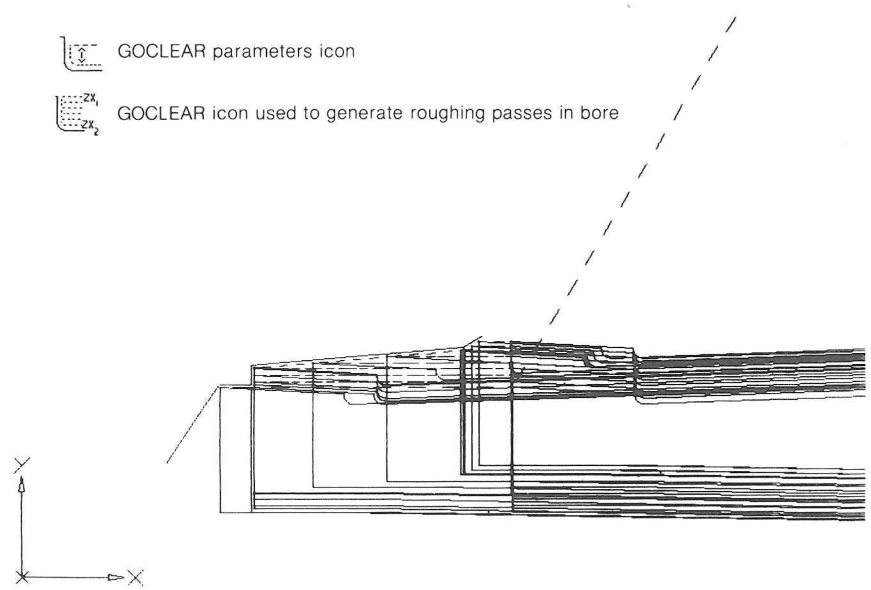


Figure 10.58 *Close-up interactive display of computer-generated rough boring of tapered hole. (See Figure 10.62 for computer statements generated by the GOCLEAR command—sequence 50, lines 11 and 12.).*

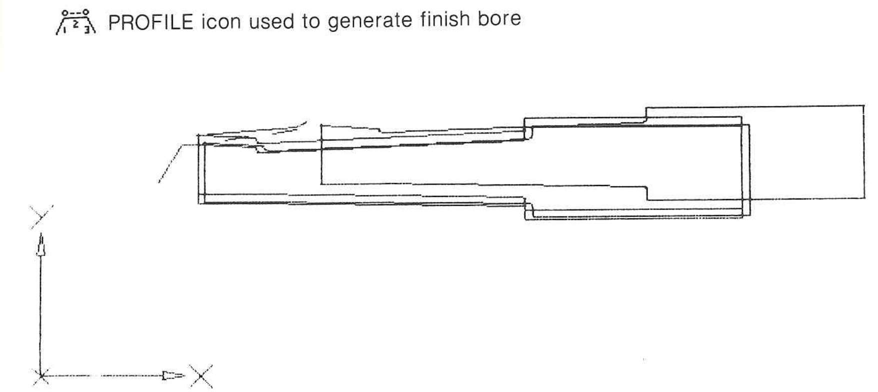


Figure 10.59 *Close-up interactive display of computer-generated finish profile cut on bore. (See Figure 10.62 for computer statement generated by the PROFILE command—sequence 50, line 21.).*

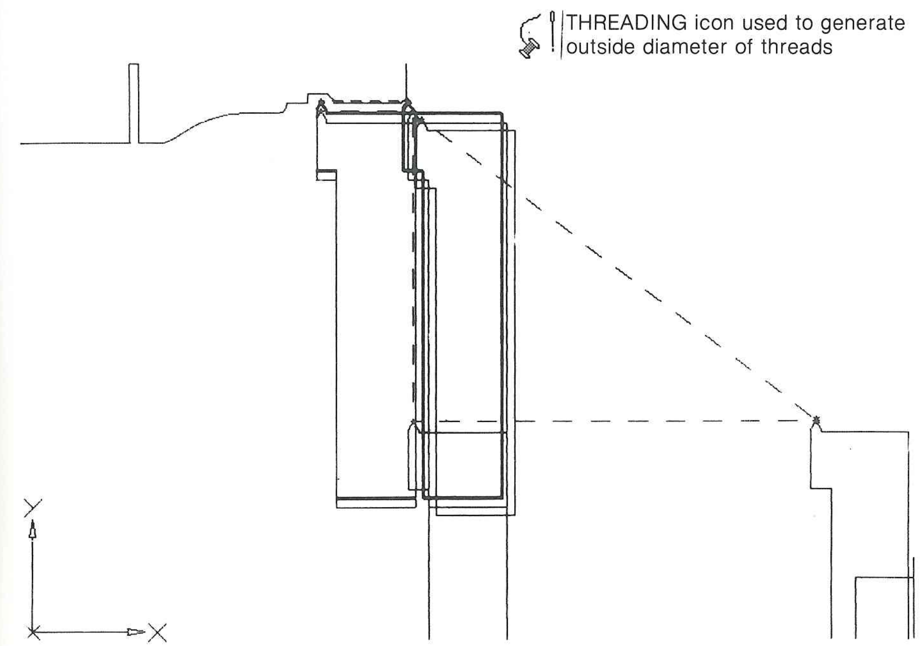


Figure 10.60 *Interactive CRT display of computer-generated threading operation. (See Figure 10.62 for computer statement generated by THREADING command—sequence 60, line 12.).*

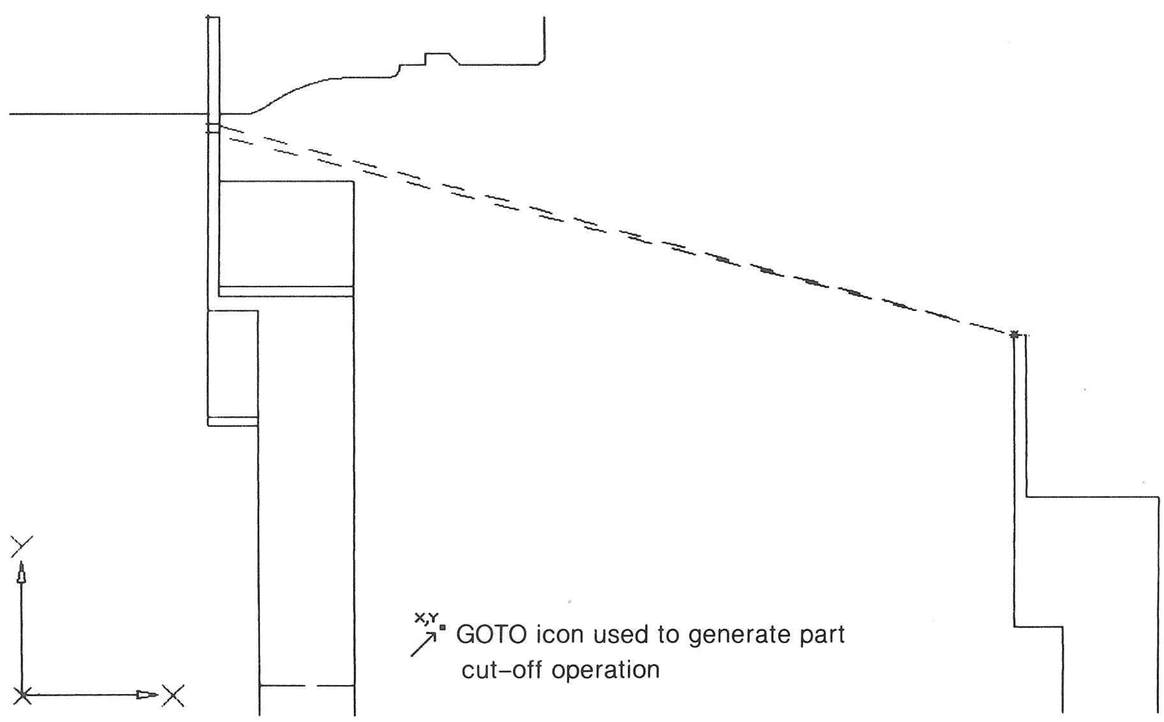


Figure 10.61 *Interactive CRT display of computer-generated part cut-off "parting" operation. (See Figure 10.62 for computer statements generated by GOTO XZ command—sequence 70, lines 11 and 13.).*


```
DATE/042190 TIME/14:44:54 CIM CAM VERSION 4.6
$GNC/
LATHE/
KCU/1
SPA/0,0.405762
SPA/-0.060727,0.370702
SPA/-0.75,0.313262
SPA/-0.75,0.25
SPA/-0.853317,0.25
SPA/-1.021944,0
KCU/2
SPA/0,0
SPA/-1.021944,0
KCU/3
SPA/0,0
SPA/0,-0.438
SPA/-0.062,-0.5
SPA/-0.875,-0.5
SPA/-1,-0.375
SPA/-1.25,-0.375
SPA/-1.25,-0.5
SPA/-1.5,-0.5
SPA/-1.5,-0.5,-0.414214
SPA/-1.625,-0.625
SPA/-2,-0.625,0.162278
SPA/-2.9,-0.925
```

Part Geometry Section

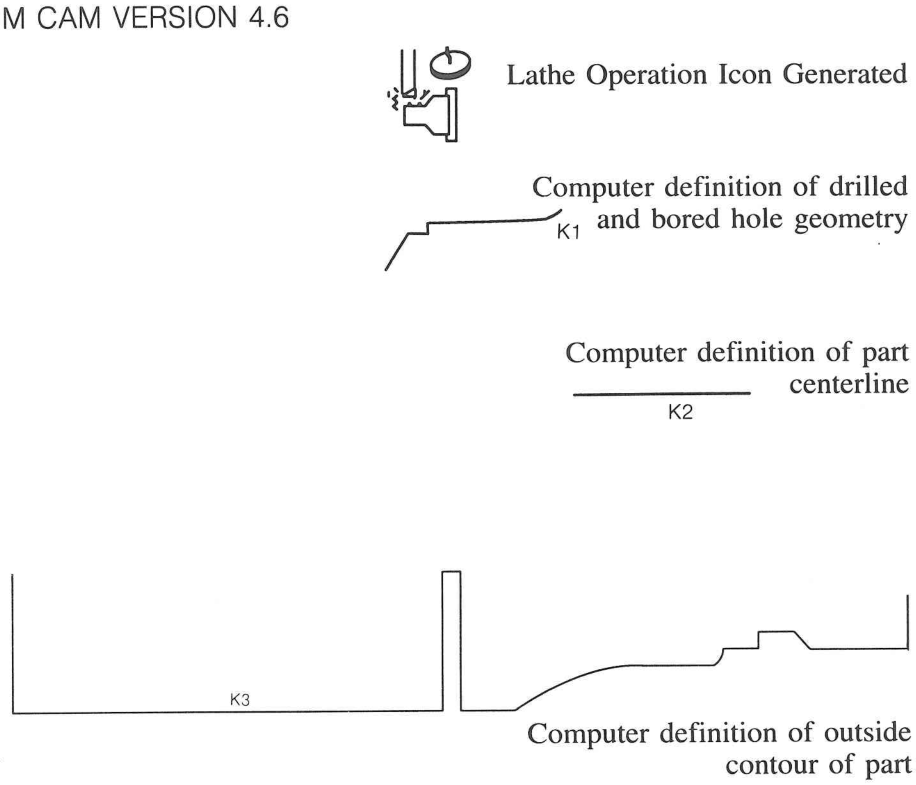
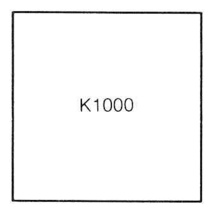
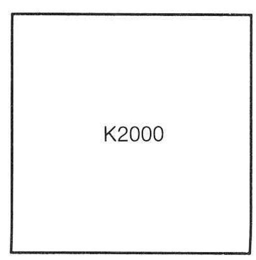


Figure 10.62 *Computer statements generated by the CAD/CAM system from which the CNC program will be created.*

```
SPA/-2.933333,-0.95,-0.162278
SPA/-3.083333,-1
SPA/-3.375,-1
SPA/-3.375,0
SPA/-3.5,0
SPA/-3.5,-1
SPA/-6.5,-1
SPA/-6.5,0
KCU/1000
SPA/2.375,2.375
SPA/-2.375,2.375
SPA/-2.375,-2.375
SPA/2.375,2.375
SPA/2.375,-2.375
TURRET/1000,4,4,1,1,90,200,1,-99,99,-99,99 FRONTTURRET
SAVE/
KCU/2000
SPA/2.875,2.875
SPA/-2.875,2.875
SPA/-2.875,-2.875
SPA/2.875,-2.875
SPA/2.875,2.875
TURRET/2000,4,4,1,1,90,200,2,-99,99,-99,99 REARTURRET
SAVE/
```



Computer definition of the lathe's front turret

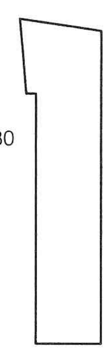


Computer definition of lathe's rear turret

Tool Library Section

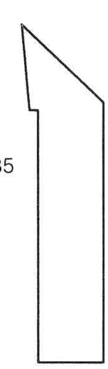
```
KCU/180
SPA/0.09375,-1.116873
SPA/0,-0.03125,-0.466308
SPA/0.03125,0
SPA/1.25,-0.106234
SPA/1.25,-5.996873
SPA/0.25,-5.996873
SPA/0.25,-1.116873
SPA/0.09375,-1.116873
TOOL/180,1,0.031,0.031,-0.031,-2.875,4.375,0.01,100
  T80DEGTURNINGTOOL
SAVE/
KCU/135
SPA/0.051337,-0.007311
SPA/1.25,-1.007691
SPA/1.25,-5.958817
SPA/0.25,-5.958817
SPA/0.25,-1.208817
SPA/0.102904,-1.208817
SPA/0,-0.033974,-0.668179
SPA/0.051337,-0.007311
TOOL/135,1,0.031,0.031,-0.031,-2.875,4.375,0.01,100
  T35DEGTURNINGTOOL
SAVE/
```

K180



Computer definition of 80° right hand turning tool

K135

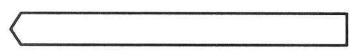


Computer definition of 35° right hand turning tool

Figure 10.62 *(Continued)*

Tool Library Section (*Continued*)

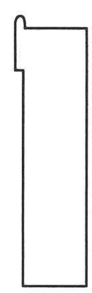
```
KCU/125
SPA/6,0.25
SPA/0.168627,0.25
SPA/0,0
SPA/0.168627,-0.25
SPA/6,-0.25
SPA/6,0.25
TOOL/125,1,0,0,0,-6.018,-1.875,0.01,100 DRILL-1/2
SAVE/
```



K125

Computer definition of 0.500 in. diameter drill

```
KCU/105
SPA/-0.0625,-0.875
SPA/-0.0625,-0.15
SPA/-0.0625,-0.108253
SPA/0,0
SPA/0.0625,-0.108253
SPA/0.0625,-0.15
SPA/1.1875,-0.15
SPA/1.1875,-5
SPA/0.1875,-5
SPA/0.1875,-0.875
SPA/-0.0625,-0.875
TOOL/105,1,0,0,0,-2.875,4.375,0,100 THREADINGTOOL
SAVE/
```

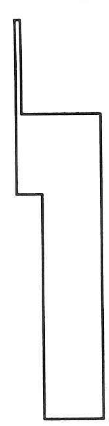


K105

Computer definition of 60° Vee threading tool

Tool Library Section (*Continued*)

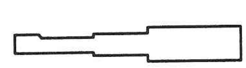
```
KCU/110
SPA/0,-3.05
SPA/0,0
SPA/0.125,0
SPA/0.125,-1.7
SPA/1.5185,-1.7
SPA/1.5185,-6.95
SPA/0.5185,-6.95
SPA/0.5185,-3.05
SPA/0,-3.05
TOOL/110,1,0,0,0,-3.149,5.571,0.005,100 PARTINGTOOL
SAVE/
```



K110

Computer definition of parting tool

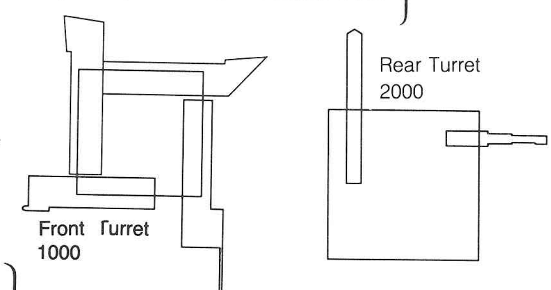
```
KCU/112
SPA/2.25,-0.525
SPA/2.25,-0.461978,0.409078
SPA/2.220527,-0.431982
SPA/0.4,-0.4
SPA/0,-0.4
SPA/0,0
SPA/0.4,-0.020963
SPA/0.4,-0.023969,0.42414
SPA/0.431013,-0.053952
SPA/2.221013,0.006521,0.404357
SPA/2.25,0.036504
SPA/2.25,0.1
```



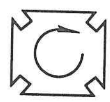
K112

Computer definition of boring bar

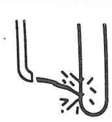
Figure 10.62 *(Continued)*

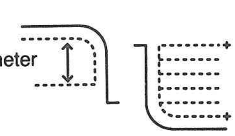


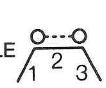
SPA/3.75,0.1	
SPA/3.75,-9.525	
SPA/2.25,-0.525	
TOOL/112,1,0,0,0,-5.278,-1.663,0.01,100 BORINGBAR.5	
SAVE/	
SLIDE/1000,2000,0,15.25	Computer commands to place needed tools in library into proper machine turret positions
STATION/1000,1,180	
STATION/1000,2,135	
STATION/1000,3,110	
STATION/1000,4,105	
STATION/2000,1,125	
STATION/2000,2,112	
KCU/99	Computer definition of material stock curves
SPA/0.06,0,0	
SPA/0.06,-1.003005,0	
SPA/-3.095137,-1.003005,0	
Start Facing Operation — NEWSEQ/10	
!1 INDEX/2000,1,0	Index rear turret to clearance
!2 FROM/2.06,4.5	Enter setup position of tools to part
!3 SPIN/1000,0	Set spindle speed
!4 FROM/2.06,4.5	
!5 FPM/200	Set rapid feedrate
!6 FPR/0.01	Set cutting feedrate
!7 SPIN/1000,0	
!8 INDEX/2000,4,0	Index rear turret to position
!9 INDEX/1000,1,1	Index front turret to position
!10 RAPID/	Set rapid motion
!11 GOTO/0.03,-1.112	*Move to clearance position to start cut (Fig. 10.52)*



!12 COOL/3	Turn coolant on
!13 GOTO/0.03,0.03	Face part (Fig. 10.52)
!14 RAPID/	Set rapid motion
!15 GODEL/0.1,-0.1	Move clear of part
!16 RAPID/	Set rapid motion
!17 GOTO/0.1,-1.05	Movements to set up tool for part roughing cuts
!18 GOTO/-3.625,-1.05	
!19 GODEL/0,-0.1	
!20 RAPID/	
!21 GOTO/0.1,-1.05	
!22 STEP/0.075,90,180,-45,1,1,-1	Set parameters for rough cutting part
!23 GOCLR/3,0.05,0.1,-1,0.10635,-0.56388	Go clear command to generate roughing cuts (Fig. 10.53)
!24 RAPID/	Set rapid motion
!25 GOZX/0.06096,-0.41762,-1	Place tool for semifinish cut
!26 PROF/3,0.02,0.05591,-0.40753,-0.92712,-0.52	Semifinish profile part (Fig. 10.54)
!27 PROF/3,0.02,-1.23,-0.52,-3.457,-1.02	
!28 PROF/3,0.02,-3.48,-1.02,-3.61068,-1.05309	
!29 GODEL/0,-0.1	Face cut clear of part
!30 RAPID/	Set rapid movement
!31 GODEL/0.1,-0.1	Move clear of part
!32 COOL/2	Turn coolant off
!33 RAPID/	Set rapid movement
!34 GOHOME/	Return turret to tool change position







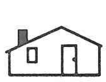


Figure 10.62 *(Continued)*

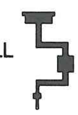
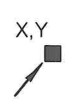
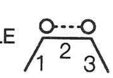
Operation	Program	Description
Drilling Operation (Fig. 55)	FILE/	Create file of machining pathes
	__M/C TIME 2.64	Machine time is 2.64 minutes
	—NEWSEQ/20	
	!1 INDEX/2000,1,5	Index rear turret
	!2 FROM/2.06,4.5	Enter set-up position of tool to part
	!3 SPIN/1000,0	Enter machine spindle speed
	!4 FROM/2.06,4.5	
	!5 FPM/200	Set rapid feed rate
	!6 FPR/0.01	Set cutting feed rate
	!7 SPIN/500,0	Enter new spindle speed
	!8 RAPID/	Set rapid movement
	!9 COOL/3	Turn coolant on
	!10 GOTO/0.1,0	Rapid tool to end of part
	!11 DRILL/0.5,0.1,-1.022,2	DRILL Icon  Drill part command (Fig. 10.55)
	!12 COOL/2	Turn coolant off
	!13 RAPID/	Set rapid movement
	!14 GODEL/0.1	Clear tool from part
	!15 RAPID/	Set rapid movement
	!16 GOHOME/	Return turret to tool change position
Grooving Operation	FILE/	Create file of drilling tool pathes
	__M/C TIME 0.29	Machine time for drilling operation
	—NEWSEQ/30	
	!1 INDEX/2000,1,0	Enter set up tool
	!2 FROM/2.06,4.5	Enter set position of tool to part
	!3 SPIN/1000,0 }	
	!4 FROM/2.06,4.5 }	Set up information
	!5 FPM/200 }	
(Fig. 56)	!6 INDEX/2000,3,0	Index rear turret to clearance position
	!7 INDEX/1000,3,3	Index front turret to parting tool
	!8 SPIN/300,0	Enter proper cutting rpm
	!9 RAPID/	Set rapid movement
	!10 GOTO/-1.25,-0.55	Rapid to start grooving operation
	!11 COOL/3	Turn coolant on
	!12 GOTO/-1.25,-0.375 }	
	!13 DWELL/2 }	Dwell at end of cut
	!14 GODEL/0,-0.15 }	Move to clear part
	!15 RAPID/ }	Set rapid movement
	!16 GOTO/-1.125,-0.55 }	Rapid to next groove point
	!17 GOTO/-1.125,-0.375 }	Cut groove area to depth (Fig. 10.56) GOTO XZ Position Icon
	!18 DWELL/2 }	Dwell at end of cut
	!19 GODEL/0,-0.15 }	Move to clear part
	!20 RAPID/ }	Set rapid movement
	!21 GOTO/-0.975,-0.525 }	GO TO XZ Position Icon  Rapid to next groove point
	!22 GOTO/-1.125,-0.375 }	
	!23 DWELL/2	Dwell at end of cut
	!24 GODEL/0,-0.15	Move to clear part
	!25 RAPID/	Set rapid movement
	!26 GODEL/0.1,-0.1	Rapid clear of part
	!27 RAPID/	Set rapid movement
	!28 COOL/2	Turn coolant off
	!29 GOHOME/	Return turret to tool change position

Figure 10.62 *(Continued)*

FILE/	Create file of machining pathes
__M/C TIME 0.17	Machine time for grooving is 0.17 minute

Finish Part Profile

—NEWSEQ/40	
!1 INDEX/2000,1,0	Setup information (!1–!6)
!2 FROM/2.06,4.5	
!3 SPIN/1000,0	
!4 FROM/2.06,4.5	
!5 FPM/200	
!6 FPR/0.01	
!7 INDEX/2000,3,0	Index rear turret to clearance position
!8 INDEX/1000,2,2	Index front turret to 35° finish turning tool
!9 RAPID/	Set rapid movement
!10 GOTO/0,-0.55	Position tool for finish face
!11 COOL/3	Turn coolant on
!12 GOTO/0,-0.22	Finish face part
!13 RAPID/	Set rapid movement
!14 GODEL/0.1,-0.1	Retract clear of part to prepare for finish profile
!15 PROF/3,0,2,0,0,4,0,0	PROFILE Icon  Finish turn outside profile of part (Fig. 10.57) (!15–!17)
!16 PROF/3,0,-1.25,-0.5,-3.437,-1	
!17 PROF/3,0,17,0,0,17,0,0.1	
!18 GODEL/0,-0.1	Face cut off part surface
!19 RAPID/	Set rapid movement
!20 COOL/2	Turn coolant off
!21 GODEL/0.1,-0.1	*Retract from part*
!22 RAPID/	Set rapid movement
!23 GOHOME/	Return turret to tool change position
FILE/	Create file of machining pathes
__M/C TIME 0.51	Machine time for finishing the outside profile is 0.51 minute

Rough and Finish Bore

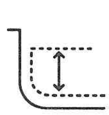
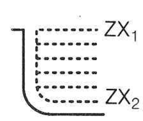
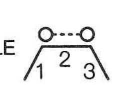
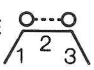
—NEWSEQ/50	
!1 INDEX/2000,1,0	Set up information (!1–!6)
!2 FROM/2.06,4.5	
!3 SPIN/1000,0	
!4 FROM/2.06,4.5	
!5 FPM/200	
!6 FPR/0.01	
!7 INDEX/2000,2,6	Index boring bar into position
!8 RAPID/	Set rapid movement
!9 GOTO/0.1,0.24	Place boring tool in front of drilled hole
!10 COOL/3	GOCLEAR SET-UP Parameter Icon  Turn coolant on
!11 STEP/0.02,90,180,-45,1,1,-1	Set roughing tool path parameters
!12 GOCLR/1,0.01,0.1,0.24,0.09626,0.38934	 GOCLEAR Icon Roughing tool pathes created (Fig. 10.58)
!13 RAPID/	Set rapid movement
!14 GOTO/0.25,-0.75	Position for finish face
!15 PROF/1,0,-0.75104,0.2229, -0.75,0.31574	PROFILE Icon  Finish face c'bore
!16 GODEL/0.05,-0.05	Retract from counterbore face
!17 RAPID/	Set rapid movement
!18 GOZX/0.05704,0.31949,1	Retract from bore

Figure 10.62 *(Continued)*

Statement	Description
!19 RAPID/	Set rapid movement
!20 GOZX/0.05591,0.40951,-1	Position tool to finish profile bore
!21 PROF/1,0,0.00043,0.40951,-0.75,031574	Finish profile bore (see Fig. 10.59)
!22 GODEL/0.05,-0.05	Move clear of side of bore
!23 RAPID/	Set rapid movement
!24 GOZX/0.24252,0.21574,-1	Retract from bore
!25 COOL/2	Turn coolant off
!26 RAPID/	Set rapid movement
!27 GOHOME/	Return turret to tool change position
FILE/	Create file of machining pathes
__M/C TIME 0.67	Machine time for roughing and finishing bore is 0.67 minute

PROFILE Icon 

Threading Operation

Statement	Description
—NEWSEQ/60	
!1 INDEX/2000,1,0	Set up information (!1–!6)
!2 FROM/2.06,4.5	
!3 SPIN/1000,0	
!4 FROM/2.06,4.5	
!5 FPM/200	
!6 FPR/0.01	
!7 INDEX/2000,3,0	Index rear turret to clearance position
!8 INDEX/1000,4,4	Index front turret to *threading tool*
!9 SPIN/300,0	Set spindle speed
!10 COOL/3	Turn coolant on
!11 OPSKIP/	Select slash delete option on
!12 THREAD/20,1,0.1,-0.5,-1.07, -0.5,0.035,30,3,2,1,0.1	Threading tool pathes "O.D." (See Fig. 10.60)
!13 OPSKIP/	Turn slash delete off
!14 COOL/2	Turn coolant off
!15 RAPID/	Set rapid movement
!16 GODEL/0.1,-0.1	Retract tool clear of part
!17 RAPID/	Set rapid movement
!18 GOHOME/	Return turret to tool change position
FILE/	Crete file of machining pathes
__M/C TIME 0.12	Machine time for producing threads is 0.12 minute

THREADING Icon 

Part Cutoff Operation

Statement	Description
—NEWSEQ/70	
!1 INDEX/2000,1,0	Set up information (!1–!5)
!2 FROM/2.06,4.5	
!3 SPIN/1000,0	
!4 FROM/2.06,4.5	
!5 FPM/200	
!6 INDEX/2000,3,0	Index rear turret to clearance position
!7 OPSKIP/	Turn slash delete option on
!8 INDEX/1000,3,3	Index front turret to part off tool
!9 SPIN/300,0	Set spindle speed
!10 RAPID/	Set rapid movement

Figure 10.62 *(Continued)*

!11	GOTO/-3.5,-1.1	Position tool to part (see Fig. 10.61)
!12	COOL/3	Turn coolant on
!13	GOTO/-3.5,0	Cut part off (GOTO Icon X,Y)
!14	RAPID/	Set rapid movement
!15	GOTO/-3.5,-1.2	Retract from part
!16	RAPID/	Set rapid movement
!17	COOL/2	Turn coolant off
!18	GOHOME/	Return turret to tool change position
!19	OPSKIP/	Turn slash delete option off
FILE/		Create file of machining pathes
__ M/C TIME 0.1		Machine time for cutting part off
EOF/EOF/		End of file

Figure 10.62 *(Concluded)*

The machining path commands when finished are placed in one master file by filing out of the sequence mode of operation (see Figure 10.63). The programmer now has a file of computer commands that will generate a CNC machine program. The machine program is created by running the computer program through a postprocessor program in the computer. The postprocessor program is a translator that converts the computer program to a CNC program for the particular machine you will be using. There is normally a postprocessor file for each different machine you have. This process is very simply completed by activating first the program to convert and then the postprocessor required, through typed in commands. The output of our example program for the screw jack can be seen in Figure 10.64. It is also indicated on the CNC program what

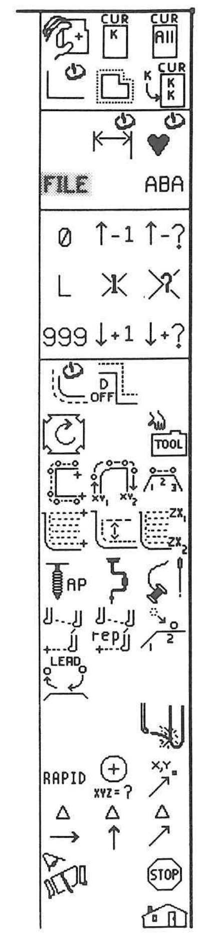


Figure 10.63 *FILE icon command used to exit the tool path generation mode creating a master computer machining file.*

Oct 1 17:35 1990 tc1 Page 1	Date and time of run—File name
###SHEAB_1	Postprocessor name
###DATE 09/20/90 TIME 19:29 HRS	Date and time of computer source file Figure 10.62
### PARTNO/Jack Screw 87231-2	Part identification
CNC Machine Commands	*Computer statement Generating CNC Command*

Sequence #10 Rough—Face & Turn Figures 10.52–10.54

%	Rewind Stop Code
N0010G70	
N0020G90	Setup SEQ. 10—Lines 1–7
N0030G00T00	
N0040T0700	Line 8
N0050T0101	Line 9—80° right-hand turning tool
N0060G95	Lines 1–7
N0070G92X4.5Z5.203	Line 7
N0080G97S1000M03	Lines 10 & 11 — Rough face part
N0090G00X1.112Z.03	Line 12 — Rough face part
N0100M08	Line 13 — Rough face part
N0110G01X-.03F.01	Lines 14 & 15 — Rough face part
N0120G00X.07Z.13	Lines 16 & 17
N0130X1.05Z.1	
N0140G01Z-3.625	
N0150X1.15	
N0160G00X1.05Z.1	
N0170X1.	
N0180X.925	
N0190G01Z-2.8354	
N0200X.9588Z-2.8824	
N0210X.9838Z-2.9157	
N0220X1.Z-2.9394	
N0230G00Z.1	
N0240X.85	
N0250G01Z-2.7137	
N0260G03X.925Z-2.8354I1.244K.6827	
N0270G00Z.1011	
N0280X.775	
N0290G01Z-2.5543	
N0300G03X.85Z-2.7137I1.319K.5233	
N0310G00Z.1022	
N0320X.7	
N0330G01Z-2.2962	
N0340G03X.775Z-2.5543I1.394K.2652	Lines 18–25—Rough turn part (N0140–N0370)
N0350G00Z.1033	
N0360X.6319	
N0370G01Z-1.53	

Figure 10.64 *Postprocessor output of CNC data for jack screw example to run on a Sheldon lathe.*

N0380G02X.675Z-1.656I.1629K.126	
N0390G01Z-2.031	
N0400G03X.7Z-2.2962I1.419K0.	
N0410G00Z.1044	
N0420X.5639	
N0430G01Z-1.4732	
N0440G02X.6319Z-1.53I.0949K.1829	
N0450G00Z.1054	
N0460X.4176Z.061	
N0470G01X.4073Z.02	
N0480G02X.4431Z.0051I.0003K.051	
N0490G01X.5051Z-.0569	
N0500G02X.52Z-.093I.0361K.0361	
N0510G01Z-.906	
N0520G02X.5161Z-.9255I.051K0.	
N0530G01X.5051Z-1.2449	
N0540G02X.52Z-1.281I.0361K.0361	
N0550G01Z-1.4876	
N0560G02X.645Z-1.656I.051K.1684	Lines 25–28—Semifinish part profile (N0460–N0650)
N0570G01Z-2.031	
N0580G03X.9348Z-2.90041I.449K0.	
N0590G01X.9598Z-2.9337	
N0600G02X1.02Z-3.1143I.2408K.1806	
N0610G01Z-3.406	
N0620G02X1.0051Z-3.4421I.051K0.	
N0630G01Z-3.4949	
N0640G02X1.02Z-3.531I.0361K.0361	
N0650G01Z-3.6107	
N0660X1.12	Line 29—Face clear of part
N0670G00X1.22Z-3.5107	Lines 30 & 31—Clear part
N0680M09	Line 32—Turn coolant off
N0690X4.5Z5.203T0000	Lines 33 & 34—Send turret to tool change

Sequence #20 Drill Operation Figure 10.55

N0700T0505	SEQ. 20—Line 1—Index rear turret to .500 drill
N0710G95	
N0720G92X-4.5Z2.06	Lines 2–7—Setup information
N0730G97S500M03	
N0740G00X0.Z.1M08	Lines 8–10—position drill to part
N0750G01Z-.4905F.01	
N0760G00Z.1	
N00770Z-.461	Line 11—Drill part—0.500 in. dia. hole
N0780G01Z-1.022	
N0790G00Z.1	
N0800M09	Line 12—Turn coolant off
N0810Z.2	Lines 13 & 14—Clear tool from part
N0820X-4.5Z2.06T0000	Lines 15 & 16—Send turret to tool change

Figure 10.64 *(Continued)*

Sequence #30 Grooving Operation Figure 10.56

```
N0830T0303
N0840G95
N0850G92X3.304Z4.929
N0860G97S300M03
N0870G00X.55Z-1.25
N0880M08
N0890G94
N0900G01X.375F200.
N0910G04F2.
N0920X.525
N0930G00X.55Z-1.125
N0940G01X.375
N0950G04F2.
N0960X.525
N0970G00Z-.975
N0980G01X.375Z-1.125
N0990G04F2.
N1000X.525
N1010G00X.625Z-1.025
N1020M09
N1030X3.304Z4.929T0000
```

SEQ. 30—
Line 7—Index front turret to parting tool

Lines 1–8—Setup information

Lines 9 & 10—Position groove
Line 11—Turn coolant on

Line 12—Cut groove
Line 13—Dwell at bottom of groove
Line 14—Clear part

Lines 15–24—Cut remaining groove area

Lines 25 & 26—Clear part

Line 28—Turn coolant off
Lines 27 & 29—Send turret to tool change

Sequence #40 Finish Turning Operation Figure 10.57

```
N1040T0202
N1050G94
N1060G92X4.5Z5.203
N1070G97S1000M03
N1080G00X.55Z0.
N1090M08
N1100G95
N1110G01X.22F.01
N1120G00X.32Z.1
N1130G01X.4289Z-.0091
N1140X.4909Z-.0711
N1150G02X.5Z-.093I.0219K.0219
N1160G01Z-.906
N1170G02X.4909Z-.9279I.031K0.
N1180G01Z-1.2591
N1190G02X.5Z-1.281I.0219K.0219
N1200G01Z-1.5031
N1210G02X.625Z-1.656I.031K.1529
N1220G01Z-2.031
N1230G03X.9188Z-2.9124I1.469K0.
N1240G01X.9438Z-2.9457
N1250G02X1.Z-3.1143I.2248K.1686
```

SEQ. 40—Line 8—Index front turret to 35° finish turning tool

Lines 1–7—Setup information

Lines 9 & 10—Position for finish face

Lines 11 & 12—Finish face part

Lines 13 & 14—Move clear of part

Lines 15–17—
Finish profile part

Figure 10.64 *(Continued)*


```
N1260G01Z-3.406
N1270G02X.9909Z-3.4279I.031K0.
N1280G01X1.Z-3.531
N1290Z-3.631
N1300X1.1
N1310M09
N1320G00X1.2Z-3.531
N1330X4.5Z5.203T0000
```

Line 18—Face off of part
Lines 19–21—
Move clear of part/coolant off

Lines 22 & 23—
Send turret to tool change

Sequence #50 Rough and Finish Bore Figures 10.58 and 10.59

```
N1340T0606
N1350G95
N1360G92X-4.712Z2.8
N1370G97S1000M03
N1380G00X-.24Z.1
N1390M08
N1400X-.26
N1410G01Z-.74F.01
N1420X-.25
N1430G02X-.24Z-.75I0.K.01
N1440G01Z-.848
N1450G00Z.1
N1460X-.28
N1470G01Z-.74
N1480X-.26
N1490G00Z.0995
N1500X-.3
N1510G01Z-.74
N1520X-.28
N1530G00Z.099
N1540X-.32
N1550G01Z-.5487
N1560X-.3041Z-.74
N1570X-.3
N1580G00Z.0985
N1590X-.34
N1600G01Z-.3087
N1610X-.32Z-.5487
N1620G00Z.098
N1630X-.36
N1640G01Z-.0687
N1650X-.34Z-.3087
N1660G00Z.0975
N1670X-.3747
N1680G01Z-.0339
N1690X-.362Z-.0557
```

SEQ. 50—Line 7—
Index rear turret to boring bar

Lines 1–6—Setup information

Lines 8–9—Position to drilled hole
Line 10—Coolant on

Lines 11 & 12—
Rough Bore Figure 10.58

Figure 10.64 *(Continued)*

```
N1700X-.3607Z-.0599
N1710X-.36Z-.0687
N1720G00Z.097
N1730X-.3893
N1740G01Z-.0084
N1750X-.3747Z-.0339
N1760G00Z.0966
```

Lines 14–18—Finish face counterbore

```
N1770X-.25Z-.751
N1780G01Z-.75
N1790X-.3133
N1800Z-.7498
N1810X-.2633Z-.6998
N1820G00X-.3195Z.057
N1830X-.4095Z.0559
N1840G01X-.4058Z0.
N1850X-.3707Z-.0607
N1860X-.3133Z-.7498
N1870X-.2633Z-.6998
```

Lines 19–22—Finish bore Figure 10.59

```
N1880G00X-.2157Z.2425
N1890M09
N1900X-4.712Z2.8T0000
```

Lines 23 & 24—Retract from bore
Line 25—Coolant off
Lines 26 & 27—Return turret to tool change position

Sequence #60 Threading Operation Figure 10.60

```
N1910T0404
```

SEQ. 60—Line 8—Index front turret to threading tool

```
/N1920G95
/N1930G92X4.5Z5.203
/N1940G97S300M03
```

Lines 1–9—Setup information

```
/N1950G00X4.5Z.1M08
/N1960X.6
/N1970X.4907Z.0369
/N1980G33Z-1.07K0.0417
/N1990G00X.6
/N2000Z.1
```

One thread pass

```
/N2010X.4813Z.0315
/N2020G33Z-1.07K0.0417
/N2030G00X.6
/N2040Z.1
/N2050X.472Z.0261
/N2060G33Z-1.07K0.0417
/N2070G00X.6
/N2080Z.1
/N2090X.4685Z.0241
/N2100G33Z-1.07K0.0417
/N2110G00X.6
/N2120Z.1
/N2130X.465Z.0221
/N2140G33Z-1.07K0.0417
/N2150G00X.6
/N2160Z.1
```

Lines 10–13—Threading operation with operation skip option

Figure 10.64 *(Continued)*


```
/N2170X.465Z.0221
/N2180G33Z-1.07K0.0417
/N2190G00X.6
/N2200Z.1
N2210M09
N2220G00X.7Z.2
N2230X4.5Z5.203T0000
```

Line 14—Coolant off
Lines 15 & 16—Move clear of part
Lines 17 & 18—Return turret to tool change position

Sequence #70 Cutoff Operation Figure 10.61

```
/N2240T0303
```

SEQ. 70–Line 6—Index front turret to parting tool

```
/N2250G95
/N2260G92X3.304Z4.929
/N2270G97S300M03
```

Lines 1–9—Setup information

```
/N2280G00X1.1Z-3.5
/N2290M08
```

Lines 10 & 11—Position tool to part
Line 12—Turn coolant on

```
/N2300G94
/N2310G01X0.F200.
/N2320G00X1.2
/N2330M09
/N2340X3.304Z4.929T0000
N2350M02
%
```

Line 13—Cut part off
Lines 14 & 15—Retract clear of part
Lines 16–19—Turn coolant off return turret to tool change position
End of program

Figure 10.64 *(Concluded)*

computer statements generated what machine statements. By using a few more commands to the CAD/CAM system, we could transfer the information to the machine tool by electronic means or punch a tape and take it to the machine control.

You have now seen the basic powers of a CAD/CAM system, but they are far beyond what you have seen. Systems can have the cpability of creating complex three-dimensional programs for three- to five-axes CNC machining. Three-dimensional programming is common in both the toolmaking (forming die and mold) and aircraft manufacturing industries. In these industries the three-dimensional shapes must be cut from solid blocks of material.

Figure 10.65 shows an example of a five-axes machining problem where a tapered boss intersects a bullet nose shaped surface. The programmers objective is to machine the side of the boss and the intersection of the boss and bullet nose at the same time. To do this the cutter axis must be tilted while moving the machine in the X, Y, and Z axes. Many machine tool statements are required to perform this operation but only a few computer statements need to be created. Now you can see the power and importance of a CAD/CAM system's geometry and tool path creation capability to industry. We are now able to make things never thought possible a few years ago.

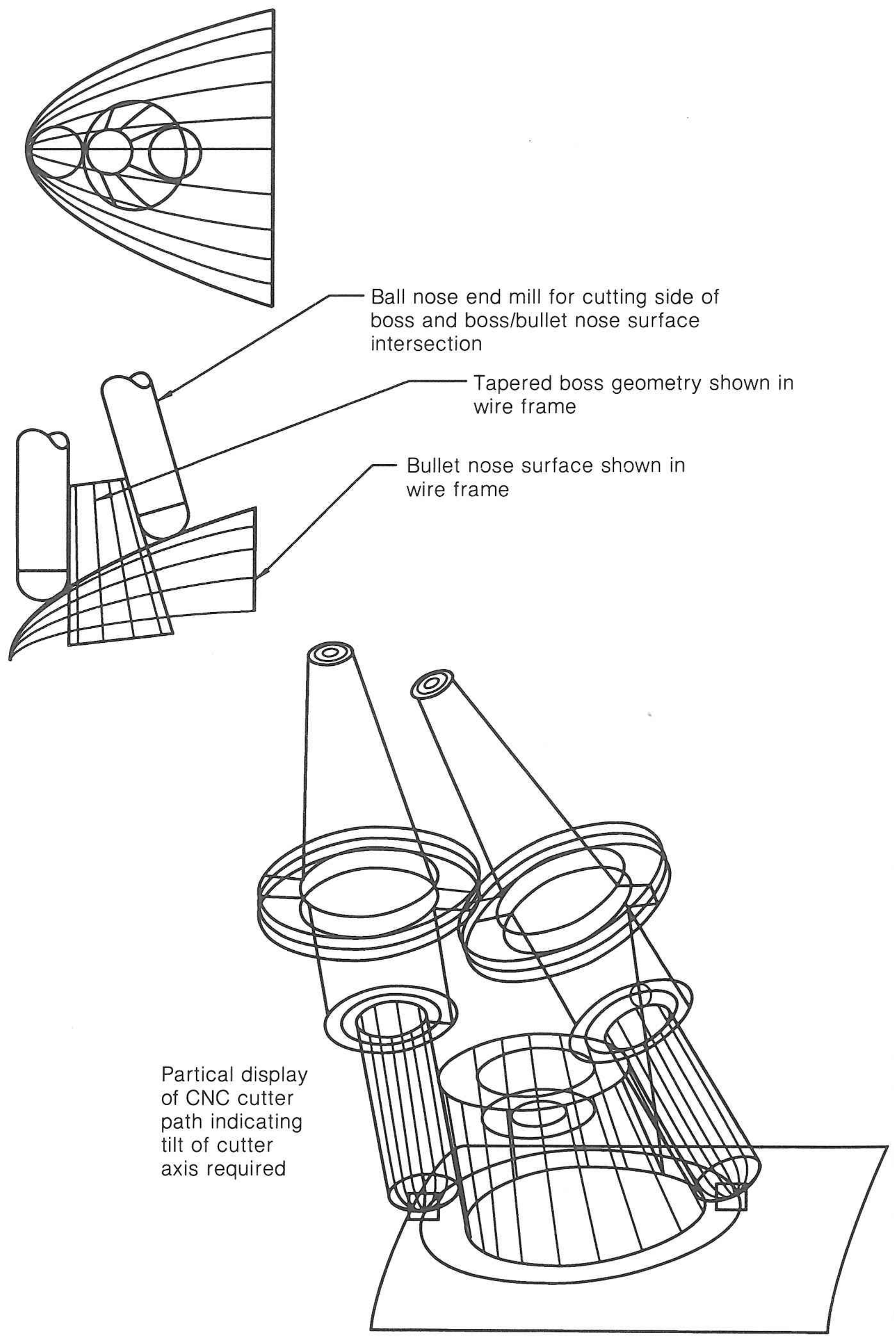


Figure 10.65 *Example of five-axes tool path display on three-dimensional operation.*

QUESTIONS

1 List the advantages of computer aided part programming when compared with manual part programming.

2 List and briefly describe the main stages in the CAPP process.

3 Make a block diagram to illustrate a basic 'stand-alone' CAPP system.

4 Explain what is meant by 'time-sharing' and list the advantages and disadvantages of using such a facility.

5 Describe three methods that may be used for menu selection from a digitizing tablet.

6 Describe three methods of cursor control that may be used to identify graphic features on a CRT screen.

7 Describe three general methods for geometrically defining a point, a straight line and a circle or curve.

8 List the various data that would be included as tool change data statements during the preparation of a part program.

9 Explain what is meant by "cutter location data."

10 Explain the function of "postprocessing" and state how a range of machines having a variety of control systems would be accommodated.

11 Describe two ways in which computer graphics are used as an aid to part programming.

12 Describe briefly the meaning of CAD/CAM.

11

ADVANCED TECHNIQUES

The power of a computer's calculation ability continues to grow at a constant rate. With the computer's growth comes added programming techniques for CNC equipment. This chapter will give an overview of some of the additional programming abilities that exist today. One needs to follow the computer industry news in order to keep abreast of new developments and stay current in the field of programming.

PARAMETRIC PROGRAMMING

A parameter is a quantity that is constant in one particular case but variable in others. A simple engineering example of a parameter is the length of a bolt. One version of the bolt will have a certain length; all other versions will be identical, that is, they will have the same thread form, diameter, and hexagon head, but they will all vary in length. Thus the length of the bolt is a parameter, constant in one particular case but variable in others.

Parametric programming involves defining parameters and then using those parameters as the basis for one part program that may be used to machine not only the original component but a number of variations as well.

Figure 11.1(a) shows a component the dimensional features of which have been defined as parameters using the symbol # and a number: #1, #2, #3, and so on.

Figures 11.1(b)–11.1(g) show six variations of the component, the variations being indicated. A range of components such as this is referred to as a family of parts.

The machine movements necessary to machine each of the variations are all included in the original component. Some components require exactly the same movements, but with varying lengths of travel. Other components do not require all of the movements to be made. Using the more usual programming techniques, the production of each component would require a separate part program. Using the parametric part programming technique, instead of defining each dimensional movement individually in the *X* and *Z* axes, the parametric reference is programmed. Thus, to turn along the stepped diameter, the entry in the main program, referred to as the "macro," would read as follows:

N07 G01 X #4 Z #2

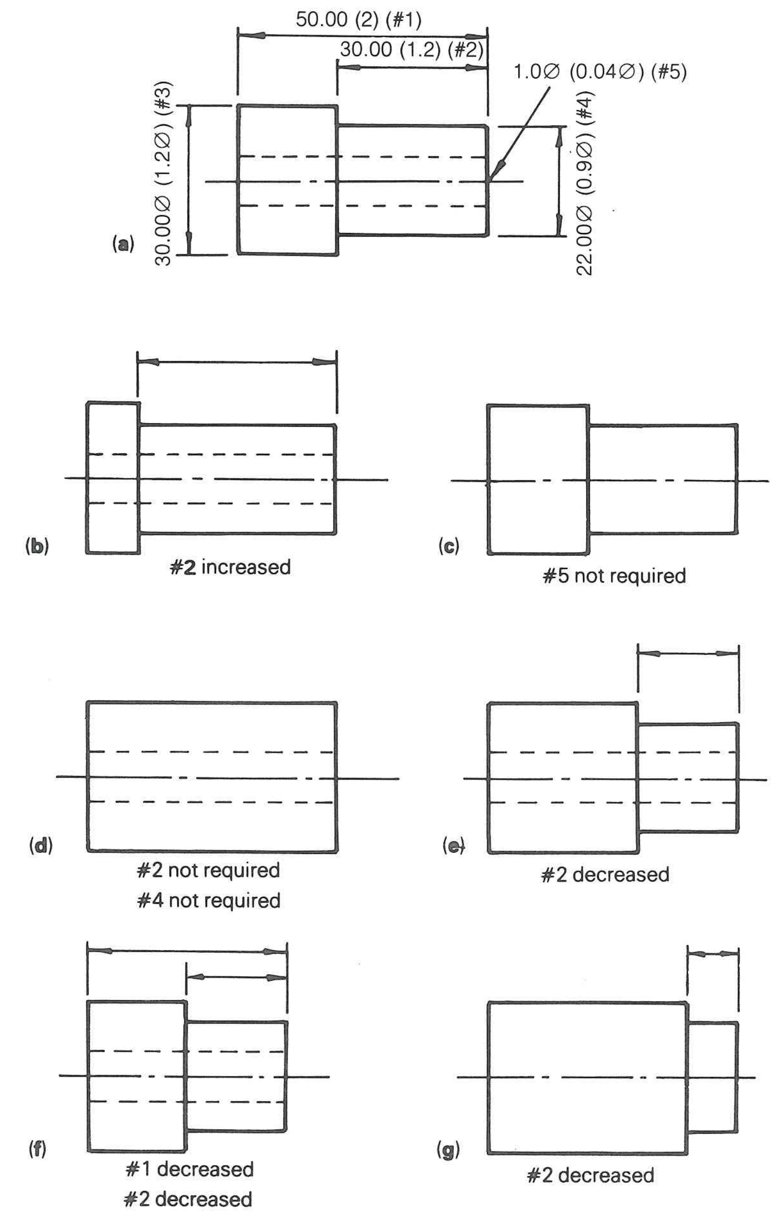


Figure 11.1 *A "family of parts." (Inch units are given in parentheses.)*

This entry would suffice for all components requiring a stepped diameter. Equally, one entry using parametric identification would suffice for facing all the components to length or drilling the hole.

Having programmed all movements and the sequence in which they are to occur, it remains to dimensionally define them. The dimensional details are entered as a list at the start of the part program. Thus the parameters and their dimensional values for the original components would read as follows (in metric units):

#1 = −50.00
#2 = −30.00
#3 = 30.00
#4 = 22.00
#5 = 10.00

As each parameter is called in the program, the dimensional entry made previously will be invoked.

To machine any of the variations in the family of parts requires a simple amendment of the original parametric values. The parameters (in metric units) to machine the component shown in Figure 11.1(b) would be:

#1 = −50.00
#2 = −40.00 (amended)
#3 = 30.00
#4 = 22.00
#5 = 10.00

and to machine the component in Figure 11.1(f):

#1 = −40.00 (amended)
#1 = −20.00 (amended)
#3 = 30.00
#4 = 22.00
#5 = 10.00

Now consider the components where the programmed movements necessary for machining the basic component are not required. By using a relatively simple programming technique, the control unit can be caused to skip the redundant blocks. The necessary program entry involves the use of certain conditional expressions in which assigned abbreviations are used, such as the following:

EQ = equal to
NE = not equal to
GT = greater than
LT = less than
GE = greater than or equal to
LE = less than or equal to

Consider Figure 11.1(d) and assume the #1 and #3 have been machined. In the program the next call will be to machine the stepped diameter. To avoid this, blocks must be skipped so an entry in the program will read as follows:

N15 IF [#4 EQ 0] GO TO N18

This statement says that if #4 is zero, move on to block number 18. Since #4 is nonexistent in the component, the parametric value will be entered as zero and, consequently, the control unit will move ahead.

The preceding description of the use of the parametric programming technique is a very simple one. It is in fact a very powerful concept and its full application is quite complex. For instance, parameters may be mathematically related, that is, they may be added together, subtracted from one another, and so on.

In addition, the parametric principle may be extended to include speeds and feeds, when all the likely variations for roughing, finishing, etc. may be given a parametric identity and called into the program as and when required.

Parametric-type programming although not uncommon is not standardized in its methods between machines or programming systems, so vendor manuals should be consulted.

DIGITIZING

Digitizing is the name given to a technique used to obtain numerical data direct from a drawing or model. To obtain numerical data from a drawing, which may or may not be dimensioned, it is placed on a special tablet, or table, and a probe is traced over the drawing outline. This movement is received by a computer and is transformed into digital or dimensional values. Only two-dimensional data can be obtained from a drawing. For three-dimensioinal data a model of the component is required, and a probe, which is electronic in operation, is traced over the surface of the model, this movement being recorded by the computer as before.

Numerical data obtained by digitzing can be used as the basis of a numerical control program. The numerical data entered into a computer, on the other hand, can be used to create the geometric data for a three-dimensional data base. This data base can then be used to create three-dimensional contouring cutter paths, using CAM/surfacing programming. The technique is only suitable for certain types of machining, such as profile milling, but the concept is likely to be developed to cater to a wider range of machine-shop activity.

FLEXIBLE MANUFACTURING SYSTEM

A flexible manufacturing system (FMS) is a computer-controlled machining arrangement which will cater to a variety of continuous metal-cutting operations on a range of components without manual intervention. The objective of such a system is to produce components at the lowest possible cost, and in particular components that are required only in small quantities. Thus a prime requirement of such a system is flexibility, that is, the capacity to switch from one type of component to another, or from one type of machining to another, without interruption in the production process.

Production costs per unit item decrease as the number of components required increases. Large production runs justify extensive capital expenditure on special-purpose machinery that does a particular job very efficiently and quickly. Machines of this type, however, are rarely adaptable to other types of work: they lack flexibility. When flexibility does exist—one skilled worker and one machine, for instance, where single components can be handled in random order—the production rate is slow and therefore costly. Modern flexible manufacturing systems aim to bridge the gap between these two extremes.

Flexible manufacturing systems have been made possible by the fact that modern machine control units can store in the computer memory a number of part programs which can be activated via a master computer program in random order, a system referred to as direct numerical control (DNC). The same master computer is also able to control the supply of workpieces to the machine. The third important factor, tooling, will be controlled by the part program itself, but if a wide range of machining is to be carried out the tooling magazine will be required to accommodate a large number of tools; for milling operations at least 60 and perhaps more than 100 may be necessary. To solve the problems of tool search time and maintenance of large tool conveyors some machine tool builders have made the conveyors themselves changeable.

A flexible manufacturing system will include at least two machines. When just two or three machines are involved, the arrangement is sometimes referred to as a "machining cell." A fully integrated system will include more machines than this and they will vary in type. Figure 11.2 illustrates the principle. Installations of this nature are, of course, very costly, but are becoming commonly used, at present. However, the modular approach to building such a system, that is, starting with two machines and then adding additional machines as and when investment funds are available, would suggest that the concept is set to become a dominant feature of machine-shop engineering.

The automatic supply of work to each machine is an essential feature of any system, large or small. The use of pallets is the most favored method, particularly for machining centers as opposed to turning centers, although they are also used for turning work. When pallets are used for turning work, the final loading of the machine usually involves a robot. Figure 11.3 shows how one robot may be positioned to service two machines.

The way pallets are used for milling operations will vary according to the type of work being handled and the space available. Figure 11.4 illustrates the use of a rack or storage retrival system. Such a system as this is relatively simple and capable of modification and extension as and when required. More

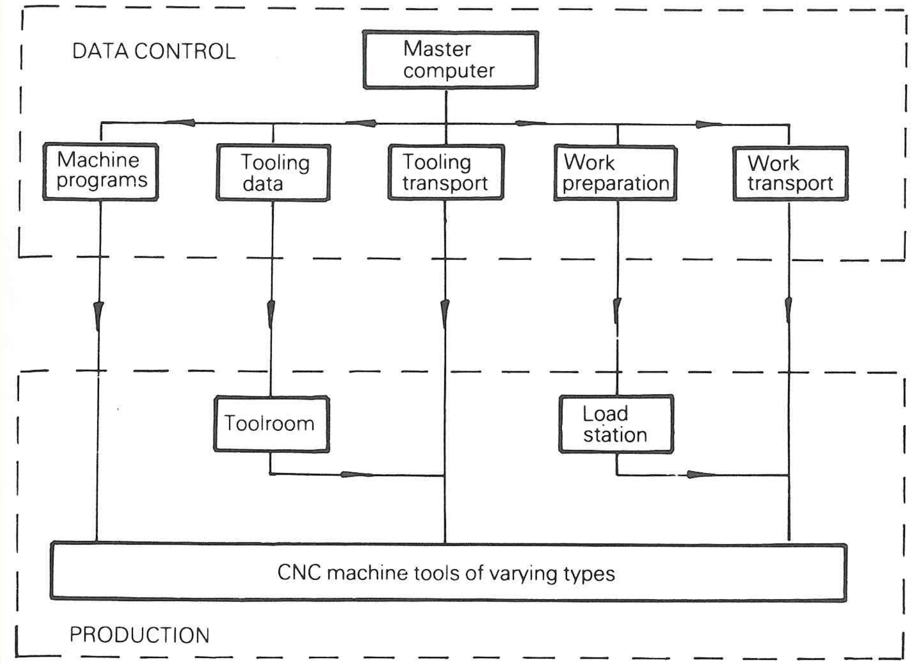


Figure 11.2 *Computerized control of a manufacturing system.*

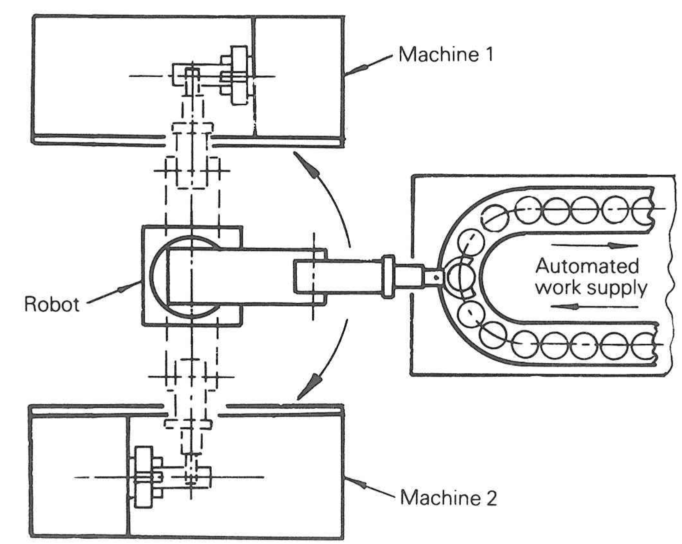


Figure 11.3 *Robot loading of turning centers.*

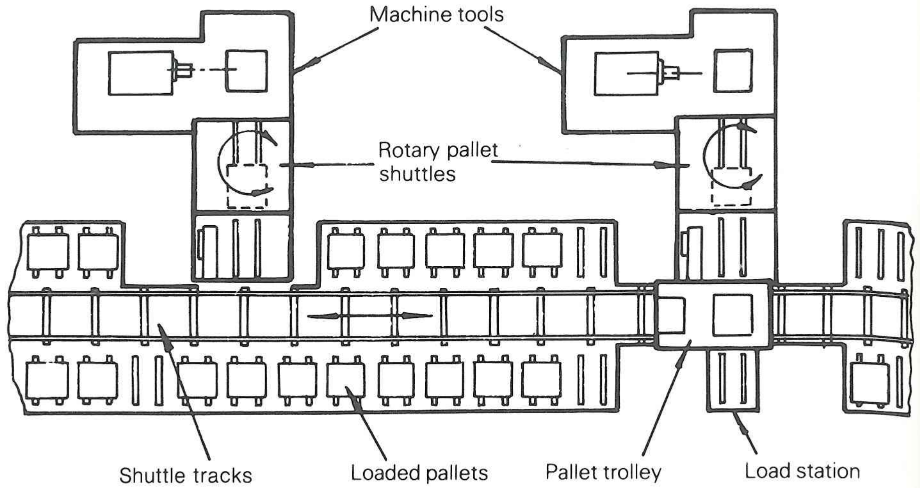


Figure 11.4 *Flexible manufacturing system using racked palletized work supply.*

complex systems involve the use of pallet conveyors and automatic guided vehicles (AGV) moving about the factory along predetermined routes, guided by inductive control wires buried in the workshop floor or computerized programmed controls. Each machine will have its own load/unload station and there will be a master load/unload station to which each pallet or AGV returns at the end of a journey. Figure 11.5 illustrates the principle.

Pallets may be fed to the machines in a predetermined order which means their positioning in the work queue is critical, otherwise a workpiece may be subjected to the wrong machining cycle! More commonly today, though, they are fed to the machine in random order and identified, usually by a photoelectric device responding to a bar code number attached to the pallet, when they arrive at the machine. On being identified the correct machining program for that workpiece will be called.

The preparation of work pallets is generally a manual operation. Their positioning and clamping in the machine are totally automated and, as a result, monitoring of their installation is necessary before machining commences. Limit switches or proximity sensors are a common feature of such control, and one method involving a combination of pneumatic and electrical principles is illustrated in Figure 11.6. Photoelectric cells, mechanical limit switches, and tactical sensors are also commonly used to check pallet and part locations.

Mention was made earlier of the high cost of flexible manufacturing systems which has, to date, limited their introduction. An intermediate approach to total automation is the use of automatic pallet-loaders dedicated to one machine. The pallet pool may involve two or more pallets and be designed in parallel rail, rectangular, oval or round form and when fully loaded will provide for an extended unsupervised production run lasting several hours, perhaps over-

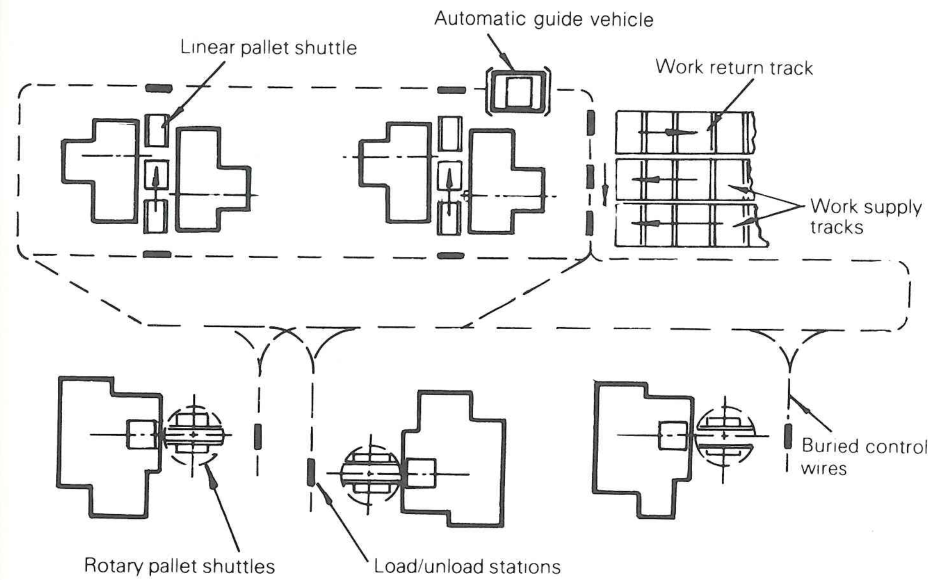


Figure 11.5 *Flexible manufacturing system using remote controlled AGV work supply.*

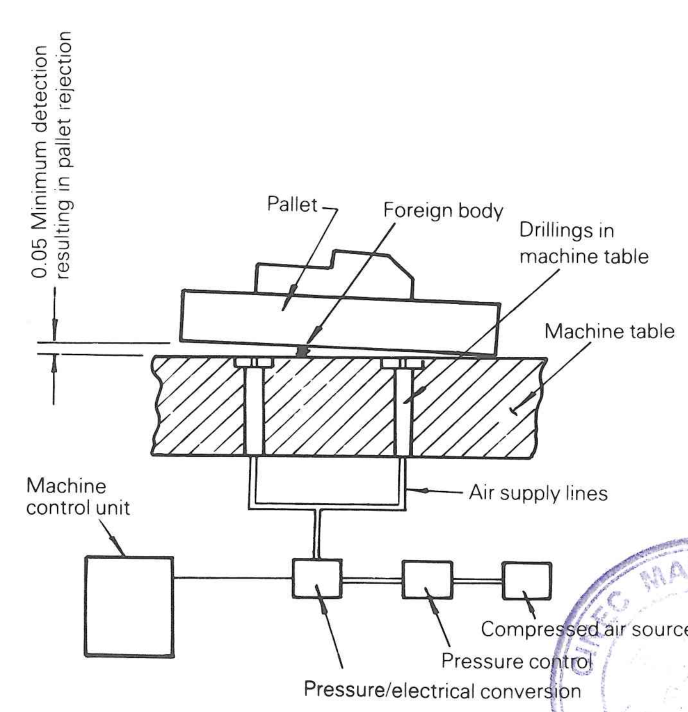


Figure 11.6 *Pallet location control.*

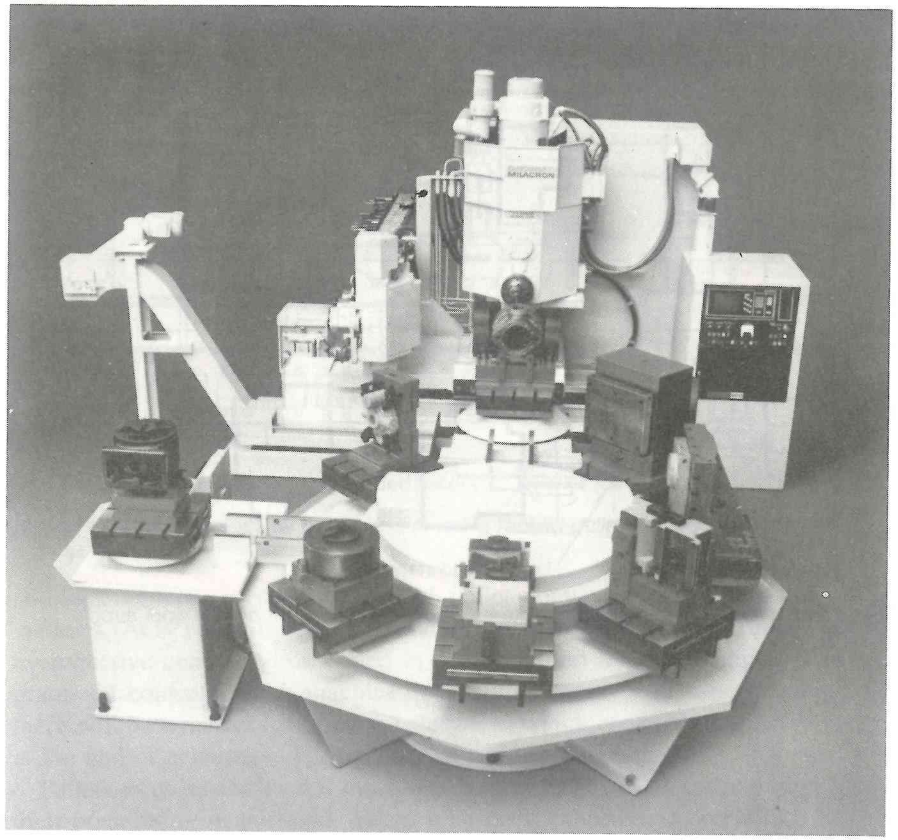


Figure 11.7 *Processing center providing unmanned production runs.*

night or throughout a weekend. Figure 11.7 illustrates such an arrangement, referred to as a "processing center" by the manufacturers; its flexibility is indicated by the range of components shown in position on the pallets. Figure 11.8 shows a robot-loaded turning center where work chutes provide a similar capability. Due to the costly nature of automation/machining cells computer software has been developed to draw up a complete animated system for testing. This software is called simulation software and allows for completely analyzing the system for bottlenecks and problems before the system is engineered and built.

ADAPTIVE CONTROL

Adaptive control is the term used to describe the facility that enables a machine control unit to recognize certain variations from the original conditions which may occur during a machining process and to make a compensating response. Unless such a response is made, the effect of the variations may be to damage

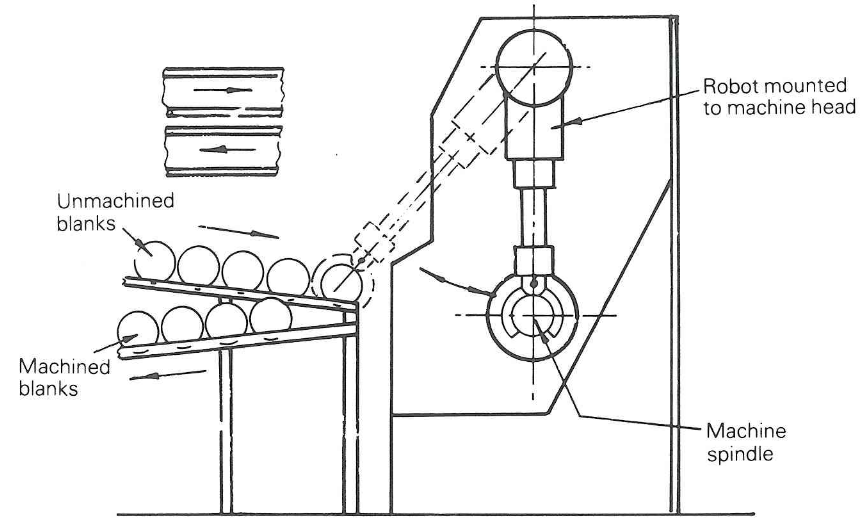


Figure 11.8 *Robot loading facility providing unmanned production runs on a turning center.*

the machine, tooling, workpiece, or cause the part to go out of prescribed tolerance. Adaptive control is basically a data "feedback system" rather like the closed-loop facility described in Chapter 1, which monitored slide positioning.

A number of unacceptable things can occur during a metal-cutting operation. For example, a tool may lose its cutting edge. On manually operated machines this would immediately be obvious to the operator, who would react accordingly. It is this type of response that adaptive control endeavours to emulate.

Now supposing the tool becomes dull on an automatic machining process. What is the likely effect? At least three things are likely to occur. First, the power necessary to turn the machine spindle is likely to increase, that is, there would be a torque variation. Second, there is likely to be a build-up in temperature between the cutting tool and the workpiece, as the tool tends to rub rather than cut. Finally, the tool itself is likely to deflect. By monitoring, that is, measuring, these variations and taking corrective action damage can be averted, and good parts consistently produced.

Torque monitoring of spindle and servo motors is one method of adaptive control that is used. The power consumption is monitored electronically and the application of the technology involves programming the control unit with data that will define the maximum and minimum torque values permitted for any particular operation. Assume that during a metal-cutting sequence the maximum torque value at the cutting tool is reached, indicating perhaps that the tool is dull or the component material is harder than anticipated. The control unit will respond to the feedback signal by lowering the feedrate and/or varying the spindle speed.

Consider another situation where, after modifications to feed and spindle speed, the torque continues to increase to a point where the spindle is overloaded. In this case the control unit would inhibit the sequence and indicate a "warning" signal on the CRT screen. The problem can then be investigated and the conditions rectified.

The torque monitoring feature can also be used to detect the minimum torque that is programmed to occur after a certain length of slide travel. If the programmed torque does not occur, there may be two possible reasons. One is that the cutting tool is broken and has not made contact, a broken drill for example; the other is that the workpiece itself is not in position. Total inhibition of the machining process may be the necessary response, or alternatively a duplicate tool, referred to as a "sister" tool, already in the magazine or turret could be called.

Torque monitoring taken to its extreme means that spindle speeds and feeds can be omitted from part programs. Provided the control unit is programmed with the values of the maximum permitted speeds and feedrates, it is possible for the adaptive control to adjust them to suit the prevailing cutting conditions. For instance, when the torque is high, the feed would reduce, but when it is low, as for example when no cutting is taking place, the feed would be rapid.

Another approach to adaptive control is one which concerns itself with monitoring the presence of workpieces by the use of surface-sensing probes. Such a probe is illustrated in Figure 11.9 and is used in milling operations. The probe is mounted in the tool magazine alongside the cutting tools and can be called into operation via the part program, in the same way as a cutting tool, and be mounted in the machine spindle. The probe is electronic in operation and the stylus is interchangeable to accommodate different applications. It can be pro-

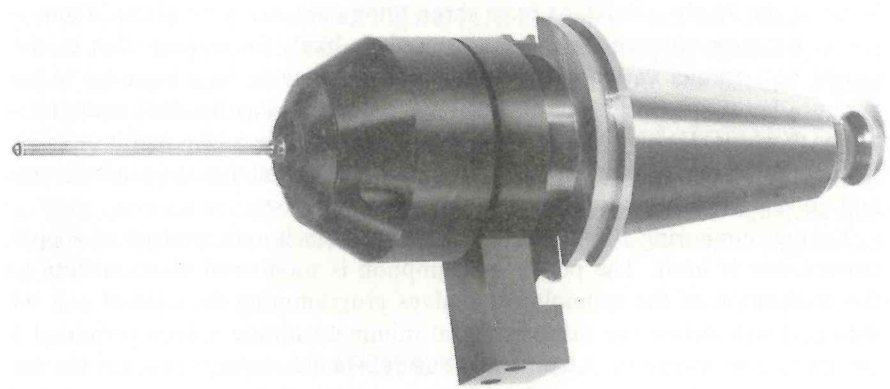


Figure 11.9 *Precision surface sensing probe.*

grammmed to detect the presence, by touching on, of a surface in three axes and, if the surface is not present, it will inhibit the machining cycle.

Probes can also be programmed to check stock size and automatically cause the work datums to be offset to locate the finished part within the bounds of the stock material, thus ensuring that the final work surface is completely machined. If insufficient stock is present, the machining is not performed. This facility is particularly useful when machining castings or forgings.

Another application of a probe is to speed up a cycle by preventing non-metal-cutting passes of the cutting tool. For example, when machining casting or forgings, the part program will need to cater for all possibilities and may well include passing cuts that will be necessary only when there is excessive stock to be removed. If the probe detects that no metal is present, the feedrate will automatically be maximized or a program block may be skipped.

An interesting method of detecting the presence of cutting tools is a device that combines pneumatic or light, and electronic principles. It is designed primarily for use on machining centers to monitor small-diameter drills, taps, and reamers, which are very prone to break. The device is in the form of a simple caliper, which is positioned at a convenient point on the machine bed. The location of the caliper is predetermined and after use the cutting tool is moved via the part program so that it positions within the caliper. When in position, a jet of air is blown or light beam is shown from one side of the caliper and if the cutting tool is missing this jet of air or light beam will blow on to a pressure-sensitive/light-sensitive electrical device housed in the opposite arm of the caliper. This will generate a signal to the machine control unit that will result in either the machining process being halted or the tool being replaced in the program by a sister tool already housed in the magazine. If the tool is present, then there will be no air flow between the two arms of the caliper and the tool will automatically be replaced in the tool magazine to await a further call. Figure 11.10 illustrates the technique.

Another method of detecting broken or dull tooling involves the use of sound sensors. A cutting tool that is cutting properly will emit a certain sound. If the tool loses its edge or breaks, the sound it makes as it attempts to cut metal will be different from the original. The sound sensors detect the variation and will cause the program to be stopped or, alternatively, will call in a sister tool to replace the original.

Adaptive control is an area of computerized numerically controlled machining that is the subject of much research and experiment, and it is an area in which there are liekly to be further very interesting developments.

IN-PROCESS MEASUREMENT

In-process measurement is the term used to describe the automatic measurement or gauging of a component while it is in position on the machine, and

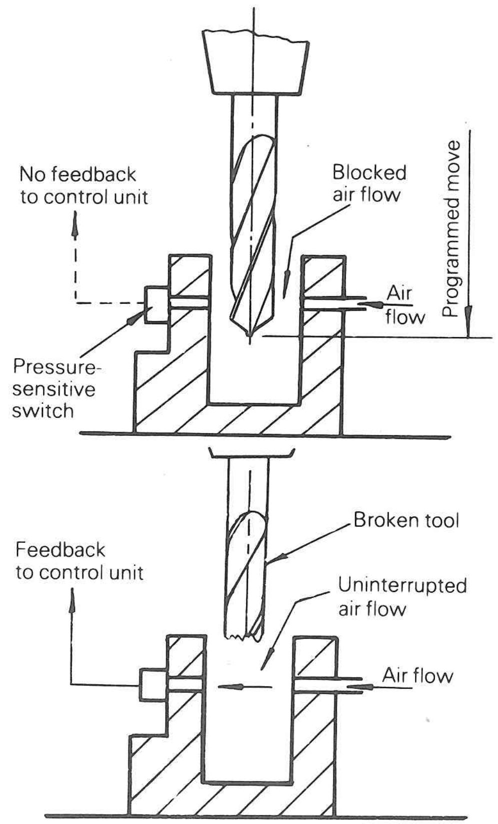


Figure 11.10 *Broken tool detection unit.*

while the correction of errors is still possible. It is not a new concept. The need for automatic measurement went hand in hand with the development of automatic machining processes, and fully automated machines performing a variety of operations were around long before the advent of computerized numerical control.

In-process measurement presents many difficulties. A machining area, with its accumulation of chips and coolant, is not an ideal place to carry out precision measurement involving delicate instruments or monitoring devices. Nevertheless, a number of very successful devices have been developed over the years, their method of operation being based on mechanical, pneumatic, optical, laser, and electronic principles.

Many of the earlier devices, though not all, were "open loop," that is, there was no feedback of data to the machine controls and so there was no automatic adjustment of the machine setting to compensate for unacceptable size variations. Correction was possible only by manual intervention, but at least this was usually possible without halting the machining process.

On modern CNC machines the accuracy with which slide movements are generally made and monitored can, in the case of some classes of work, eliminate the need for further control, since the slide movements, and therefore the relative tool movements, are made to an accuracy which may well be within the dimensional tolerances of the component. In other cases this degree of control is insufficient and, as was stated in Chapter 2, transducers which monitor slide movement or leadscrew rotation may not give an accurate indication of the tool and work relationship. For instance, a tool may wear, thus affecting the dimensional size of the component, but this will have no connection with slide movement and no compensation will be made. Similarly, the workpiece may not be precisely located, or may be impossible to locate precisely, so again some monitoring and correction of movement may be necessary to ensure that surfaces are relatively positioned. These are the sorts of situations that in-process measuring can monitor.

The modern in-process measuring device is electronic in operation. It consists of a probe, laser or vision, and is capable of monitoring positional variations in three axes. The way in which it is applied will depend on the machine type, but it can be applied to the measurement of internal and external diameters, lengths, depths of slots, hole centers, and so on. Programmed air blasts are used to clean the work prior to measurement.

One method of using touch sensors requires reference to be made to an established datum, which may be a surface which is part of the machine structure, for example a tailstock barrel, or a surface on the component. The program will bring the sensor into contact with the reference face and record its position as zero. It will then be moved to the surface being checked and the resulting move will be compared by the machine control unit with a pre-programmed value. If there is a variation, a compensation in the relative tool offset will be made.

Features such as a bored hole diameter can be measured by touching it on each side of the hole, and the resulting movement, plus the stylus diameter of the probe, will indicate the hole size. The necessary calculation will be made by the control unit and again a comparison will be made with a pre-programmed value and tool offsets initiated as required. The technique is illustrated in Figure 11.11.

Measurement of this nature is not completely divorced from the machine slide movement, and its accuracy can never be better than the resolution, that is, the smallest increment that can be determined by the control unit of the machine.

COMPUTER INTEGRATED MANUFACTURING (CIM)

It has been shown that computers are all around us in the CNC manufacturing area: Design "CAD," engineering analysis, CNC programming "CAM," sim-

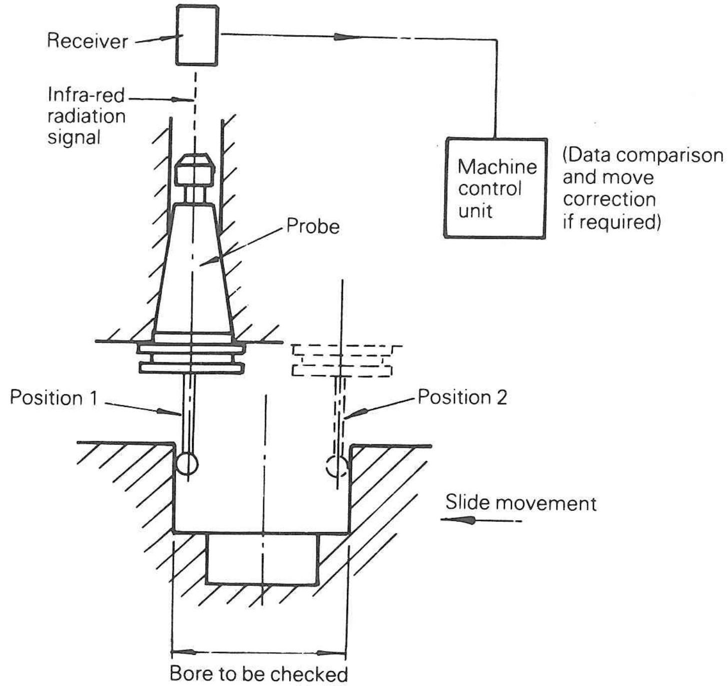


Figure 11.11 *In-process measurement by electronic probe.*

ulation of machining and manufacturing cells, and adaptive controls. We also know that computers are used in the area of business and management for accounting, order processing, forecasting, sales analysis, production schedule planning, inventory management, materials requirements planning, machine capacity requirements planning, production control and costing, purchasing, production monitoring, payroll, financial analysis, general ledger, and accounts payable.

Computer-integrated manufacturing (CIM) is the concept of a totally automated factory in which all business, engineering, and manufacturing processes are linked and controlled by a computer system. See Figure 11.12 for a graphic representation. CIM enables managers, production planners and schedulers, shop floor foremen, and accountants to all use the same database as production designers and engineers. Owing to the short lead times and complexity of manufacturing, all areas of a business must work together to make a profit.

The enterprise model of a computer system, which allows business, engineering, and manufacturing to share information and work together, is shown in Figure 11.13. The inner ring depicts the computer hardware, software, and database. The middle ring indicates the decision making level of management and computer support staff (report generation). The outer ring indicates the users and suppliers of information that must work together. All areas of busi-

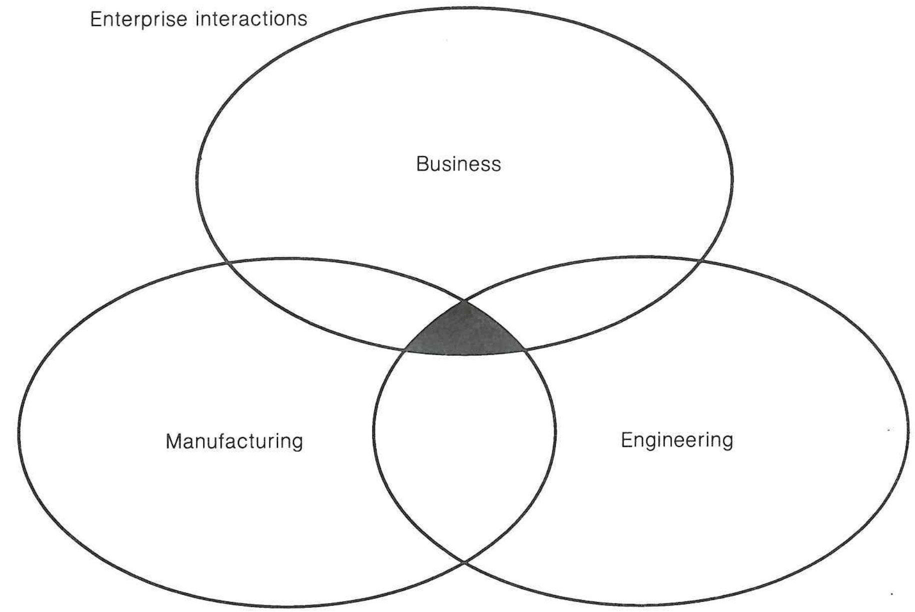


Figure 11.12 *The three areas of a corporation that must work together and share data in order to make a profit*

ness are indicated in the three rings, and information can flow between any of the areas. As in any business venture there should be improvements or goals to achieve. In implementing a CIM plan one would be looking for gains in the following areas:

- A reduction in product lead time.
- Faster, more reliable availability of quality information.
- The ability of process monitoring and tracking.
- Improved product delivery information and quality.
- Improved supplier performance and the ability to track historical performance and make predictions.
- Faster information flow and simultaneous engineering to reduce product cost.
- Improved communication of business plans to employees creating improved attitudes and quality consciousness.

The idea of CIM revolves around the center ring of shared information and people working in teams, simultaneously. The old method of passing information from department to department upon task completion is outdated and slow. To be responsive manufacturers we must keep from duplicating efforts and help each other for the common good of supplying products in a short

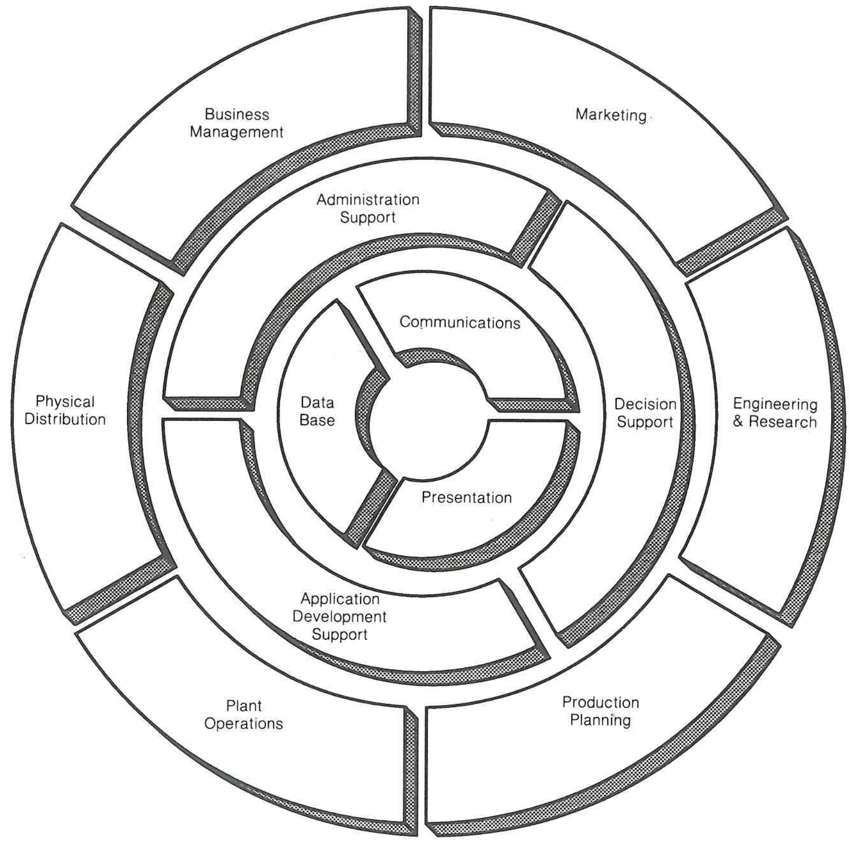


Figure 11.13 *The three rings of computer information. (Courtesy of IBM.)*

period of time. Figure 11.14 shows in simple terms the interaction of information within a business.

In Chapter 10 we discussed the use of a CAD/CAM system for part design and CNC programming. The tasks involved in product design, tool design, and manufacturing engineering are much more involved than that. A product must be conceived first but then it must be analyzed for failure and determined how best to make it. It is here that the product, tool, and manufacturing engineers need to work together and share information to solve all the problems to get a product manufactured. At the same time marketing and sales need to be involved in order to see if the product will sell. As sales are projected, we then need to make plans and order materials to build the product. When orders come in, the product needs to be tracked through its processes so, as delays occur and changes must be made, the right people in engineering, business, and manufacturing are informed. As the end product leaves the manufacturing arena and is distributed to customers, billing must occur. During this whole time

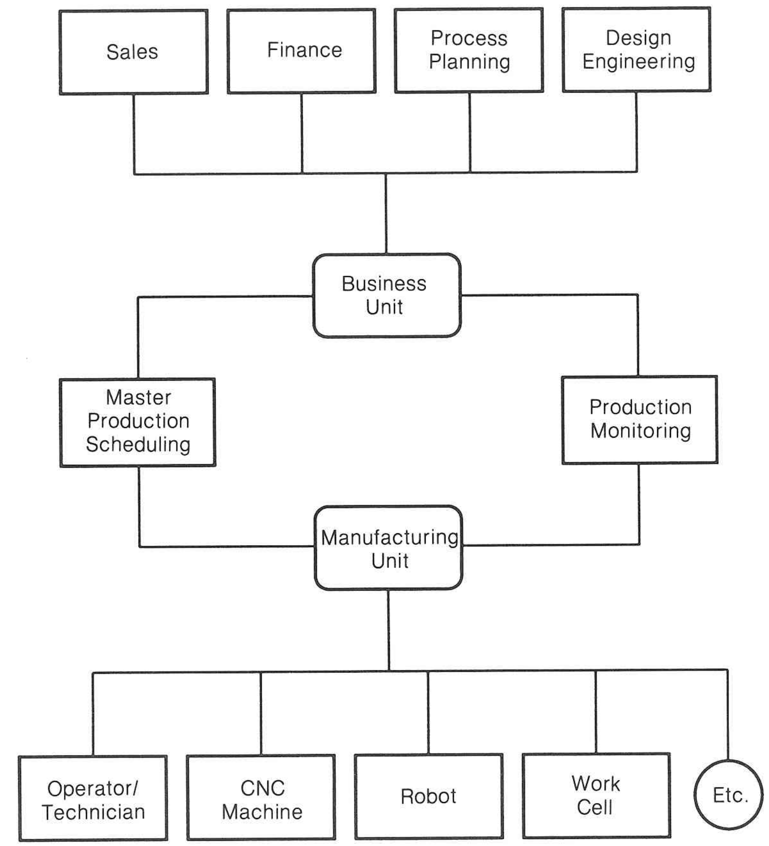


Figure 11.14 *A CIM view: the business, engineering, and manufacturing units.*

accounting and payroll continue. Now you can see the advantage of maintaining one shared computer database for all. Figure 11.15 shows the entire workflow of an enterprise, and it is easy to see how complex the information flow would be using paper instead of computers.

Looking further into business planning (Figure 11.16), we see three levels of management planning that the computer can help with. The top or first level of management deals with the future business plans, sales forecasting, and setting of future production levels. The second level is operations management, which takes care of monthly master schedules for production, makes plans for material needs, and checks capacity requirements (production equipment availability) to the master production schedule. The third level of management is operations execution at the production floor. This final level schedules the production floor day to day in order to supply the proper product at the right time. This person is in charge of initial quality and wants to know about daily deliveries of vendor parts. A common computer database for all these manage-

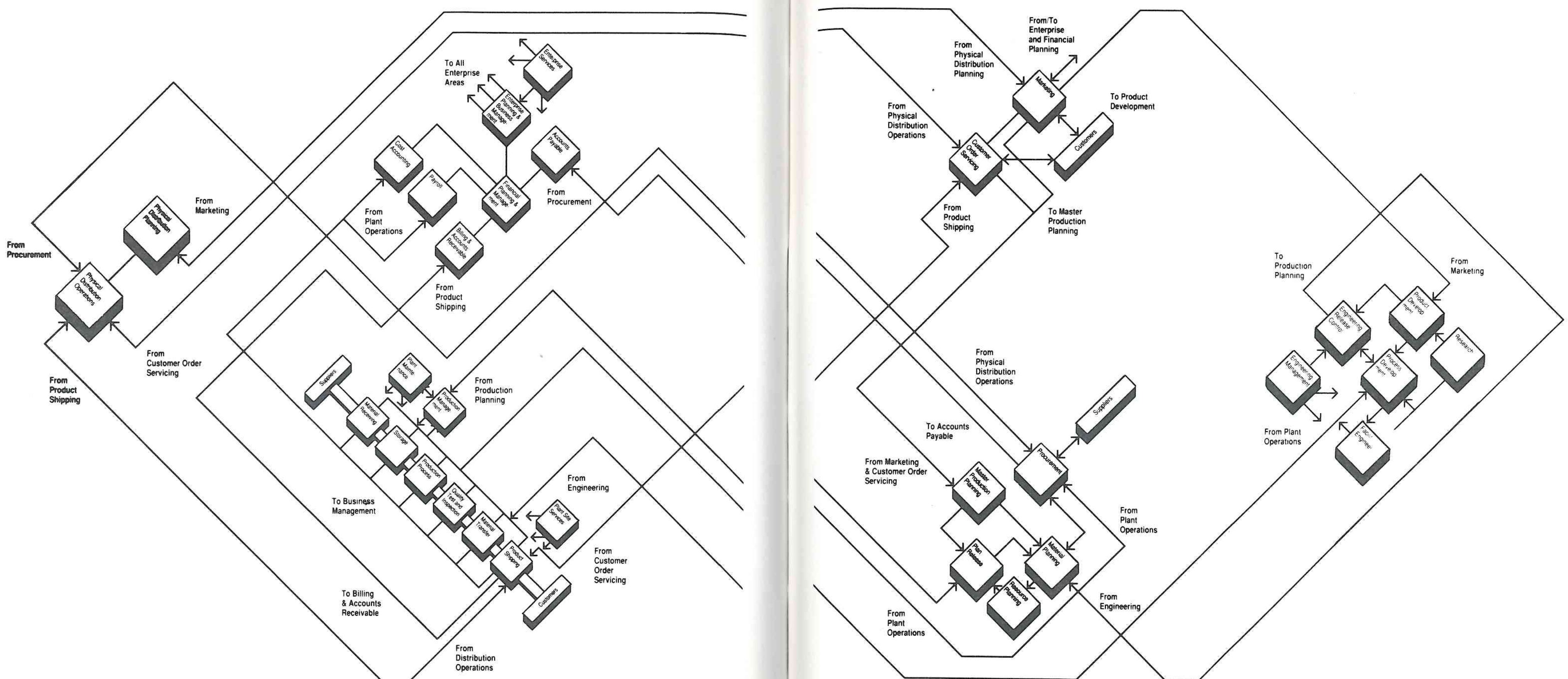


Figure 11.15 *The Enterprise workflow. (Courtesy of IBM.)*

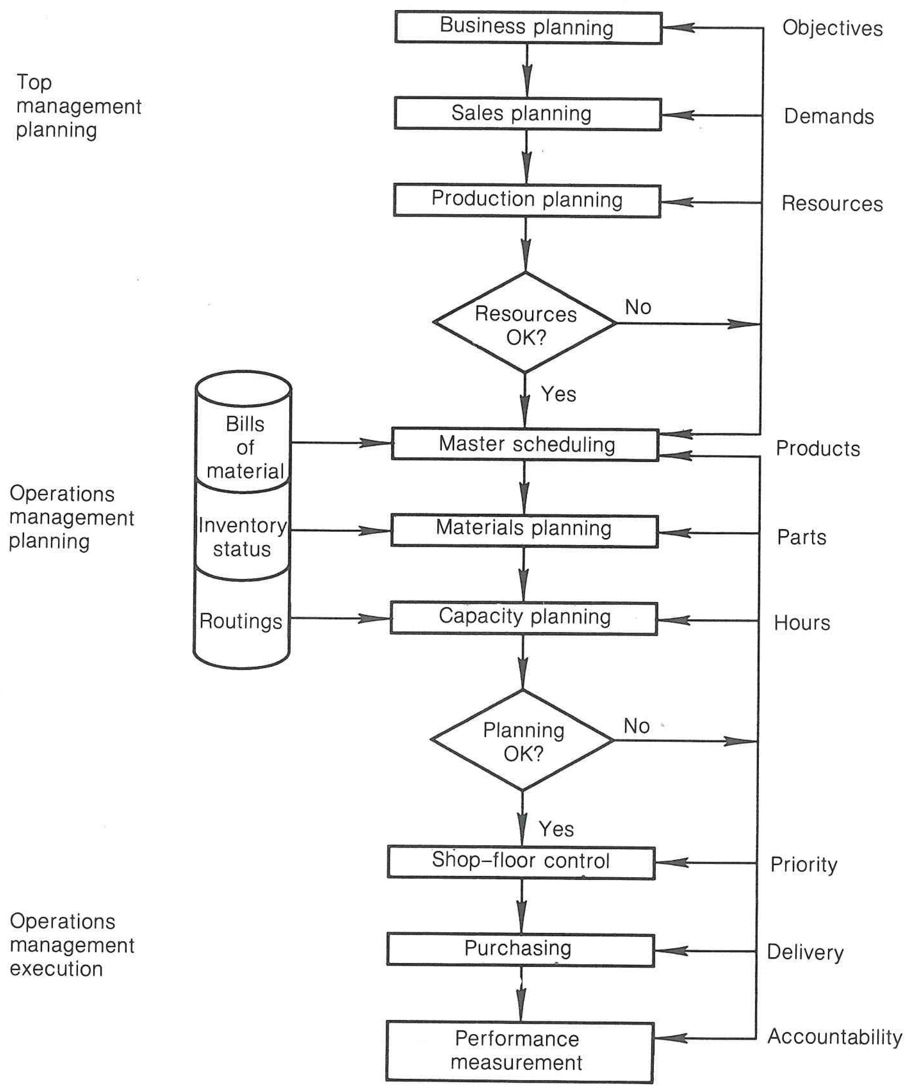


Figure 11.16 *Business management levels.*

ment levels makes sure everyone has the same information for planning and problem solving.

In Figure 11.17 are the modules of a computer software package developed by IBM to take care of the previously mentioned business tasks in a company's development of CIM. The package diagrammed is called MAPICS (Manufacturing Accounting and Production Information Control System) and is made up of 16 modules, which can be broken down into four major areas. The dia-

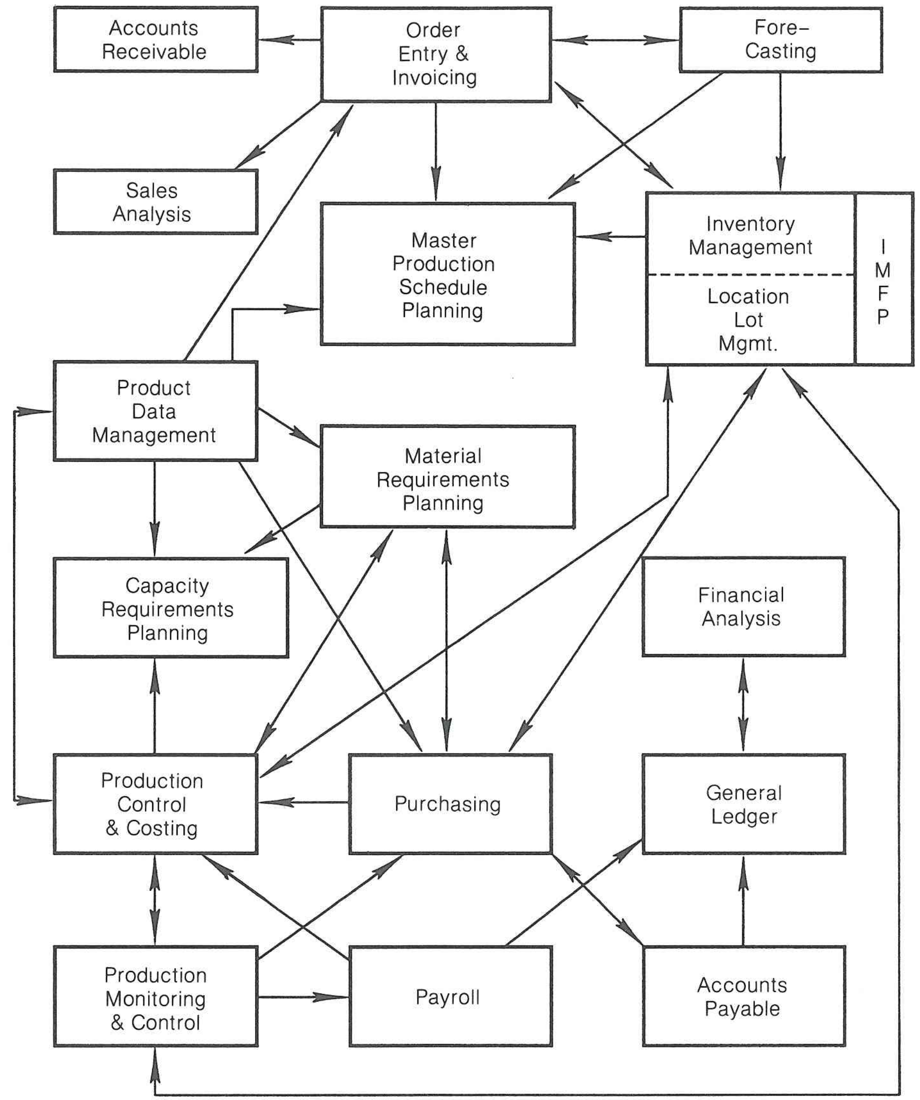


Figure 11.17 *The MAPICS applications (showing major interfaces only).*

gram shows you how various types of information will cross applications in the database use.

The first application is Plant Operations, which is made up of inventory management, production control and costing, production monitoring and control, and purchasing. Inventory management improves plant productivity through reduced and improved accuracy of parts inventory. Production Control and Costing allows management to track production orders. The computer will highlight excessive material and labor costs for correction, pinpoint part locations, show production time remaining, and report daily quantities completed. This module will allow daily priority reports to be created so work can be sequenced for meeting delivery requirements. Production Monitoring and Control through timely and accurate shop reporting helps ensure work is progressing, and orders are met promptly. This application receives orders that can be added to, modified, or split, and then print shop instruction packets. The Purchasing section allows for maintaining valid quotes giving buyers more time to negotiate, and analyzes vendor performance. The system also allows for the tracking of purchase orders and validating of vendor invoices.

The second MAPICS area of application is Marketing and Physical Distribution, which is made up of the Order Entry and Invoicing, Sales Analysis, and Forecasting modules. The order entry point is the key starting point of a manufacturing organization. Detailed reports can improve cash flow, help manage inventory costs, and analyze product performance. The order entry updates both inventory and accounts receivable records. From sales records analysis reports can be generated to forecast/predict future sales and needs. This software will help ensure reasonable ship dates, coordinate inventory and production requirements, and automatically price customer orders.

The third MAPICS application is Production Planning made up of Master Production Schedule Planning, Material Requirements Planning, Capacity Requirements Planning, and Product Data Management. Production planning supplies information for optimizing resources and minimizing costs, so production stays on schedule and meets forecasts. Master production schedule planning allows production plans to be analyzed according to inventory levels, sales projections, and resource projections so that a master production schedule can be generated. Material requirements planning software creates a purchasing or manufacturing order recommendation report to be able to meet the master production schedule. Capacity requirements planning is used to calculate the future demands on machinery and manpower, both long and short range, to help reduce bottlenecks. Product data management is used to maintain the information database for engineering, manufacturing, and accounting. This file maintains information on all part numbers, bill of materials, work stations, and routings. With the product data information, cost simulations and analyses can be performed.

The last MAPICS application is Financial Management and Business Control made up of General Ledger, Financial Analysis, Accounts Payable, Payroll, and Accounts Receivable. It is here that up to date financial information is maintained for management decision making. This financial database gives tighter controls while making sure everyone gets the same fast, accurate information. General ledger allows you to get clear reports on the company's operating performance. Financial analysis will report on significant trends so additional production planning can occur. Accounts payable gives accurate, timely information on invoice due dates, vendors, and amounts. Accounts payable software is a flexible way to manage cash outflow and analyze vendor performance. The payroll unit performs all payroll tasks as well as attendance reporting and payroll cost monitoring. Accounts receivable helps minimize collection periods, monitor cash flow in, and maintain records of good and bad customers.

It is easily seen that it would be a major advantage to have one computer system and database to control all the business functions. Everyone has the same up to date information to do their own planning and report generation. This IBM product runs on a midrange business computer. It should be noted that other companies also supply similar forms of software. The size of the computer running the software will have an effect on its capabilities. Software for some or all of these abilities are available on computer systems from personal size to main frames.

Remember, it is when we combine business control systems, engineering computer systems (CAD/CAM and computer-aided engineering analysis) and automated plant floor systems that we actually have true CIM. Many companies start out with pieces of CIM and will be building a system over a number of years. It is the computer that puts speed and ease into integration but it will take human integration also for the process to work.

QUESTIONS

1 What is a flexible manufacturing system?

2 What characteristic of computer technology makes the concept of FMS feasible?

3 What do you understand by the term in-process measurement? Explain how it might be applied to checking the size of a turned diameter.

4 What is torque monitoring? State two instances where its application may be useful in an automated machining process.

5 How are sound sensors used to monitor cutting-tool condition?

6 What is a sister tool?

7 Describe the technique of digitizing.

8 Describe parametric programming and explain the advantages of the technique.

9 Write a brief description of CIM.

10 What three major areas of a company make up the three rings of CIM?

11 What is it that takes place in a computerized business control system such as MAPICS?

APPENDIX A

ELECTRONICS INDUSTRIES ASSOCIATION SPECIFICATION

EIA Standards RS-274-D, "Interchangeable Variable Block Data Format for Positioning, Contouring, and Contouring/Positioning Numerically Controlled Machines," and RS-358-B, "Sublet of American National Standard Code for Information Interchange for Numerical Machine Control Perforated Tape" are two standards that contain useful information pertinent to the subject matter of this book. Copies of these standards may be obtained from the Electronic Industries Association, 200 Eye Street, N.W., Washington, DC 20006.

APPENDIX B

CUTTING SPEED AND FEED INFORMATION, CARBIDE GRADES, AND POWER REQUIREMENTS FORMULAS

Information Sheet

How to Calculate Revolutions per Minute when the Cutting Speed is Given (surface feet per minute)

From the past experience and tests, the following values of cutting speeds (CS) for the materials and operation shown in the tables will be used as *maximum values* in this shop. These values are given to use for *high-speed-steel tools:*

$$\text{RPM} = \frac{4\,(\text{CS})}{D} \qquad \text{CS} = \frac{D(\text{RPM})}{4}$$

D = diameter of tool.

Material	Turning	Drilling	Shaping & Milling	Reaming
Mild Steel or Cold Rolled Steel	90 ft/min	80 ft/min	70 ft/min	40 ft/min
S. A. E. 1144	100 ft/min	90 ft/min	90 ft/min	50 ft/min
S. A. E. 4130	50 ft/min	45 ft/min	40 ft/min	25 ft/min
1% Carbon Steel	50 ft/min	45 ft/min	40 ft/min	25 ft/min
High Speed Steel	45 ft/min	40 ft/min	40 ft/min	20 ft/min
Rolled Brass	150–200 ft/min	120 ft/min	125 ft/min	70 ft/min
Cast Brass	100–125 ft/min	100 ft/min	100 ft/min	50 ft/min
Cast Iron	75 ft/min	70 ft/min	60 ft/min	35 ft/min
Cast Aluminum	250 ft/min	250 ft/min	200 ft/min	100 ft/min
Wrought Aluminum	400 ft/min	400 ft/min	400 ft/min	200 ft/min

Lathe cutting speeds for carbide tools (surface feet per minute)

Material Machined	Depth of Cut (in.)	Feed per Revolution (in.)	Surface Feet per Minute
Aluminum	.005–.015	.002–.005	700–1000
	.020–.090	.005–.015	450–700
	.100–.200	.015–.030	300–450
	.300–.700	.030–.090	100–200
Brass, bronze	.005–.015	.002–.005	700–800
	.020–.090	.005–.015	600–700
	.100–.200	.015–.030	500–600
	.300–.700	.030–.090	200–400
Cast iron (medium)	.005–.015	.002–.005	350–450
	.020–.090	.005–.015	250–350
	.100–.200	.015–.030	200–250
	.300–.700	.030–.090	75–150
Machine steel	.005–.015	.002–.005	700–1000
	.020–.090	.005–.015	550–700
	.100–.200	.015–.030	400–550
	.300–.700	.030–.090	150–300
Tool steel	.005–.015	.002–.005	500–750
	.020–.090	.005–.015	400–500
	.100–.200	.015–.030	300–400
	.300–.700	.030–.090	100–300
Stainless steel	.005–.015	.002–.005	375–500
	.020–.090	.005–.015	300–375
	.100–.200	.015–.030	250–300
	.300–.700	.030–.090	75–175
Titanium alloys	.005–.015	.002–.005	300–400
	.020–.090	.005–.015	200–300
	.100–.200	.015–.030	175–200
	.300–.700	.030–.090	50–125

Note: Cutting speeds obtained depend on operation setup and machine tool.

Suggested feed per tooth for *high speed* steel milling cutters

Material	Face Mills	Helcal Mills	Slotting and Side Mills	End Mills	Form Relieved Cutters	Circular Saws
Plastic	.013	.010	.008	.007	.004	.003
Magnesium and Alloys	.022	.018	.013	.011	.007	.005
Aluminum and Alloys	.022	.018	.013	.011	.007	.005
Free Cutting Brasses and Bronzes	.022	.018	.013	.011	.007	.005
Medium Brasses and Bronzes	.014	.011	.008	.007	.004	.003
Hard Brasses and Bronzes	.009	.007	.006	.005	.003	.002
Copper	.012	.010	.007	.006	.004	.003
Cast Iron, Soft (150–180 B.H.)	.016	.013	.009	.008	.005	.004
Cast Iron, Medium (180–220 B.H.)	.013	.010	.007	.007	.004	.003
Cast Iron, Hard (220–300 B. H.)	.011	.008	.006	.006	.003	.003
Malleable Iron	.012	.010	.007	.006	.004	.003
Cast Steel	.012	.010	.007	.006	.004	.003
Low Carbon Steel, Free Machining	.012	.010	.007	.006	.004	.003
Low Carbon Steel	.010	.008	.006	.005	.003	.003
Medium Carbon Steel	.010	.008	.006	.005	.003	.003
Alloy Steel, Annealed (180–220 B. H.)	.008	.007	.005	.004	.003	.002
Alloy Steel, Tough (220–300 B. H.)	.006	.005	.004	.003	.002	.002
Alloy Steel, Hard (300–400 B. H.)	.004	.003	.003	.002	.002	.004
Stainless Steels, Free Machining	.010	.008	.006	.005	.003	.002
Strainless Steels	.006	.005	.004	.003	.002	.002
Monel Metals	.008	.007	.005	.004	.003	.002

Courtesy of Carboloy Inc., A Seco Tools Co.

Suggested feed per tooth—milling

Material	Face Mills		Helical mills		Slotting & side mills		End mills		Form relieved cutters		Circular saws	
	Hss	Carb	Hss	Carb	Hss	Carb	Hss	Carb	Hss	Carb	Hss	Carb
Plastics	.013	.015	.010	.012	.008	.009	.007	.007	.004	.005	.003	.004
Magnesium and Alloys	.022	.020	.018	.016	.013	.012	.011	.010	.007	.006	.005	.005
Aluminum and Alloys	.022	.020	.018	.016	.013	.012	.011	.010	.007	.006	.005	.005
Free Cutting Brasses & Bronzes	.022	.020	.018	.016	.013	.012	.011	.010	.007	.006	.005	.005
Medium Brasses and Bronzes	.014	.012	.011	.010	.008	.007	.007	.006	.004	.004	.003	.003
Hard Brasses and Bronzes	.009	.010	.007	.008	.006	.006	.005	.005	.003	.003	.002	.003
Copper	.012	.012	.010	.009	.007	.007	.006	.006	.004	.004	.003	.003
Cast Iron, Soft (150–180 B.H.)	.016	.020	.013	.016	.009	.012	.008	.010	.005	.006	.004	.005
Cast Iron, Medium (180–220 B.H.)	.013	.015	.010	.013	.007	.010	.007	.008	.004	.005	.003	.004
Cast Iron, Hard (220–300 B.H.)	.001	.012	.008	.010	.006	.007	.006	.006	.003	.004	.003	.003
Malleable Iron	.012	.014	.010	.011	.007	.008	.006	.007	.004	.004	.003	.004
Cast Steel	.012	.014	.010	.011	.007	.008	.006	.007	.004	.005	.003	.004
Low Carbon Steel, Free Machining	.012	.016	.010	.013	.007	.009	.006	.008	.004	.005	.003	.004
Low Carbon Steel	.010	.014	.008	.011	.006	.008	.005	.007	.003	.004	.003	.004
Medium Carbon Steel	.010	.014	.008	.011	.006	.008	.005	.007	.003	.004	.003	.004
Alloy Steel, Annealed (180–220 B.H.)	.008	.014	.007	.011	.005	.008	.004	.007	.003	.004	.002	.004
Alloy Steel, Tough (220–300 B.H.)	.006	.012	.005	.010	.004	.007	.003	.006	.002	.004	.002	.003
Alloy Steel, Hard (300–400 B.H.)	.004	.010	.003	.008	.003	.006	.002	.005	.002	.003	.001	.003
Stainless Steels, Free Machining	.010	.014	.008	.011	.006	.008	.005	.007	.003	.004	.002	.004
Stainless Steels	.006	.010	.005	.008	.004	.006	.003	.005	.002	.003	.002	.008
Monel Metals	.008	.010	.007	.008	.005	.006	.004	.005	.003	.003	.002	.003

Courtesy of Carboloy Inc., A Seco Tools Co.

Suggested feed per tooth for *sintered carbide* tipped cutters

Material	Face Mills	Helical Mills	Slotting and Side Mills	End Mills	Form Relieved Cutters	Circular Saws
Plastic	.015	.012	.009	.007	.005	.004
Magnesium and Alloys	.020	.016	.012	.010	.006	.005
Aluminum Alloys	.020	.016	.012	.010	.006	.005
Free Cutting Brasses and Bronzes	.020	.016	.012	.010	.006	.005
Medium Brasses and Bronzes	.012	.010	.007	.006	.004	.003
Hard Brasses and Bronzes	.010	.008	.006	.005	.003	.003
Copper	.012	.009	.007	.006	.004	.003
Cast Iron, Soft (150–180 B. H.)	.020	.016	.012	.010	.006	.005
Cast Iron, Medium (180–220 B. H.)	.016	.013	.010	.008	.005	.004
Cast Iron, Hard (220–300 B. H.)	.012	.010	.007	.006	.004	.003
Malleable Iron	.014	.011	.008	.007	.004	.004
Cast Steel	.014	.011	.008	.007	.005	.004
Low Carbon Steel, Free Machining	.016	.013	.009	.008	.005	.004
Low Carbon Steel	.014	.011	.008	.007	.004	.004
Medium Carbon Steel	.014	.011	.008	.007	.004	.004
Alloy Steel, Annealed (180–220 B. H.)	.014	.011	.008	.007	.004	.004
Alloy Steel, Tough (220–300 B. H.)	.012	.010	.007	.006	.004	.003
Alloy Steel, Hard (300–400 B. H.)	.010	.008	.006	.005	.003	.003
Stainless Steels, Free Machining	.014	.011	.008	.007	.004	.004
Stainless Steels	.010	.008	.006	.005	.003	.003
Monel Metals	.010	.008	.006	.005	.003	.003

Courtesy of Carboloy Inc., A Seco Tools Co.

Cutting data for drilling (meters per minute)

(Reproduced by kind permission of Guhring Ltd.)

Material of workpiece	Drill type	Material of drill	Point angle	Cutting speed m per min.	Feed scale No. (see p. 468)	Coolant
Free-cutting mild steel hardness up to 500 N/mm²	N/GT 50	HSS	118	30 – 50	4	Soluble oil
Non-alloyed carbon steel with ≤ 0.4% carbon ≤ 800 N/mm	N	HSS	118	20 – 30	4	Soluble oil
Non-alloyed carbon steel with > 0.4% carbon, hardness 800–1000 N/mm² and purified alloy steel with a hardness ≤ 700 N/mm²	N/GT 100	HSS	118	16 – 20	3	Soluble oil
Non-alloyed tool steels with a hardness of 800–1000 N/mm² and refined alloy steels with a hardness of 700–1000 N/mm²	N/GT 100	HSS	118	12 – 16	3	Soluble oil
Alloyed tool steels hardness 800–1000 N/mm² and refined alloy steels with a hardness of 1000–1200 N/mm²	N/GV	HSCO (HSS)	118 (130)	10 – 16	2	Soluble oil
Refined alloy steels with a hardness of > 1200 N/mm²	N/GV	HSCO	130°	5 – 8	1	Soluble oil, cutting oil
Chrome-molybdenum, stainless steel	N	HSCO	130	8 – 12	1	Soluble oil, cutting oil
Stainless, austenitic, nickel-chrome, heat resisting steels	N/Ti (Specials)	HSCO	130	3 – 8	1	Cutting oil or cutting oil with molybdenum disulphide additives
Manganese steels containing up to 10% molybdenum	H (Specials)	HSCO	130	3 – 5	1	Dry: preheat to 200 –300
Spring steels	N/GV	HSCO (HSS)	130	5 – 10	1	Soluble oil, cutting oil
Nimonic alloys	W/Ti (Specials)	HSCO	130°	3 – 8	1	Cutting oil or cutting oil with molybdenum disulphide additives
Ferro-tic	N/Ti	HSCO	118°/130	3 – 5	1	Dry: compressed air
Titanium and titanium alloys	Ti (Specials)	HSCO	130	3 – 5	1	Cutting oil or cutting oil with molybdenum disulphide additives
Grey cast iron up to GG 26 and malleable iron	N	HSS (HSCO) (double angle point)	118 /90°	16 – 25	5	Dry: soluble oil
Hard cast iron up to 350 brinell	N	HSCO (double angle point)	118 /90°	8 – 12	4	Dry: soluble oil
Brass to MS 58	H	HSS	118	60 – 80	6	Dry: cutting oil
Brass from MS 60	H (N)	HSS	118	30 – 60	5	Soluble oil, cutting oil
Red copper	W/GT 50	HSS	130	30 – 60	5	Soluble oil, cutting oil

Cutting data for drilling (meters per minute) (contd.)

Material of workpiece	Drill type	Material of drill	Point angle	Cutting speed m per min.	Feed scale No.	Coolant
Electrolytic copper	N	HSS	130°	20 – 30	5	Soluble oil, cutting oil
German silver	N	HSS	118°	20 – 30	3	Soluble oil, cutting oil
Copper nickel and copper-tin alloys	N	HSS	130°	20 – 30	3	Soluble oil, cutting oil
Copper-aluminium alloys	N	HSS	130	10 – 30	3	Soluble oil, cutting oil
Alloys of copper and beryllium	H	HSS	130°	10 – 16	2	Soluble oil, cutting oil
Copper-manganese and copper-silicon alloys	N	HSS	130°	25 – 30	3	Soluble oil, cutting oil
Pure aluminium	W/GT 50	HSS	130°	40 – 60	5	Soluble oil
Aluminium-manganese and aluminium-chrome alloys	W/GT 50	HSS	130°	40 – 60	5	Soluble oil
Aluminium alloyed with lead, antimony or tin	W/GT 50	HSS	130°	60 –100	5	Soluble oil
Aluminium-copper alloys containing silicon, magnesium, lead, tin, titanium or beryllium	W/GT 50	HSS	130°	40 – 60	5	Soluble oil
Aluminium-silicon alloys containing copper, magnesium, manganese or chrome	W/GT 50	HSS	130°	40 – 60	5	Soluble oil
Aluminium-magnesium alloys with silicon, manganese or chrome	W/GT 50	HSS	130°	60 –100	5	Soluble oil
Magnesium and magnesium alloys (Electron)	W/GT 50	HSS	130°	80 –100	5	Dry
Zinc, Zamac	N	HSS	118	30 – 40	4	Soluble oil
Hard duroplastics	H	HSS/HM	80	10-20 50-100	3/4	Dry: Compressed air
Soft thermoplastics	W/GT 50	HSS	130°	16 – 40	3	Water: Compressed air
Hardboard and the like	W/H*	HSS	130°	16 – 25	3	Dry: Compressed air
Eternit, slate, marble	H	HSS (HM)	80	3 – 5	from hand	Dry: Compressed air
Graphite	N	HSS (HM)	80	3 – 5	from hand	Dry: Compressed air
Ebonite, Vulcanite	H	HSS	80°	16 – 30	6	Dry: Compressed air
Perspex	H	HSS	130°	16 – 25	3	Water

*) W = Drilling in the direction of the layers
H = Drilling at right angles to the layers

N.B. The foregoing recommendations hold good only if the following conditions are met with:

a) uniform consistency of the material to be drilled;
b) drills are to B.S. 328 and DIN 338;
c) Guhring drills of HSS and HSCO quality are used;
d) maximum depth does not exceed 3 times the drill diameter;
e) good machine condition and rigid mounting of the workpiece;
f) no drilling bushes are used;
g) correct quality of coolant and sufficient flow
h) no excessive run-out of the machine spindle or drill.

Bearing these points in mind the figures in the tables may be increased or reduced accordingly.

Feed rate scales in mm. per spindle revolution
Scale No.:

Drill dia. mm.	1 (mm. per rev. s)	2 (mm. per rev. s)	3 (mm. per rev. s)	4 (mm. per rev. s)	5 (mm. per rev. s)	6 (mm. per rev. s)
80	0,4	0,5	0,63	0,8	1,0	1,25
63	0,315	0,4	0,5	0,63	0,8	1,0
50	0,25	0,315	0,4	0,5	0,63	0,8
40	0,2	0,25	0,315	0,4	0,5	0,63
31,5	0,16	0,2	0,25	0,315	0,4	0,5
25	0,16	0,2	0,25	0,315	0,4	0,5
20	0,125	0,16	0,2	0,25	0,315	0,4
16	0,1	0,125	0,16	0,2	0,25	0,315
12,5	0,08	0,1	0,125	0,16	0,2	0,25
10	0,08	0,1	0,125	0,16	0,2	0,25
8	0,063	0,08	0,1	0,125	0,16	0,2
6,3	0,05	0,063	0,08	0,1	0,125	0,16
5	0,04	0,05	0,063	0,08	0,1	0,125
4	0,04	0,05	0,063	0,08	0,1	0,125
3,15	0,032	0,04	0,05	0,063	0,08	0,1
2,5	0,025	0,032	0,04	0,05	0,063	0,08
2	0,02	0,025	0,032	0,04	0,05	0,063

Cutting data for turning (meters per minute)
(Reproduced by kind permission of Anderson Strathclyde PLC)

MATERIAL	SPEED FT/MIN M/MIN			FEED INS/REV MM/REV			DEPTH INS MM			GRADE	
	ROUGH	FINISH		ROUGH	FINISH		ROUGH	FINISH		ROUGH	FINISH
ALUMINUM ALLOYS	800	1600	2500	.04	.02	.008	.25	.18	.01	CG	CF
	250	500	750	1.	.5	.2	6.5	4.5	.25		
ALUMINUM CASTINGS	800	1600	2500	.04	.02	.008	.25	.18	.01	CG	CF
	250	500	750	1.	.5	.2	6.5	4.5	.25		
ALUMINUM CASTINGS, HT. TREATED	300	600	1600	.04	.02	.008	.25	.18	.01	CG	CF
	90	180	500	1.	.5	.2	6.5	4.5	.25		
BRASS	600	750	1000	.04	.02	.008	.25	.18	.01	CG	CF
	180	230	300	1.	.5	.2	6.5	4.5	.25		
BRONZE, PHOSPHOR	300	600	800	.04	.02	.008	.25	.18	.01	CG	CF
	90	180	250	1.	.5	.2	6.5	4.5	.25		
CAST IRON, ALLOY	150	350	500	.04	.02	.008	.25	.18	.01	CR	CG
	45	105	150	1.	.5	.2	6.5	4.5	.25		
CAST IRON, CHILLED 400B	30	60	100	.04	.02	.008	.25	.18	.01	CG	CG
	9	18	30	1.	.5	.2	6.5	4.5	.25		
CAST IRON, CHILLED 600B	25	50	60	.04	.02	.008	.25	.18	.01	CG	CG
	8	15	18	1.	.5	.2	6.5	4.5	.25		
CAST IRON, GREY	250	550	650	.04	.02	.008	.3	.2	.01	CR	CG
	75	165	190	1.	.5	.2	7.5	5.	.25		
CAST IRON, NODULAR, FERRITIC	150	300	500	.04	.02	.008	.2	.1	.01	CR	CG
	45	90	150	1.	.5	.2	5.	2.5	.25		
CAST IRON, NODULAR, PEARLITIC	150	300	450	.04	.02	.008	.2	.1	.01	CR	CG
	45	90	135	1.	.5	.2	5.	2.5	.25		
COPPER	600	1100	2000	.04	.02	.008	.25	.18	.01	CR	CG
	180	330	600	1.	.5	.2	6.5	4.5	.25		
FIBRE	300	500	700	.04	.02	.008	.25	.18	.01	CG	CF
	90	150	210	1.	.5	.2	6.5	4.5	.25		
HARD RUBBER, ASBESTOS	600	800	1000	.04	.02	.008	.3	.2	.01	CF	CF
	180	250	300	1.	.5	.2	7.5	5.	.25		
LEAD BRONZE, ALLOY	750	1000	1500	04	.02	.008	.25	.18	.01	CW	CG
	230	300	450	1.	.5	.2	6.5	4.5	.25		

MATERIAL	SPEED FT/MIN M/MIN			FEED INS/REV MM/REV			DEPTH INS MM			GRADE	
	ROUGH	FINISH		ROUGH	FINISH		ROUGH	FINISH		ROUGH	FINISH
MALLEABLE IRON, LONG CHIP	150	500	650	.04	.02	.008	.3	.2	.01	M1	M1
	45	150	190	1.	.5	.2	7.5	5.	.25		
MALLEABLE IRON, SHORT CHIP	250	400	600	.04	.02	.008	.3	.2	.01	M1/1	M1/1
	75	120	180	1.	.5	.2	7.5	5.	.25		
PORCELAIN	50	60	80	.04	.02	.008	.1	.01	.003	CF	CF
	15	18	25	1.	.5	.2	2.5	.25	.08		
RIGID PLASTICS, WOOD	600	800	1300	.04	.02	.008	.3	.2	.01	CW	CF
	180	250	390	1.	.5	.2	7.5	5.	.25		
STEEL, ALLOY, ANNEALED	300	500	650	.03	.01	.004	.5	.25	.015	SG	SF
	90	150	190	.75	.25	.1	13.	6.5	.4		
STEEL, ALLOY, HARDENED 250B	250	400	600	.03	.01	.004	.3	.2	.01	SG	SF
	75	120	180	.75	.25	.1	7.5	5.	.25		
STEEL, ALLOY, HARDENED 300B	200	300	500	.03	.01	.004	.25	.18	.01	SG	SF
	60	90	150	.75	.25	.1	6.5	4.5	.25		
STEEL, ALLOY, HARDENED 400B	150	250	350	.03	.01	.004	.25	.18	.01	SG	SF
	45	75	100	.75	.25	.1	6.5	4.5	.25		
STEEL, CARBON NORMALISED 125B	600	800	1100	.03	.01	.004	.5	.25	.015	SG	SF
	180	250	330	.75	.25	.1	13.	6.5	.4		
STEEL, CARBON NORMALISED 150B	400	650	1000	.03	.01	.004	.5	.25	.015	SG	SF
	120	190	300	.75	.25	.1	13.	6.5	.4		
STEEL, CARBON NORMALISED 250B	300	500	650	.03	.01	.004	.3	.2	.01	SG	SF
	90	150	190	.75	.25	.1	7.5	5.	.25		
STEEL, CAST 150B	200	300	500	.05	.01	.006	.5	.25	.015	SG	SF
	60	90	150	1.25	.25	.15	13.	6.5	.4		
STEEL, CAST 250B	150	250	350	.05	.01	.006	.3	.2	.01	SG	SF
	45	75	100	1.25	.25	.15	7.5	5.	.25		
STEEL, MANGANESE	60	100	200	.04	.02	.008	.3	.2	.01	CW	CG
	18	30	60	1.	.5	.2	7.5	5.	.25		
STEEL, STAINLESS, AUSTENTIC	300	400	500	.08	.015	.008	.25	.18	.01	CW	CW
	90	120	150	2.	.4	.2	6.5	4.5	.25		
STEEL, STAINLESS, MARTENSITIC	300	400	600	.08	.015	.008	.25	.18	.01	SG	SF
	90	120	180	2.	.4	.2	6.5	4.5	.25		
STEEL, TOOL, HARDENED	30	60	100	.04	.02	.008	.2	.1	.01	SG	SF
	9	18	30	1.	.5	.2	5.	2.5	.25		
STONE, HARD GRANITE	25	35	50	.04	.02	.008	.2	.1	.01	CR	CR
	8	10	15	1.	.5	2	5.	2.5	.25		
STONE, SOFT MARBLE	150	200	250	.04	.02	.008	.2	.1	.01	CF	CF
	45	60	75	1.	.5	.2	5	2.5	.25		

Kennametal Grade System

Three general purpose grades—KC850, K420, and K68—can machine all the jobs in most shops. Characteristics of these grades are:

KC850 is a rugged, multi-coated general purpose grade that delivers consistent results at accelerated speeds and feeds when machining steels. It is ideal for a wide range of jobs from roughing to semi-finishing.

K420 combines an optimum balance of toughness, with an ability to resist edge wear and cratering. It can handle heavy roughing of steel at low speeds, to light roughing at moderate speeds.

K68 was created specifically for machining aerospace metals such as superalloys and refractory metals, plus cast irons, non-ferrous metals and their alloys, and non-metals.

KC850, K420 and K68 are recommended as a means to minimize tool inventory and simplify grade selection. However, the nineteen grades listed in the table below will more precisely cover any metalcutting condition likely to be encountered. Properly applied, these tungsten carbide, titanium carbide, coated carbide, ceramic and cermet grades will provide optimum results.

Grade	Hardness R_A	Typical Machining Applications
K090	94.5	Hot pressed cermet for medium roughing and finishing cast irons and heat-treated steels at high to moderate speeds and moderate to high chip loads.
K060	93-94	Cold pressed ceramic for roughing and finishing cast iron below 300 BHN, and steels up to 23-24 R_C at speeds up to 2000 sfm.
K165	93.5	Titanium carbide for finishing steels and cast irons at high to moderate speeds and light chip loads.
K7H	93.5	For finishing steels at higher speeds and moderate chip loads.
K5H	93.0	For finishing and light roughing of steels at moderate speeds and chip loads through light interruptions.
KC850	—	A rugged coated grade that produces consistent results at accelerated feeds and speeds when machining steels.
KC810	—	Coated for general machining of steels over a wide range of speeds in moderate roughing to semifinishing applications.
K45	92.5	The hardest of this group. General purpose grade for light roughing to semifinishing of steels at moderate speeds and chip loads, and for many low speed, light chip load applications.
K4H	92.0	For light roughing to semifinishing of steels at moderate speeds and chip loads, and for form tools and tools that must dwell.
K2S	91.5	For light to moderate roughing of steels at moderate speeds and feeds through medium interruptions.
K2884	92.0	A general purpose grade for milling steel over a wide range of conditions from moderate to heavy chip loads.
K21	91.0	For moderate to heavy roughing of steels at moderate speeds and heavy chip loads through medium interruptions where mechanical and thermal shock are encountered.
K420	91.3	For heavy roughing of steels at low to moderate speeds and heavy chip loads through interruptions where mechanical and severe thermal shocks are encountered.
KC210	—	Coated for general machining of cast iron over a wide range of speeds in moderate roughing to finishing applications.
K11	93.0	The hardest of this group. For precision finishing of cast irons, nonferrous alloys, nonmetals at high speeds and light chip loads, and for finishing many hard steels at low speeds and light chip loads.
K68	92.6	General purpose grade for light roughing to finishing of most high temperature alloys, refractory metals, cast irons, nonferrous alloys, and nonmetals at moderate speeds and chip loads through light interruptions.
K6	92.0	For moderate roughing of most high temperature alloys, cast irons, nonferrous alloys and nonmetals at moderate to low speeds and moderate to heavy chip loads through light interruptions.
K8735	92.0	For broaching and milling gray, malleable and nodular cast irons at moderate speeds and chip loads.
K1	90.0	The most shock resistant of this group. For heavy roughing of most high temperature alloys, cast irons, and nonferrous alloys at low speeds and heavy chip loads through heavy interruptions.

Note: Above grade designations are trademarks of Kennametal Inc.

Tooling grades—Metric ISO

HOYBIDE GRADE	WORKPIECE MATERIAL AND OPERATION	ISO CODE	COLOUR CODE
SG	For all general purpose steel cutting operations, with the exceptions of stainless steels, high-nickel alloys and heat and creep-resistant alloys.	P20	Blue
CG	For all machining operations on cast irons, meehanite, non-ferrous metals and short-chip alloy irons.	K20	Red
CW	For machining stainless steels, high-nickel alloys and heat and creep-resistant alloys. It can also be used as a universal grade for steels, cast irons, non-ferrous metals, plastics, wood, etc.	P30 M30 K20 K30	Yellow
GT	TiN coated, for machining steels, stainless steel, and general purpose operations on other metals at higher speeds with longer tool life.	P20 P30 P35 K20	Gold

The grades shown above are normally stocked in a wide range of sizes of square, triangular and rhombic, clamp and pin type, negative and positive rake conventional inserts.

Technical Data

Kennametal Grade Selection

Suggested grades and machining conditions for various work materials and types of cut.

Work Material	Hardness		FINISHING				LIGHT ROUGHING				HEAVY ROUGHING			
			Up to 1/16 Depth Under .005 Feed		1/16–3/16 Depth .005–.015 Feed		1/8–3/8 Depth .010–.030 Feed		1/4–1/2 Depth .020–.050 Feed		3/8–3/4 Depth .030–.070 Feed		5/8 & Up Depth .040–.100 Feed	
	BHN	RC	Speed	Grade	Speed	Grade	Speed	Grade	Speed	Grade	Speed	Grade	Speed	Grade
Free Machining Steels, Plain Carbon Steels, Alloy Steels, and 400 & 500 Series Stainless Steels*	150		1500–2000		1000–1600		500–1200		375–500		300–450		275–400	K21
	200		1200–1800		900–1200	K45	450–1000		325–450		275–425		250–375	
	250	24	800–1500	K7H	700–1000		400–900		300–400		250–400	K21	225–350	
	300	32	650–850		500–750		350–600		275–350	KC850/KC810 K45	225–350		200–325	
	350	37	550–700		400–600		300–450	KC850/KC810 K45	225–300		200–325		150–275	K420
	400	43	450–600		350–550	K7H	250–400		200–275		175–300		125–200	
	425	45	400–550		300–500		225–350		175–250		125–200	K420		
	450	47	375–500	K090/K060	250–450		200–300		125–200					
	475	49	300–375		200–300	K090/K060	125–200							
	500	51	250–375		125–200									
	525	53	200–275											

Coated Grades KC850 and KC810, permit increased metal removal rates by operating at the higher cutting speeds suggested for the steels in the above categories. In addition KC850 and KC810 are applied to other metals which are normally machined with the combined crater and edge-wear resistant grades.

Work Material	BHN	FINISHING Up to 1/16" Depth			LIGHT ROUGHING 1/16" to 3/8" Depth			HEAVY ROUGHING 1/4" to 3/4" Depth		
		Speed	Feed	Grade	Speed	Feed	Grade	Speed	Feed	Grade
Gray Cast Iron	110–220	400–550	.010–.020	K11 K090/K060	300–400	.020–.030	K68–K11	200–300	.025–.040	K6 K68
	220–320	250–350	.005–.010	K11 K090/K060	125–250	.010–.020	K68–K11	100–150	.020–.030	K6 K68
Nodular Iron	140–250	450–600	.010–.020	KC210 / K7H/K090/K060	325–450	.020–.030	KC210 / K45–K68	200–325	.025–.040	K420 K68
	250–400	325–425	.005–.010	K11 K090/K060	250–325	.010–.020	K45–K68	150–250	.020–.030	K420 K68
Malleable Iron	110–220	350–500	.010–.020	K7H K090/K060	250–350	.020–.030	K45–K68	150–250	.025–.040	K420 K68
	200–280	300–400	.005–.010	K11 K090/K060	200–300	.010–.020	K45–K68	125–200	.020–.030	K420 K68
Chilled Cast Iron	470–650	60–80	.008–.012	K11 K090/K060	40–60	.012–.015	K68–K11	30–40	.015–.030	K6 K68

Coated Grade KC210 offers increased metal removal capabilities in cast iron machining by operating at the higher cutting speeds suggested.

Work Material	BHN	FINISHING Up to 1/16" Depth			LIGHT ROUGHING 1/16" to 3/8" Depth			HEAVY ROUGHING 1/4" to 3/4" Depth		
		Speed	Feed	Grade	Speed	Feed	Grade	Speed	Feed	Grade
200 & 300 Series Stainless Steels*	150–250	300–600	.005–.015	K68–K11	200–300	.010–.020	K6–K68	125–200	.015–.025	K6 K68
Ultra-High Strength Steels	200–350	400–600	.010–.020	K7H K090/K060	300–400	.020–.030	K21–K45	200–300	.025–.040	K420 K21
	400–500	300–400	.008–.015	K7H/K090/K060	200–300	.015–.020	K21–K45	100–200	.020–.035	K420 K21
	500–560	200–250	.005–.010	K7H K090/K060	100–200	.010–.015	K21–K45			
Maraging Steel, 18% Ni	275–325	400–600	.010–.020	K7H K090/K060	300–400	.015–.030	K21–K45	200–300	.025–.040	K420 K21
	480–510	300–400	.005–.010	K11 K090/K060	200–300	.010–.015	K45–K68	100–200	.015–.020	K420 K21
Maraging Steel, 25% Ni	175–225	250–500	.010–.020	K7H K090/K060	175–250	.015–.030	K21–K45	100–175	.025–.040	K420 K21
	480–510	150–350	.005–.010	K11 K090/K060	125–150	.010–.015	K45–K68	75–125	.015–.020	K420 K21
High Speed Steels	200–250	400–500	.005–.010	K7H K090/K060	300–400	.010–.015	K45–K7H	200–300	.015–.020	K420 K21
	Over 250	300–400	.005–.010	K7H K090/K060	200–300	.010–.015	K45–K7H	100–200	.015–.020	K420 K21
Hot Work Tool Steels	150–250	400–500	.005–.010	K7H K090/K060	300–400	.010–.015	K45–K7H	200–300	.015–.025	K420 K21
	Over 250	200–300	.005–.008	K7H K090/K060	100–200	.008–.012	K45–K7H			
Cold Work Tool Steels	200–250	400–500	.005–.010	K7H K090/K060	300–400	.010–.015	K45–K7H	200–300	.015–.025	K420 K21
	Over 250	200–300	.005–.008	K7H K090/K060	100–200	.010–.012	K45–K7H			
High Manganese Steel	170–210	200–400	.010–.020	K7H K090/K060	100–200	.020–.030	K45–K7H	50–100	.030–.040	K420 K21
Magnesium Alloys		1200–1800	.005–.010	K68–K11	900–1200	.010–.018	K68–K11	600–900	.012–.020	K6 K68
Aluminum Alloys	Lo Si	1500–2000	.005–.010	K68–K11	1200–1800	.008–.012	K68–K11	900–1200	.012–.020	K6 K68
	Hi Si	1200–1800	.004–.008	K68–K11	900–1200	.007–.010	K68–K11	600–900	.010–.015	K6 K68
Titanium, Pure	110–275	300–450	.005–.012	K68–K11	175–300	.010–.020	K68–K11	150–225	.020–.025	K6 K68
Titanium, Alloyed	300–380	200–300	.005–.010	K68–K11	150–200	.008–.015	K68–K11	75–150	.012–.020	K6 K68
	350–440	125–200	.004–.008	K68–K11	75–125	.006–.010	K68–K11	50–100	.010–.015	K6 K68
Copper Alloys	20–60 RB	900–1200	.005–.010	K68–K11	800–1000	.008–.012	K68–K11	600–800	.012–.015	K6 K68
	60–100 RB	800–1000	.004–.008	K68–K11	600–800	.007–.010	K68–K11	400–600	.010–.014	K6 K68
Brass & Bronze	to 200	500–1000	.002–.010	K68–K11	300–600	.008–.016	K68–K11	150–300	.012–.020	K6 K68
S Monel	to 320	250–300	.003–.008	K68–K11	175–275	.008–.012	K68–K11	150–200	.012–.018	K6 K68
K Monel	to 250	250–300	.005–.010	K45–K7H	175–275	.008–.015	K45–K7H	150–200	.012–.020	K420 K21
Zinc Alloys	Cast	500–1000	.002–.008	K68–K11	250–600	.010–.015	K68–K11			
Zirconium	140–280	300–400	.005–.008	K68–K11	200–300	.008–.015	K68–K11	150–200	.012–.018	K6 K68
Manganese	140–220	275–325	.005–.008	K68–K11	200–275	.008–.015	K68–K11	150–200	.012–.018	K6 K68
Thermoplastics		600–1100	.002–.005	K11 K090/K060	200–500	.006–.012	K6–K68			
Thermosetting Plastics		900–1800	.002–.009	K11 K090/K060	500–1000	.008–.015	K6–K68			
High Temperature Alloys & Refractory Metals	Detailed machining information is contained in Kennametal publication B-451									

*Detailed machining information on specific stainless steels is contained in Kennametal publication A78-275.

Note: Above grade designations are trademarks of Kennametal Inc.

A78-370-3 Kennametal Inc. Latrobe, PA 15650

Ceramic technical data

application considerations for successful machining of high temperature alloys

Productivity 10:1 vs carbide

When machining high temperature alloys with Kennametal ceramic cutting tools, significant productivity and machining cost advantages can be realized. There are several important factors which tooling engineers must consider to optimize their particular applications.

1. depth of cut notching (d.o.c. notching)

Notching (exaggerated abrasion wear concentrated at the depth of cut intersection of the insert) is the most common mode of failure for ceramic tools when machining high temperature alloys. Excessive d.o.c. notching can lead to premature tool failure and insert breakage.

to minimize d.o.c. notching:

+ use round inserts whenever possible
+ use the longest lead angle possible
  - when using round inserts, lead angle is optimized when d.o.c. = 5% - 15% of the diameter of the insert.
  - maximum d.o.c. = 25% of the diameter of the insert.
+ use flood coolant (no interference from clamps etc)
+ program the optimum tool path for ceramic tools.
  - maximize the effective tool lead angle.
  - prechamfer the workpiece to eliminate stress points on the cutting edge of the insert. *When using a seperate prechamfer operation, feed the insert at a 90⁰ angle to the chamfer to minimize notching of the insert during the prechamfer.*
  - avoid double notching [ex: using an 80⁰ or 55⁰ insert to profile in two directions with the same nose radius can lead to a double notch situation and premature tool failure.]

2. edge chipping

possible cause	solution
- wrong grade for application - speed too low - feed too high - incorrect edge prep. - vibration	- see recommendations - see recommendations - see recommendations - see recommendations - verify workpiece and tool rigidity; reduce spindle and/or tool overhang.
- interruptions in cut	- increase edge prep. - increase speed - decrease feed - switch to thicker insert - verify rigidity as above

3. excessive wear

possible cause	solution
- wrong grade for application - wrong speed (to high or low) - wrong edge prep.	- see recommendations - see recommendations - see recommendations

4. insert breakage

possible cause	solution
- improper clamping	- use carbide chip-breaker to distribute the clamping forces - do not over torque the lock pin when using Kenloc style inserts (12 inch/lbs. max)
- scale and/or heavy interruptions on workpiece - thermal shock	- increase edge prep. and/or insert thickness - remove any coolant flow restrictions, must have flood coolant at insert / workpiece interface

5. coolant

The use of unrestricted flood coolant is recommended when machining high temperature alloy materials with Kennametal ceramics.

6. edge preparation

- standard edge preparation:

Kyon 2500 = .002 -.004 x 20⁰ T- land (T)
Kyon 2000 = .002 -.004 hone only
or .007 -.009 x 20⁰ T- land (T)

These standard edge preparations will be suitable for the majority of applications machining nickel base alloys. Kennametal realizes however, that the need may arise for special or non-stocked edge preparations. Please refer to page 101 for edge preparation specifications and recommendations.

Ceramic technical data

Ceramic application recommendations for nickel and iron base high temperature alloys

work piece material	machining conditions	finishing - light rough, turn, bore	roughing - turning, boring	milling
nickel and iron base high temperature alloys	depth of cut (in.)	up to .150	up to .250	up to .250
machinability index 15 - 30	feed rate (ipr)	.003 - .010	.005 - .012	(ipt) .004 - .012
hardness to 38 Rc	insert geometry	positive and negative Kendex Kenloc (Kyon2000 only)		SNC shear angle Kendex
(popular examples)	grade	surface speeds (sfm)		
Inco - 625	Kyon 2500 Kyon 2000	700 - 1400	400 - 800	1500 - 2000
Inco - 718	Kyon 2500 Kyon 2000	600 - 1250	400 - 800	1200 - 1600
Inco X-750	Kyon 2500 Kyon 2000	700-1250	400 - 1000	1200 - 1600
Inco - 901	Kyon 2500 Kyon 2000	400 - 1000	300 - 800	1200 - 1500
IN - 100	Kyon 2500 Kyon 2000	250 - 600	200 - 400	- - -
Rene 95	Kyon 2500 Kyon 2000	200 - 600	200 - 400	- - -
Hastelloy X	Kyon 2500 Kyon 2000	600 - 1000	400 - 800	1000 - 1400
Waspalloy	Kyon 2500 Kyon 2000	500 - 1200	400 - 1000	1200 - 1600
A - 286	Kyon 2500 Kyon 2000	500 - 800	300 - 600	---

work piece material	machining conditions	finishing - light rough, turn, bore	roughing - turning, boring	milling
cobalt based high temperature alloys	depth of cut (in.)	up to .060	- - -	up to .060
	feed rate (ipr)	.004 - .012	- - -	(ipt) .003 - .008
	insert geometry	positive and negative Kendex	- - -	negative Kendex
(popular examples)	grade	surface speeds (sfm)		
Haynes Alloy 25 Stellite	K090	600 - 1200	- - -	800 - 1500

Ceramic technical data

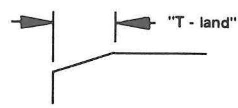
Ceramic application considerations for successful machining of carbon, alloy, tool and stainless steels

1. edge preparation

A "T- land" is recommended for normal applications of Kennametal ceramic cutting tools in steel work-piece materials.

When turning and boring:
feed rate (ipr) = 1 - 1.5 x leg length (L) of T- land.
example: feed rate = 1 - 1.5 x (.008) [std. K090 leg length]
feed rate = **.008 - .012 ipr**

When milling:
feed rate (ipt) = .75 - 1 x leg length (L) of T- land
example: feed rate = .75 - 1 x (.008 [std. K090 leg length])
feed rate = **.006 - .008 ipt**



NOTE: standard edge preparation for K060 / K090 is a .008 x 20° T - land. The need may arise for special or non-stocked edge preparations. Please see page 486 for edge preparation specifications and recommendations.

2. coolant

The use of coolant is not recommended for turning, boring and milling of steels with Kennametal ceramic cutting tools.

3. edge chipping

possible cause	solution
- wrong grade for application - speed to low - feed to high - incorrect edge prep - vibrations	- see recommendations - see recommendations - see recommendations - see recommendations - check rigidity of tool & workpiece; reduce tool and/or spindle overhang.
- interruptions in cut	- increase edge prep. - increase speed - decrease feed - use thicker insert - verify rigidity as above

4. excessive wear

possible cause	solution
- wrong grade for application - speed to high or low - wrong edge preparation	- see recommendations - see recommendations - see recommendations

5. insert breakage

possible cause	solution
- improper clamping	- use carbide chip-breaker to distribute clamping forces. - do not over torque the lock pin when using Kenloc inserts (12 in/lbs maximum)
- scale and/or heavy interruptions on workpiece	- increase insert thickness and/or edge preparation
- thermal shock	- turn off coolant

Ceramic application recommendations for carbon, alloy, tool and stainless steels

work piece material	machining conditions	finishing & semi-rough, turn & bore		*roughing - turning, boring	milling
free machining carbon steels AISI 1100 and 1200 series machinability index 80 - 100 hardness 140 - 190 BHN	depth of cut (in.)	up to .060	up to .150	up to .300	up to .150
	feed rate (ipr)	.005 - .010	.007 - .015	.010 - .030	(ipt) .003 - .006
	insert geometry	positive & negative Kendex negative Kenloc		- - -	negative Kendex
	grade	surface speeds (sfm)			
	K060 K090	1000 - 3000 1000 - 3000		- - - - - -	1200 - 3000

*See footnote on p. 477

Ceramic technical data

Ceramic application recommendations for carbon, alloy, tool and stainless steels (cont.)

work piece material	machining conditions	finishing & semi-rough, turn & bore		*roughing - turning, boring	milling
plain carbon steel AISI 1000 series machinability index 80 - 100 hardness 180 - 240 BHN	depth of cut (in.)	up to .060	up to .150	up to .300	up to .150
	feed rate (ipr)	.005 - .010	.007 - .015	.010 - .030	(ipt) .003 - .006
	insert geometry	positive & negative Kendex negative Kenloc		- - -	negative Kendex
	grade	surface speeds (sfm)			
	K060 K090	1000 - 1500 1000 - 1500		- - - - - -	1200 - 2000
alloy steels AISI 1300, 4000, 5000 6000, 8000 & 9000 series machinability index 80 - 100 hardness 190 - 330 BHN 330 - 450 BHN 450 - 700 BHN	K090 K090 K090	500 - 1500 500 - 1200 300 - 700		- - - - - - - - -	800 - 2000 700 - 1800 450 - 1500
tool steels wrought high speed, shock resistant, hot & cold worked tool steels machinability index 40 - 60 hardness 200 - 330 BHN 330 - 450 BHN 450 - 700 BHN	K090 K090 K090	500 - 1500 500 - 1200 300 - 700		- - - - - - - - -	800 - 2000 700 - 1500 400 - 700
stainless steels Austenitic stainless 200 - 300 series machinability index 35 - 50 hardness 140 -190 BHN	K090	600 - 1200		- - -	900 - 1800
martensitic & ferritic stainless 400 - 500 & PH series machinability index 45 - 55 hardness 175 - 210 BHN	K090	650 - 1500		- - -	900 - 2000

* A special consideration; K090 roll turning inserts, because of their large size, thickness, and special K- lands, can handle feed rates and depths of cut not normally associated with ceramic applications.

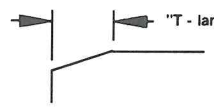
Ceramic technical data

Ceramic application considerations for successful machining of cast irons

1. edge preparation

When turning and boring:
feed rate (ipr) = 1 - 3 x leg length (L) of T- land.
example: feed rate = 1-3 x (.008) [std. Kyon 3000 leg length]
feed rate = **.008-.024 ipr**

When milling:
feed rate (ipt) = .75 - 1.5 x leg length (L) of T- land
example: feed rate = .75-1.5 x (.008) [std. Kyon 3000 leg length]
feed rate = **.006-.012 ipt**



NOTE: standard edge preparation for Kyon 3000, K090 and K060 is a .008 x 20° T-land. The need may arise for special or non-stock edge preparations. Please see page 486 for edge preparation specifications and recommendations.

2. coolant

Coolant is not recommended when using K060 or K090 for cast iron applications. However, coolant can safely be used with Kyon 3000 in cast iron applications and can be an effective means of dust control.

3. edge chipping

possible cause	solution
- wrong grade for application - speed to low - feed to high - incorrect edge prep - vibrations	- see recommendations - see recommendations - see recommendations - see recommendations - check rigidity of tool & workpiece; reduce tool and/or spindle overhang.
- interruptions in cut	- increase edge prep. - increase speed - decrease feed - use thicker insert - verify rigidity as above

4. excessive wear

possible cause	solution
- wrong grade for application - speed to high or low - wrong edge preparation	- see recommendations - see recommendations - see recommendations

5. insert breakage

possible cause	solution
- improper clamping	- use carbide chip-breaker to distribute clamping forces. - do not over torque the lock pin when using Kenloc inserts (12 in/lbs maximum)
- scale and/or heavy interruptions on workpiece	- increase insert thickness and/or edge preparation
- thermal shock	- turn off coolant

Ceramic application recommendations for cast iron

work piece material	machining conditions	finishing	roughing	heavy roughing	milling
cast irons **grey cast iron** and all other cast irons that produce a discontinuous chip machinability index 68 - 78	depth of cut (in.)	up to .150	up to .500	up to .500	up to .150
	feed rate (ipr)	.005 - .020	.015 - .030	up to .040	(ipt) .003 - .012
	insert geometry	positive and negative Kendex and Kenloc		neg. Kendex and Kenloc	pos. / neg. Kendex &Kenloc
	grade	surface speeds (sfm)			
hardness 190 - 330 BHN	K060 K090 Kyon 3000	1000-3500 1000-3500 600-3500	1000-3500 600-3500	600 - 3500	800 - 4500
330 - 450 BHN	K090 Kyon 3000	700-2000 600-2000	700-2000 600-2000	600 - 2000	1000 - 3000
450 - 700 BHN	K090 Kyon 3000	500-1200 500-1500	500-2000 500-1500	500 - 1500	500 - 1500

Ceramic technical data

Ceramic application recommendations for cast iron (cont.)

work piece material	machining conditions	finishing	roughing	heavy roughing	milling
cast irons **alloy and ductile cast irons** that produce a curled chip hardness 140 - 260 BHN	depth of cut (in.)	up to .150	up to .500	up to .500	up to .150
	feed rate (ipr)	.005 - .020	.015 - .030	up to .040	(ipt) .003 - .012
	insert geometry	positive and negative Kendex and Kenloc		neg. Kendex and Kenloc	pos. / neg. Kendex &Kenloc
	grade	surface speeds (sfm)			
	K060	1000 - 2000			
	K090	1000 - 2000	1000 - 2000		
	Kyon 3000		800 - 1500	800 - 1500	1000 - 3000

Cermet technical data

considerations for successful application of Kennametal TiC /TIN Cermets

1. edge preparation

The standard edge preparations for Kennametal cermets are as follows;

insert style	grade	edge prep.
Kenloc chip control inserts	KT125 KT150 KT175	.001 - .002 hone
positive and negative Kendex turning inserts	KT125 KT150	.002 - .004 x 20º T- land
milling geometries	KT150 KT175	I.C.< 1/2" .003 - .005 x 20º I.C. ≥ 1/2" .005 - .007 x 20º

NOTE: the need may arise for special or non–stock edge preparations. Please see page 486 for edge preparation specifications and recommendations.

2. workpiece materials

Carbon, alloy, stainless steels and malleable (ductile) irons are the primary materials for TiC/TIN applications. [Free machining aluminum and non-ferrous alloys (copper, zinc, and brass) can also be machined successfully.] Note that these are all materials which machine with a ductile chip; to take full advantage of the increased chemical stability that Kennametal cermets offer.

Grey cast iron is <u>not</u> a primary workpiece material for cermets. The abrasion resistance required makes silicon nitride and alumina ceramics a more suitable chioce as a cutting tool for grey cast iron.

3. workpiece material hardness - 40 Rc max.

TiC / TIN cermets have metallic binders, and as such do not exhibit the high hot hardness necessary for successful machining of hardened workpiece materials. >40 Rc is better suited for ceramic or CBN cutting tool materials.

4. speeds and feeds

Kennametal cermets do offer potential <u>increased productivity</u> vs. uncoated and coated carbides. This increased productivity will mostly be due to increased speeds.

workpiece material	speed potential (sfm) vs. uncoated carbide
carbon, stainless steels ductile irons	+30% - 50% (approx.)
alloy steels	+20% (approx.)

Cermets are more feed and fracture sensitive than uncoated or coated carbide. The current practical upper feed limit for a tough cermet (KT175) is approximately .022 - .025 ipr turning carbon steel.

The majority of successful cermet applications (turning, boring and milling) will be at conventional carbide speeds and feeds. The criteria for success will be <u>extended tool life</u>, superior surface finish and economics of application.

5. coolant

TiC / TIN cermets are more sensitive to thermal shock than uncoated or coated carbide. Therefore, as a general rule, coolant is not recommended when rough turning, boring,or milling.

There are cases however,as in very light finish turning and boring, when the use of coolant can be beneficial to workpiece surface finish.

6. edge chipping

possible cause	solution
- wrong grade for application	- see recommendations
- speed to low	- see recommendations
- feed to high	- see recommendations
- incorrect edge prep	- see recommendations
- vibrations	- check rigidity of tool and workpiece; reduce tool and/or spindle overhang.
- interruptions in cut	- increase edge prep - increase speed - decrease feed - use thicker insert - verify rigidity as above

7. excessive wear

possible cause	solution
- wrong grade for application	- see recommendations
- speed to high or low	- see recommendations
- incorrect edge prep	- see recommendations

8. insert breakage

possible cause	solution
- improper clamping	- do not over torque the lock pin when using Kenloc inserts. (12 in/lbs maximum)
- scale and/or heavy interruptions on workpiece	- increase insert thickness and/or edge preparation.
- thermal shock	- <u>turn coolant off</u>

Cermet technical data

application recommendations

		grade	surface speed (sfm)	depth of cut (doc)	feed rate (in. / rev.)
free machining and plain carbon steels 140 - 240 BHN	finishing	KT125	300 - 1500	.100 max.	.015 max.
	semi-finish/ light roughing	KT150 / KT175	150 - 1000	.150 max.	.020 max.
	milling	KT175	300 - 1000	as required	.002 - .010 ipt
alloy steels -190 - 330 BHN	finishing	KT125	300 - 1200	.100 max.	.015 max.
	semi-finish/ light roughing	KT150 / KT175	150 - 800	.150 max.	.020 max.
	milling	KT175	300 - 800	as required	.002 - .010 ipt
alloy steels 330 - 400 BHN	finishing	KT125	300 - 800	.100 max.	.015 max.
	semi-finish/ light roughing	KT150 / KT175	100 - 500	.150 max.	.020 max.
	milling	KT175	200 - 600	as required	.002 - .010 ipt
stainless steel austenitic 200 - 300 series 140 - 190 BHN	finishing	KT125	300 - 1200	.100 max.	.015 max.
	semi-finish/ light roughing	KT150 / KT175	300 - 1000	.150 max.	.015 max.
	milling	KT175	300 - 1000	as required	.002 - .010 ipt
stainless steel martensitic 400 - 500 PH series 175 - 210 BHN	finishing	KT125	600 - 1000	.100 max.	.015 max.
	semi-finish/ light roughing	KT150 / KT175	600 - 800	.150 max.	.015 max.
	milling	KT175	400 - 800	as required	.002 - .010 ipt
alloy cast iron (ductile, malleable)	finishing	KT125	300 - 1500	.125 max.	.015 max.
	semi-finish/ light roughing	KT150 / KT175	200 - 1200	.250 max.	.020 max.
	milling	KT175	300 - 1500	as required	.002 - .015 ipt
free machining aluminum and non ferrous alloys (copper, zinc,brass)	finishing	KT125	1000 - 4000	.125 max.	.015 max.
	semi-finish/ light roughing	KT150 / KT175	700 - 4000	.250 max.	.020 max.
	milling	KT175	500 - 3000	as required	.002 - .015 ipt

Polycrystalline technical data

;ecommendations for successful application of Kennametal Polycrystalline diamond and CBN tools

1. edge preparation

The standard edge preparations for Kennametal polycrystalline inserts are as follows;

insert style	grade	edge prep.
positive Kendex	all KD100 PCD grades	sharp
negative Kendex	KD100 KD120 KD050 KD220 KD200	sharp .001 -.002 hone .004 x 20° T-land .004 x 20° T-land .008 x 20° T-land & .001 -.002 hone
negative Kenloc	KD100 KD120 KD050 KD220	sharp .001 - .002 hone .004 x 20° T-land .004 x 20° T-land
Top Notch threading and grooving	KD100	sharp
Top Notch profiling	KD120	.001 - .002 hone
positive screw-on mini screw-on brazed tools	all KD100 PCD grades	sharp

NOTE: the need may arise for special or non-stock edge preparations. See page 486 for edge preparation specifications and recommendations

2. tool geometries

When applying KD100 (PCD) in alunimum or other nonferrous applications, it is beneficial to use a positive tool geometry to minimize cutting pressure and eliminate built-up edge.

When applying KD050, KD120, KD220 or KD200 to machine hardened ferrous alloys, it is beneficial to use a negative tool geometry to maximize insert strength, stabilize the cutting edge and improve tool life. This can be valid even in situations where the depth of cut is very low (< .010).

3. workpiece material hardness (CBN applications)

When machining hardened ferrous materials with Kennametal CBN s, maximum economic benefits can be realized if the workpiece hardness is 45 - 65 Rc. An exception to this is the sintered irons which are relatively soft (200 - 250 BHN). The abrasiveness of these irons however, warrant the use of CBN tools.

4. coolant

The use of flood coolant is strongly recommended when applying KD050, KD120 and KD220. Coolant is not recommended when applying KD200 (solid CBN), nor is it necessary when applying KD100 in nonferrous applications.

5. edge chipping

possible cause	solution
- wrong grade for application	- see recommendations
- speed to low	- see recommendations
- feed to high	- see recommendations
- incorrect edge prep.	- see recommendations
- vibrations	- check rigidity of tool and workpiece; reduce tool and/or spindle overhang
- interruptions in cut	- increase edge prep. - increase speed - decrease feed - use thicker insert - verify rigidity as above

6. excessive wear

possible cause	solution
- wrong grade for application	- see recommendations
- speed to high or low	- see recommendations
- incorrect edge prep.	- see recommendations

7. insert breakage

possible cause	solution
- improper clamping	- do not over torque the lock pin when using Kenloc inserts (12 in/lbs max.)
- scale and/or heavy interruptions on workpiece	- increase insert thickness and/or edge preparation

8. Polycrystalline tip dislodged

possible cause	solution
- braze wetting between carbide and polycrystalline tip.	- flood coolant to tip of insert - lower speed; see recommendations.

Polycrystalline diamond technical data

application recommendations

non-ferrous materials		grade	surface speed (sfm)	depth of cut (doc)	feed rate in / rev.
free machining aluminim alloys 80 - 120 BHN	finishing	KD100	2000-4000	to .060"	.005-.010
	roughing	KD100	1800-3500	to .125"	.008-.020
	milling	KD100	1000-10,000	to .125"	.003-.010 ipt
high silicon aluminum hypereutectic	finishing	KD100	2000-3000	to .060"	.005-.010
	roughing	KD100	1200-2500	to .125"	.008-.020
	milling	KD100	1200-3000	to .125"	.003-.010 ipt
nonferrous free machining alloys copper, zinc, brass alloys 80 - 120 BHN	finishing	KD100	1200-3500	to .060"	.005-.010
	roughing	KD100	1200-3000	to .125"	.008-.020
	milling	KD100	1800-3500	to .125"	.003-.010 ipt
non- metallics nylons, acrylics, phenolic resin materials	finishing	KD100	2000-4500	to .060"	.005-.010
	roughing	KD100	550-2500	to .125"	.008-.020
	milling	KD100	1000-4500	to .125"	.003-.010 ipt

Polycrystalline CBN technical data
application recommendations

hardened steels

		grade	surface speed (sfm)	depth of cut (doc)	feed rate in/rev.
cold work tool steel 570-750 BHN 55-65 Rc	finishing	KD050	250 - 500	.050" max.	.002 - .008
	semi-finishing / light roughing	KD120 KD220	200 - 450 200 - 350	.100" max. .125" max.	.002 - .010 .002 - .012
	rough turning	KD200	200 - 350	.150" max.	.002 - .012
	milling		500 - 700	.150" max.	.006 - .010 (ipt)
high-speed steel 570-750 BHN 55-65 Rc	finishing	KD050	250 - 500	.050" max.	.002 - .008
	semi-finishing / light roughing	KD120 KD220	200 - 450 200 - 350	.100" max. .125" max.	.002 - .010 .002 - .012
	roughing	KD200	200 - 350	.150" max.	.002 - .012
bearing steel 570-750 BHN 55-65 Rc	finishing	KD050	250 - 500	.050" max.	.002 - .008
	semi-finishing / light roughing	KD120	200 - 450	.100" max.	.002 - .010
	roughing	---	---	---	---
hot work die steel 485-650 BHN 50-60 Rc	finishing	KD050	250 - 500	.050" max.	.002 - .008
	semi-finishing / light roughing	KD120	200 - 450	.100" max.	.002 - .010
	roughing	---	---	---	---
surface hardened steel 570-750 BHN 56-65 Rc	finishing	KD050	250 - 500	.050" max.	.002 - .008

Polycrystalline CBN technical data
application recommendations

hardened cast irons

		grade	surface speed (sfm)	depth of cut (doc)	feed rate in/rev.
white chilled iron 485-650 BHN 50-60 Rc	finishing	KD120	250 - 450	.060" max.	.002 - .006
	semi-finishing / light roughing	KD220	200 - 350	.100" max.	.005 - .010
	rough turning	KD200	200 - 350	.150" max.	.008 - .016*
	milling		500 - 1000	.150" max.*	.006 - .015 (ipt)*
high chromium iron 570-750 BHN 55-65 Rc	finishing	KD120	100 - 200	.050" max.	.002 - .005
	semi-finishing / light roughing	KD220	100 - 200	.080" max.	.002 - .008
	roughing	KD200	100 - 200	.125" max.	.005 - .010
Ni Hard 570-750 BHN 55-65 Rc	finishing	KD120	125 - 250	.050" max.	.002 - .006
	semi-finishing / light roughing	KD220	125 - 250	.080" max.	.002 - .008
	roughing	KD200	125 - 250	.125"*	.005 - .015*
	milling		650 - 1000		.008 - .012 (ipt)*
sintered Iron 200-250 BHN	finishing	KD120	800 - 1000	.050" max.	.002 - .006
	semi-finishing / light roughing	KD220	700 - 1000	.050" max.	.002 - .006

* Use round chamfered inserts.

high temperature alloys

high temperture nickle base alloys 325-500 BHN 35-50 Rc	finishing	KD120	400 - 600	.080" max.	.003 - .007
high temerature cobalt base hard facing alloys 570-750 BHN 55-65 Rc	finishing	KD120	600 - 800	.050" max.	.004 - .008
	semi-finishing / light roughing	KD220	600 - 800	.050" max.	.004 - .010
	roughing	KD200	600 - 800	.050" max.	.004 - .012

technical data

edge preparation

Ceramics, cermets and polycrystalline cutting tool materials have a relatively high material hardness. Because of this, the cutting edge is more brittle than the conventional carbide metal cutting tools.

In order to optimize the performance of these advanced cutting tool materials, it is critical that the edge preparation be matched to the cutting tool material, the workpiece material and the machining operation being performed.

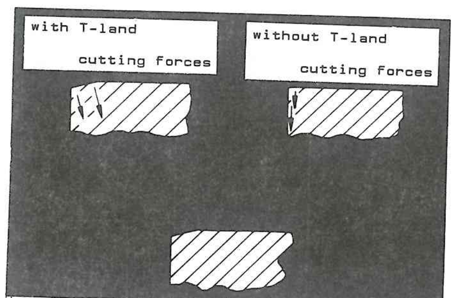
General recommendations for standard edge preparations by cutting tool material and machining conditions have been given in each of the three sections of this catalog.

There are four basic choices when considering edge preparation for advanced materials:

1. **T - land**
2. **hone**
3. **T - land plus hone**
4. **up - sharp**

1. T - land

T- lands act to protect the cutting edge by eliminating a sharp cutting edge, thereby reducing edge chipping, and by redistributing the cutting forces back through the larger body cross section of the insert which makes the insert stronger.



typical T- lands	
angle 'A'	width 'W'
20° / 25°	.002 - .004 .005 - .007 .007 - .009 .012 - .014
30° / 35°	.005 - .007 .008 - .010 .012 - .014
10° / 15°	.014 - .016 .030 - .032 .060 - .062

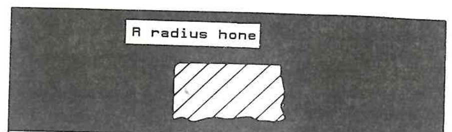
There is a trade off. Increasing either the width 'W' or the angle 'A' of the T- land to strengthen the insert will also increase the cutting forces and could have a potentially detrimental effect on both the cutting tool and/or the workpiece material.

The optimum situation is to have the minimum edge preparation (in terms of T- land size and angle) which will perform the desired operation, and protect the insert cutting edge adequately, thereby achieving satisfactory tool life.

2. hone

Hones protect the cutting edge by eliminating the sharp edge and reducing edge chipping. Hones do not provide the high level of edge protection of a T- land. However, they can be very satisfactory when using ceramics, cermets or polycrystalline inserts in finishing operations, such as when depth of cut and feed rates are kept at a minimum, and minimum cutting pressure is desired.

Kennametal recommends the following range of radius type hones:

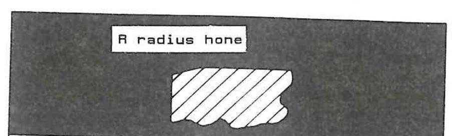


hone designation	'R' radius
A	.001 - .002
B	.003 - .004
C	.005 - .006

3. T- land plus hone

In some situations, like rough turning steel with ceramics, minute chipping can occur at the intersections of the T- land and the rake surface or flank surface of the insert. This condition can be corrected by applying a radius hone to both intersections. This can effectively improve the performance of the edge preparation while keeping T- land size and angle constant.

Kennametal recommends the following modifications:



hone designation	'R' radius
A	.001 - .002
B	.003 - .004
C	.005 - .006

technical data

edge preparation

4. up - sharp

Normally, Kennametal does not recommend up-sharp edges for ceramic, cermet or polycrystalline inserts. The high material hardness results in an edge that is highly susceptable to chipping if left 'up-sharp'.

However, if it has been determined that an up-sharp edge is necessary, it can be ordered as a special.

how to order an insert with a non-standard edge preparation:

When ordering any Kennametal advanced cutting tool material with a non-standard edge preparation, please detail the type and size of edge preparation desired.

ex: CNGA-432T KY3000 with .012 x 30° T- land

When considering special edge preparations, please contact your local Kennametal Tooling Systems Engineer. He is uniquely qualified to make recommendations to "fine tune" your advanced cutting tool applications, and to help you maximize your metalcutting productivity.

metalcutting safety

Modern metalcutting operations involve high energy, high spindle or cutter speeds, and high temperatures and cutting forces. Hot, flying chips may be projected from the workpiece during metalcutting. Although advanced cutting tool materials are designed and manufactured to withstand the high cutting forces and temperatures that normally occur in these operations, they are susceptible to fragmentating in service, particularly if they are subjected to over-stress, severe impact or otherwise abused. Therefore, precautions should be taken to adequately protect workers, observers and equipment against hot, flying chips, fragmented cutting tools, broken workpieces or other similar projectiles. Machines should be fully guarded and personal protective equipment should be used at all times.

When grinding advanced cutting tool materials, a suitable means for collection and disposal of dust, mist or sludge should be provided. Overexposure to dust or mist containing metallic particles can be hazardous to health particularly if exposure continues over an extended period of time and may cause eye, skin and mucous membrane irritation and temporary or permanent respiratory disease. Certain existing pulmonary and skin conditions may be aggravated by exposure to dust or mist. Adequate ventilation, respiratory protection and eye protection should be provided when grinding and workers should avoid breathing of and prolonged skin contact with dust or mist. General Industry Safety and Health Regulations, Part 1910, U.S. Department of Labor, published in Title 29 of the Code of Federal Regulations should be consulted. Read the applicable Material Safety Data Sheet before grinding.

Cutting tools are only one part of the worker-machine-tool system. Many variables exist in machining operations, including the metal removal rate; the workpiece size, shape, strength and rigidity; the chucking or fixturing; the load carrying capability of centers; the cutter and spindle speed and torque limitations; the holder and boring bar overhang; the available power; and the condition of the tooling and the machine. A safe metalcutting operation must take all of these variables, and others, into consideration.

Kennametal has no control over the end use of its products or the environment into which those products are placed. Kennametal urges that its customers adhere to the recommended standards of use of their metalcutting machines and tools, and that they follow procedures that ensure safe metalcutting operations.

For more information, we suggest you write for Kennametal's Metalcutting Safety booklet Number A86-69(10)E6, if you do not already have one. Quantities are available, free, for distribution to your operating personnel.

Tool Design Data Formula—thrust reaming—boring—core drilling

Reaming, Boring & Core Drilling Thrust

Thrust = 160,000 · K · FPR · (Width of cut)

$$\text{Thrust} = 160{,}000 \cdot K \cdot FPR \cdot \left(\frac{D\text{-}d}{2}\right) \times 1.5 \text{ (dull tool factor)}$$

Material	*K*	Material	*K*
Dow Metal	.20	1320 Steel	1.00
Bearing Bronze	.40	6140 Steel	1.00
Alum. Alloy	.48	1020 Steel	1.05
Cast Iron	.80	2320 Steel	1.05
Malleable C.I.	.90	3120 Steel	1.05
1112 Steel	.90	2345 Steel	1.10
1315 Steel	.95	1095 Steel	1.40
1120 Steel	.96	Ann Copper	1.15

Note: Use this formula when tool has lead angle.

Spotface and Counterboring Thrust

Thrust = 232,000 · K · FPR · (Width of cut)

$$\text{Thrust} = 232{,}000 \cdot K \cdot FPR \cdot \left(\frac{D\text{-}d}{2}\right) \times 1.5 \text{ (dull tool factor)}$$

Material	Brinell	K (Thrust)
Mag. H-Alloy	63	.27
Alum. Alloy 24 ST	154	.82
Alum. Alloy 2S	37	2.70
Leaded Brass	124	.21
Phos. Bronze	156	.82
Monel Metal	207	1.36
C.I. (40,000 PSI)	210	.71
1020 Steel C.D.	180	1.00
6150 Steel H.R.	195	1.72
6150 Steel H.T.	241	2.09
1095 Steel Ann.	156	3.04

Note: Use this formula when tool has no lead angle such as Spotface tool.

Tool Design Data—Formula-horsepower tapping

Tapping

$$\left(\frac{1/N \cdot RPM}{10}\right) \times \text{HP from chart} \times 1.3^{*} = \text{HP}$$

N = No. Thd's per inch

HP required for tapping various materials (approx.)

(to be use for estimating purposes only)

Note—tap drill sizes to give approx. 75% full thread

Tap Size	rpm	Drill size Frac.	Drill size Dec.	HP for various materials based on a feed of 10″ per minute: Class 1	Class 2	Class 3	Class 4	Class 5	Class 6
1/4 — 20	200	13/64	0.203	0.058	0.086	0.113	0.140	0.165	0.189
1/4 — 28	280	7/32	0.219	0.043	0.064	0.084	0.104	0.122	0.140
5/16 — 18	180	F	0.257	0.082	0.121	0.160	0.198	0.232	0.266
5/16 — 24	240	I	0.272	0.064	0.094	0.125	0.155	0.181	0.207
3/8 — 16	160	5/16	0.312	0.112	0.165	0.218	0.270	0.317	0.363
3/8 — 24	240	Q	0.332	0.078	0.114	0.151	0.188	0.219	0.251
7/16 — 14	140	U	0.368	0.150	0.221	0.292	0.362	0.426	0.486
7/ 16 — 20	200	W	0.386	0.109	0.161	0.212	0.263	0.308	0.353
1/2 — 13	130	27/64	0.422	0.185	0.272	0.360	0.447	0.523	0.600
1/2 — 20	200	29/64	0.453	0.125	0.184	0.243	0.302	0.353	0.405
9/16 — 12	120	31/64	0.484	0.227	0.334	0.441	0.548	0.641	0.735
9/16 — 18	180	33/64	0.516	0.157	0.231	0.305	0.379	0.444	0.508
5/8 — 11	110	17/32	0.531	0.275	0.405	0.534	0.664	0.777	0.890
5/8 — 18	180	37/64	0.578	0.176	0.259	0.342	0.425	0.497	0.570
3/4 — 10	100	21/32	0.656	0.366	0.539	0.711	0.883	1.034	1.184
3/4 — 16	160	11/16	0.687	0.237	0.349	0.460	0.572	0.669	0.767
7/8 — 9	90	49/64	0.766	0.477	0.702	0.926	1.151	1.347	1.544
7/8 — 14	140	13/16	0.812	0.317	0.467	0.616	0.765	0.895	1.026
1 — 8	80	7/8	0.875	0.614	0.903	1.192	1.481	1.734	1.987
1 — 14	140	15/16	0.937	0.327	0.481	0.635	0.789	0.924	1.058
1-1/8 — 7	70	63/64	0.984	0.780	1.146	1.513	1.879	2.20	2.521
1-1/8 — 12	120	1-3/64	1.047	0.518	0.762	1.006	1.250	1.463	1.676
1-1/4 — 7	70	1-7/64	1.109	0.874	1.286	1.697	2.108	2.468	2.828
1-1/2 — 6	60	1-11/32	1.343	1.238	1.821	2.404	2.986	3.500	4.00
1-1/2 — 12	120	1-27/64	1.422	0.700	1.030	1.359	1.689	1.977	2.265

Note: HP values for all other materials can be based on machinability value for tapping.
*Add 30.5% to horsepower for dull tools and machine friction.
Select class designation from following sheet
Example 1/2–13 tap 1040 material

$$\left(\frac{1/13 \cdot 425}{10}\right) \times 0.523 \times 1.3 = 2.225 \text{ HP}$$

Tool Design Data
Formula—Horsepower classification of material

Classifications of materials for computing horsepower required for tapping

Class 1	Class 4
Aluminum—Rod; Cast; Die-Cast Brass—Bar; Cast Zinc—Die-Cast	Iron—Malleable Steel—1010; 1035; 1112 X-1340 Cast
Class 2	**Class 5**
Brass—Stamping; tubing Bronze, Phosphor; Tubing; Cast Bar; Maganese Cast Aluminum Copper; German Silver	Iron, Wrought Steel—1040; 1095 T-1330–T-1350; Stamping
Class 3	**Class 6**
Brass—Forging Bronze—Naval Iron—Cast; Nickel Steel—Semi; Casting	Monel Metal Steel—5120–5210 6115–6195 4130–4820 2015–2515 3115–3450 Forging; Nitralloy Stainless—Free

Determining Power Requirements in Machining†

TABLE 17.2-1 Shop Formulas for Turning, Milling, Drilling and Broaching—English Units

PARAMETER	TURNING	MILLING	DRILLING	BROACHING
Cutting speed, fpm	$V_c = .262 \times D_t \times rpm$	$V_c = .262 \times D_m \times rpm$	$V_c = .262 \times D_d \times rpm$	V_c
Revolutions per minute	$rpm = 3.82 \times \frac{V_c}{D_t}$	$rpm = 3.82 \times \frac{V_c}{D_m}$	$rpm = 3.82 \times \frac{V_c}{D_d}$	—
Feed rate, in/min	$f_m = f_r \times rpm$	$f_m = f_t \times n \times rpm$	$f_m = f_r \times rpm$	—
Feed per tooth, in	—	$f_t = \frac{f_m}{n \times rpm}$	—	f_t
Cutting time, min	$t = \frac{L}{f_m}$	$t = \frac{L}{f_m}$	$t = \frac{L}{f_m}$	$t = \frac{L}{12 V_c}$
Rate of metal removal, in^3/min	$Q = 12 \times d \times f_r \times V_c$	$Q = w \times d \times f_m$	$Q = \frac{\pi D^2}{4} \times f_m$	$Q = 12 \times w \times d_t \times V_c$
Horsepower required at spindle*	$hp_s = Q \times P$	$hp_s = Q \times P$	$hp_s = Q \times P$	—
Horsepower required at motor*	$hp_m = \frac{Q \times P}{E}$	$hp_m = \frac{Q \times P}{E}$	$hp_m = \frac{Q \times P}{E}$	$hp_m = \frac{Q \times P}{E}$
Torque at spindle	$T_s = \frac{63030\ hp_s}{rpm}$	$T_s = \frac{63030\ hp_s}{rpm}$	$T_s = \frac{63030\ hp_s}{rpm}$	—

SYMBOLS:
- D_t = Diameter of workpiece in turning, inches
- D_m = Diameter of milling cutter, inches
- D_d = Diameter of drill, inches
- d = Depth of cut, inches
- d_t = Total depth per stroke in broaching, inches
- E = Efficiency of spindle drive
- f_m = Feed rate, inches per minute
- f_r = Feed, inches per revolution
- f_t = Feed, inches per tooth
- hp_m = Horsepower at motor
- hp_s = Horsepower at spindle
- L = Length of cut, inches
- n = Number of teeth in cutter
- P = Unit power, horsepower per cubic inch per minute
- Q = Rate of metal removed, cubic inches per minute
- rpm = Revolutions per minute of work or cutter
- T_s = Torque at spindle, inch-pounds
- t = Cutting time, minutes
- V_c = Cutting speed, feet per minute
- w = Width of cut, inches

*Unit power data are given in table 17.2-3 for turning, milling and drilling, and in figure 17.2-1 for broaching.

†Reprinted by permission from *Machining Data Handbook*, 3rd edition, © 1980, Metant Research Associates, Inc.

Determining Power Requirements in Machining†

TABLE 17.2–3 Average Unit Power Requirements for Turning, Drilling and Milling—English Units

MATERIAL	HARDNESS	UNIT POWER° hp/in³/min					
		TURNING P_t HSS AND CARBIDE TOOLS (feed .005-.020 ipr)		DRILLING P_d HSS DRILLS (feed .002-.008 ipr)		MILLING P_m HSS AND CARBIDE TOOLS (feed .005-.012 ipt)	
	Bhn	Sharp Tool	Dull Tool	Sharp Tool	Dull Tool	Sharp Tool	Dull Tool
STEELS, WROUGHT AND CAST: Plain Carbon, Alloy Steels, Tool Steels	85-200	1.1	1.4	1.0	1.3	1.1	1.4
	35-40 R_c	1.4	1.7	1.4	1.7	1.5	1.9
	40-50 R_c	1.5	1.9	1.7	2.1	1.8	2.2
	50-55 R_c	2.0	2.5	2.1	2.6	2.1	2.6
	55-58 R_c	3.4	4.2	2.6	3.2*	2.6	3.2
CAST IRONS: Gray, Ductile and Malleable	110-190	0.7	0.9	1.0	1.2	0.6	0.8
	190-320	1.4	1.7	1.6	2.0	1.1	1.4
STAINLESS STEELS, WROUGHT AND CAST: Ferritic, Austenitic and Martensitic	135-275	1.3	1.6	1.1	1.4	1.4	1.7
	30-45 R_c	1.4	1.7	1.2	1.5	1.5	1.9
PRECIPITATION HARDENING STAINLESS STEELS	150-450	1.4	1.7	1.2	1.5	1.5	1.9
TITANIUM	250-375	1.2	1.5	1.1	1.4	1.1	1.4
HIGH TEMPERATURE ALLOYS: Nickel and Cobalt Base	200-360	2.5	3.1	2.0	2.5	2.0	2.5
Iron Base	180-320	1.6	2.0	1.2	1.5	1.6	2.0
REFRACTORY ALLOYS: Tungsten	321	2.8	3.5	2.6	3.3*	2.9	3.6
Molybdenum	229	2.0	2.5	1.6	2.0	1.6	2.0
Columbium	217	1.7	2.1	1.4	1.7	1.5	1.9
Tantalum	210	2.8	3.5	2.1	2.6	2.0	2.5
NICKEL ALLOYS	80-360	2.0	2.5	1.8	2.2	1.9	2.4
ALUMINUM ALLOYS	30-150 500 kg	0.25	0.3	0.16	0.2	0.32	0.4
MAGNESIUM ALLOYS	40-90 500 kg	0.16	0.2	0.16	0.2	0.16	0.2
COPPER	80 R_B	1.0	1.2	0.9	1.1	1.0	1.2
COPPER ALLOYS	10-80 R_B 80-100 R_B	0.64 1.0	0.8 1.2	0.48 0.8	0.6 1.0	0.64 1.0	0.8 1.2

°Power requirements at spindle drive motor, corrected for 80% spindle drive efficiency.
* Carbide
†Reprinted by permission from *Machining Data Handbook*, 3rd edition, ©1980, Metcut Research Associates, Inc.

APPENDIX C

PROGRAMMING EXERCISES

Ideally, any part programming exercise should be supported by program proving and production of the component. Most of the inch and metric exercises that follow will fall within the capacity of equipment now generally available in educational institutions, and it is likely that most benefit will be gained if the exercises are completed with that equipment in mind.

To cater to students who do not have access to suitable equipment two imaginary machine types follow together with their control systems. These alternative systems are fairly typical and are illustrative of the way manufacturers summarize the capabilities of machines and control systems in their promotional literature and instruction manuals.

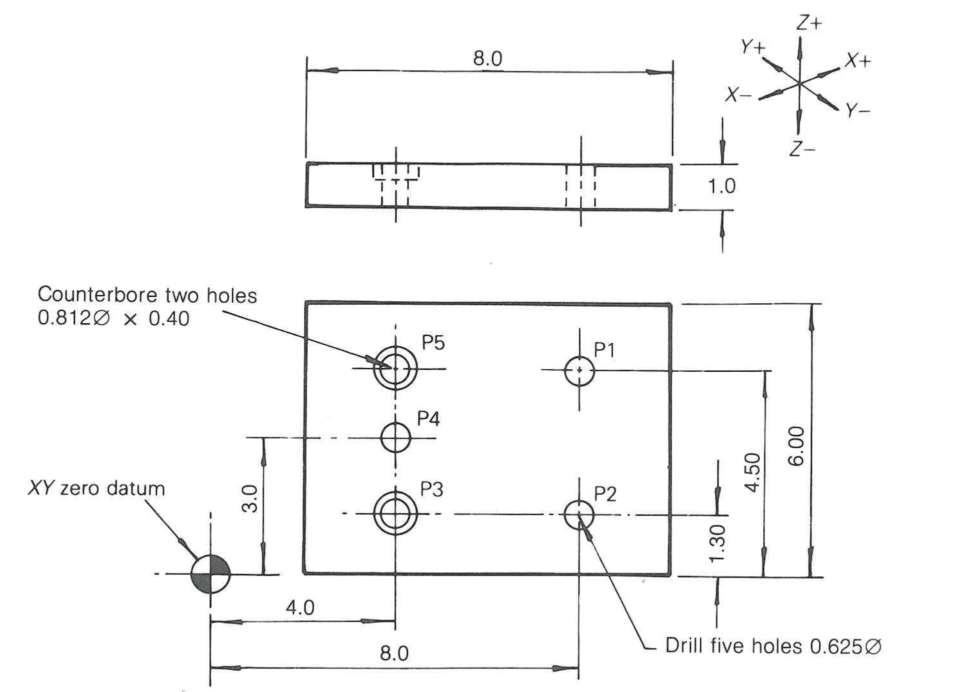
It is anticipated, if the exercises are attempted using the alternative systems, that, as when using college-based equipment, the student will receive further guidance and advice from his or her lecturer. Partially completed programs for Exercises 1, 2, 9, and 10 are included at the end of the chapter.

Details of the machining, tooling and programming requirements are included alongside each detail drawing with the exception of the last exercise of each type, where the intention is that the student should complete all stages from detail drawing to finished product, making decisions relating to work holding, tooling, feeds and speeds followed by part programming and, in cases where the programming has been related to equipment available, by program proving and final machining.

Appropriate speeds and feeds may be selected from the data given in Chapter 7, Appendix B, or from a *Machining Data Handbook*.

The components have been dimensioned according to traditional standards in some cases, and in others the method of dimensioning which is increasingly being favored for numerical control is used. Students will be able to judge for themselves which is the more appropriate conventional or coordinate.

The direction diagrams included on Exercises 1, 2, 9, and 10 indicate relative tool movement, that is, the directions shown will be those used in the part program. The diagrams relating to the turning exercise assume a rear mounted tool post.



Material: mild steel
Inch Programming
Exercise
(see example program 1)

EXERCISE NO.	1

Machining

Center drill and drill five holes 0.625∅ in sequence P1 to P5

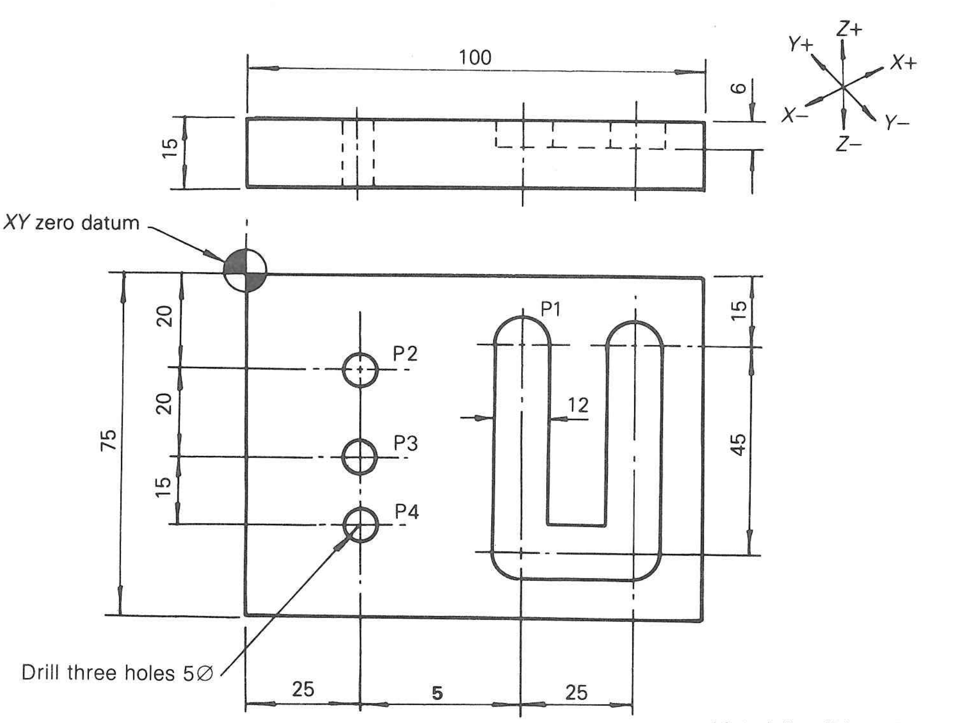
Counterbore two holes 0.812∅ × 0.40 deep in sequence P3 to P5

Tooling

HSS No. 2 center drill
HSS drill – .625∅
HSS counterbore .812∅

Programming

Absolute dimensions
Point-to-point positioning
Linear interpolation
Z datum clearance 0.100
Time dwell (to clear counterbore)
Flood coolant
Spindle speed rev/min
Feed inches
Allocate tool offsets
Do *not* use drill cycle



Material: mild steel
Metric Programming
Exercise
(see example program 2)

EXERCISE NO.	2

Machining

Mill 12 slot, P1 start

Center drill and drill three holes 5∅ in sequence P2 to P4

Tooling

HSS 12∅ end mill
HSS No. 2 center drill
HSS drill 5∅

Programming

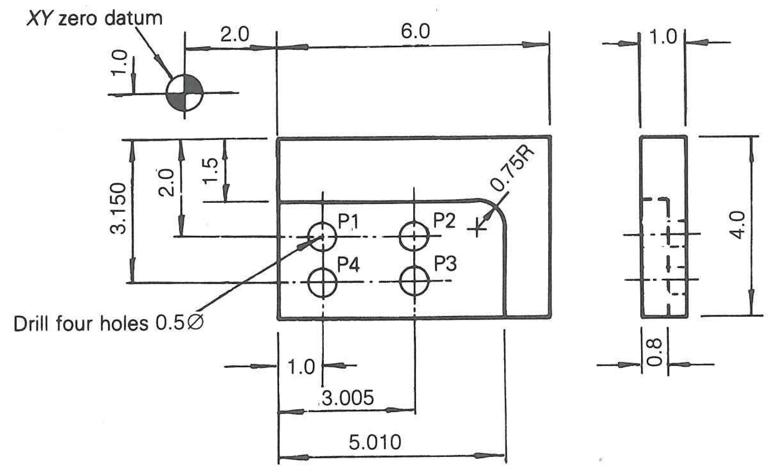
Incremental dimensions
Linear interpolation
Point-to-point positioning
Quill clamp (when milling slot)
Drill cycle
Spray mist coolant
Spindle speed rev/min
Feed mm/min
Allocate tool offsets
Z datum clearance 4mm

EXERCISE NO.	3

Machining
Mill 0.800 step in two passes
Center drill and drill four holes .5∅ sequence P1 to P4

Tooling
Tungsten carbide insert end mill, 1.5∅, four teeth
HSS No. 2 center drill
HSS drill 0.5∅

Programming
Absolute dimensions
Linear interpolation
Point-to-point positioning
Quill clamp (when milling steps)
Flood coolant
Drill cycle
Spindle speed rev/min
Feed inches/min
Allocate tool offsets
Z datum clearance 0.200



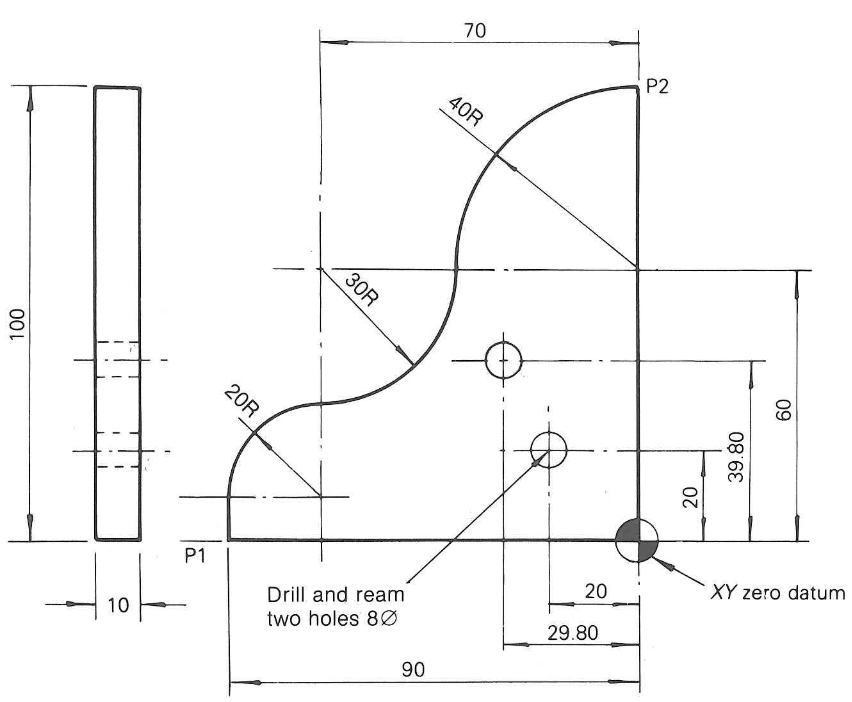
Material: brass
(Inch Programming Exercise)

EXERCISE NO.	4

Machining
Center drill, drill and ream two holes 8∅
Mill radial profile P1 to P2

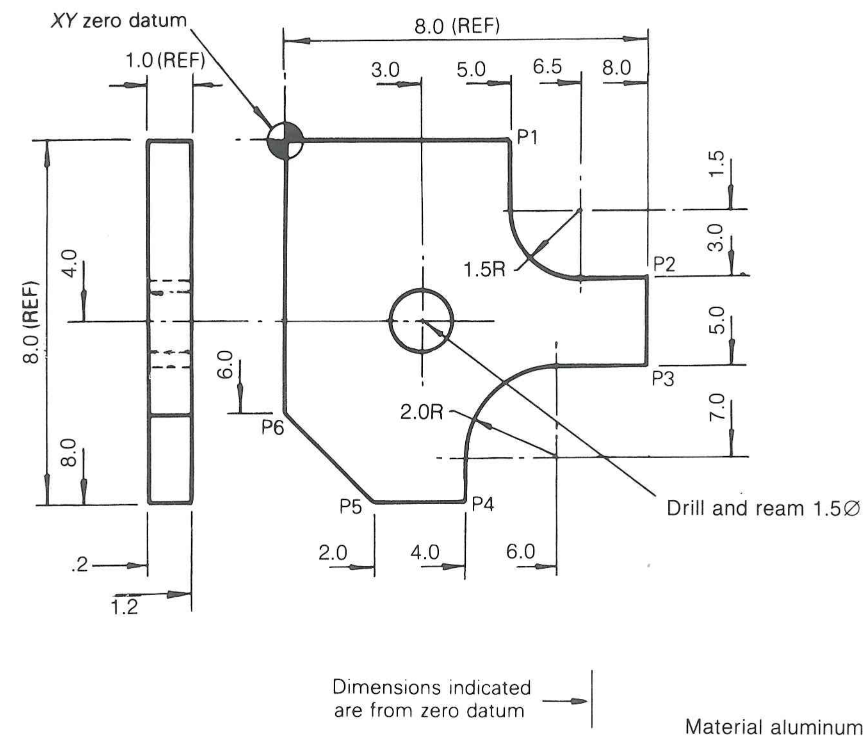
Tooling
HSS No. 2 center drill
HSS drill 7.5∅
HSS reamer 8∅
Cemented carbide insert end mill 20∅, three teeth

Programming
Absolute dimensions
Point-to-point positioning
Linear interpolation
Circular interpolation
Z datum clearance 2mm
Drill cycle
Ramp down inhibit
Cutter compensation
Flood coolant
Spindle speed rev/min
Feed mm/min
Allocate tool offsets



(Metric Programming Exercise)

Material: mild steel (flame-cut profile)



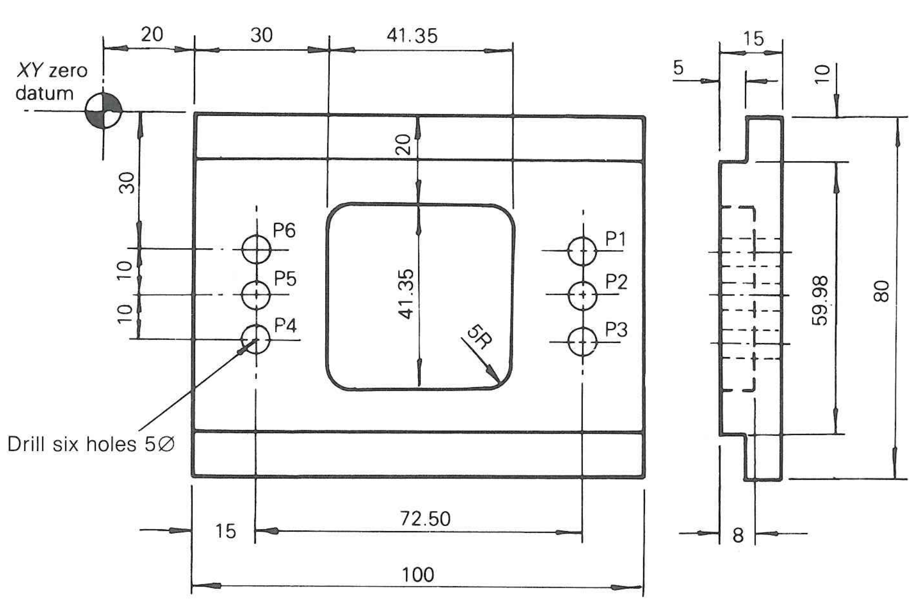
Material aluminum alloy (Inch Programming Exercise)

EXERCISE NO.	5

Machining
Drill and ream 1.5∅ hole
Rough mill corners and finish profile with continuous cut P1 to P6

Tooling
HSS No. 3 center drill
HSS drill 0.812∅
HSS drill 1.468∅
HSS reamer 1.5∅
Cemented carbide insert end mill 1.25∅ three teeth

Programming
Absolute dimensions
Z datum clearance 0.200
Linear interpolation
Circular interpolation
Cutter compensation
Spray mist coolant
Spindle speed rev/min
Feed inches/rev
Allocate tool offsets



Material: aluminum alloy (Metric Programming Exercise)

EXERCISE NO.	6

Machining
Mill top face
Mill 10 × 5 steps
Mill pocket
Drill six holes 5∅, P1 to P6

Tooling
Cemented carbide insert shell end mill, 60∅, five teeth
HSS end mill 10∅
HSS No. 2 center drill
HSS drill 5∅

Programming
Incremental dimensions
Z datum clearance 2mm
Linear interpolation
Point-to-point positioning
Cutter compensation
Pocket mill cycle
Quill clamp (steps and pocket)
Drill cycle
Spindle speed rev/min
Feed mm/min
Spray mist coolant
Allocate tool offsets

EXERCISE NO.	7

Machining

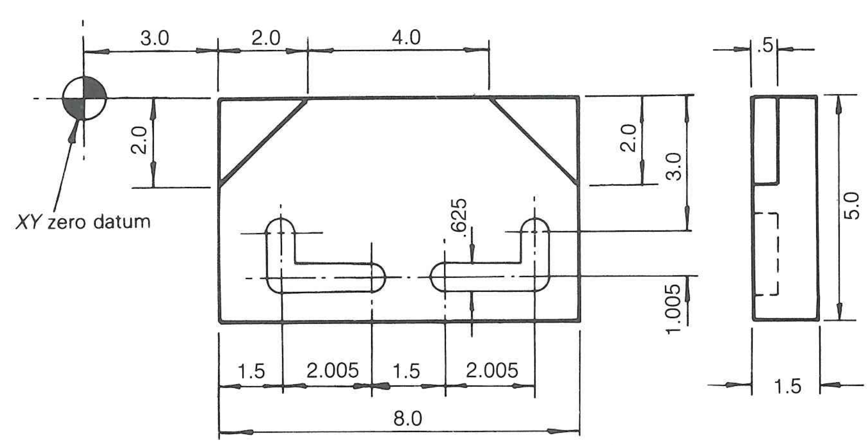
Mill top face
Mill angular steps
Mill 90° slots

Tooling

Cemented carbide insert shell end mill, 5.0∅, five teeth
HSS end mill .625∅

Programming

Incremental dimensions
Z datum clearance 2 inches
Linear interpolation
Quill clamp (steps and slots)
Mirror image
Spray mist coolant
Spindle speed rev/min
Feed inches/rev
Allocate tool offsets
Do *not* use cutter radius compensation

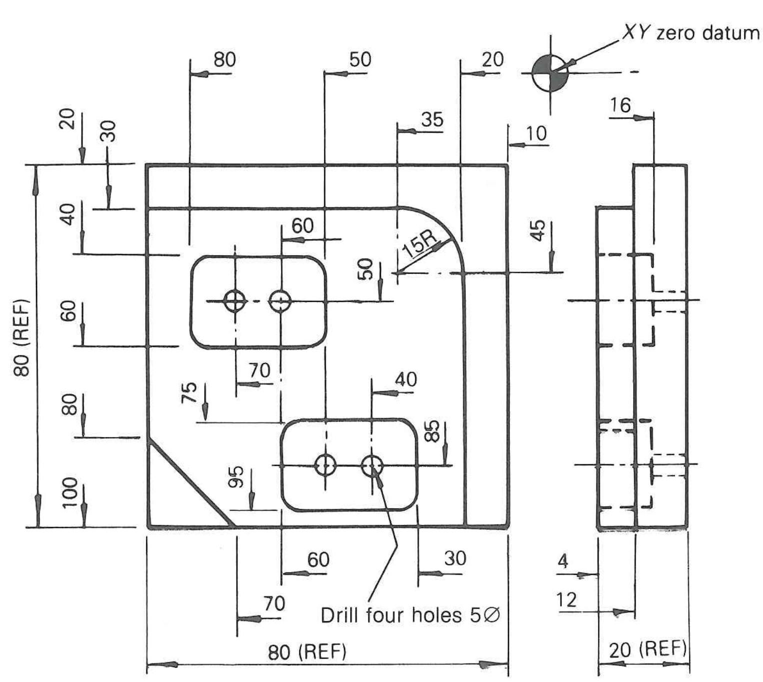


Material: brass
(Inch Programming Exercise)

EXERCISE NO.	8

Machining

Material supplied pre-machined 20 × 80 × 80
All other features to be machined
Complete an operation schedule (see Figure 8.16)
Detail tooling to be used
Detail work-holding and setting arrangements
Determine appropriate speeds and feeds
Complete a part program
Prove the part program
Machine the component
(Note: *Z* datum clearance 4mm)



Material: aluminum alloy
(Metric Programming Exercise)

EXERCISE NO.	9

Machining

Stock material 5.0∅ hand loaded and positioned

Face one end

Turn 4.5∅, 3.5∅ and 2.5∅, two passes per diameter

Part off to length

Tooling

Light turning and facing, cemented carbide insert

Parting off, cemented carbide insert .125 wide

Programming

Absolute positioning

Linear interpolation

Feed inches/rev

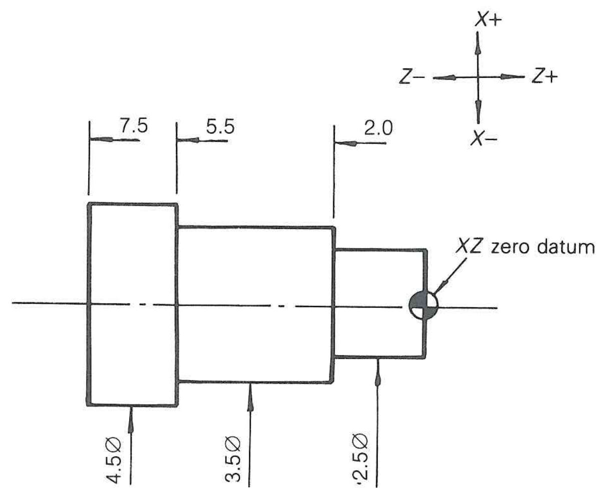
Spindle speed rev/min

Spray mist coolant

Allocate tool offset numbers

Ignore tool-tip radius

Axes indicated assume a rear-mounted turret



Material: brass
(Inch Programming
Exercise; See Example Program 9)

EXERCISE NO.	10

Machining

Material pre–faced billet 100.∅ hand loaded and located against back face of chuck

Face second end to length

Center drill

Drill 20∅ in two stages

Bore 40∅, 60∅ and 80∅

Depth of cut 2mm per pass

Tooling

No. 3 HSS center drill

HSS drill 10∅

HSS drill 20∅

Internal turning tool, cemented carbide insert

Facing tool, cemented carbide insert

Programming

Absolute dimensions

Linear interpolation

Feed mm/rev

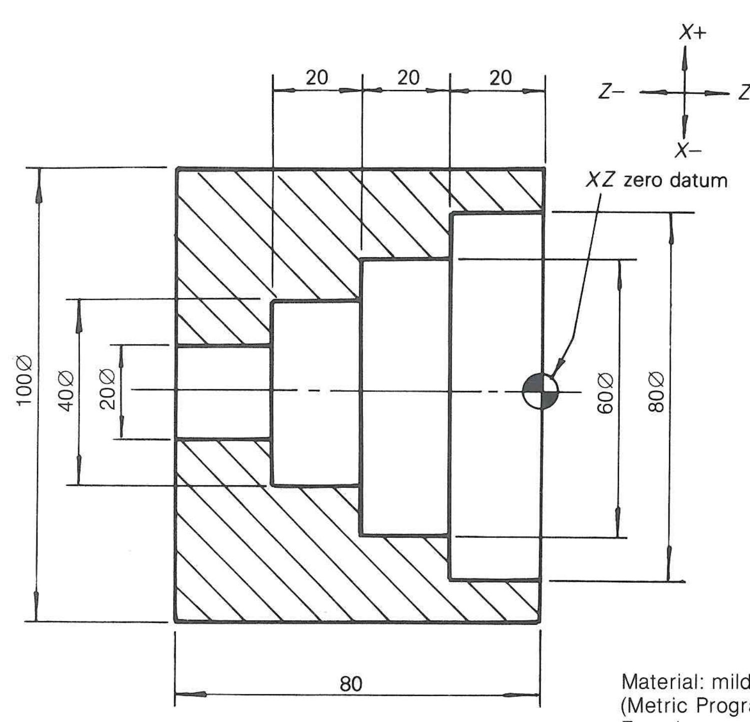
Surface cutting speed m/min

Ignore tool-tip radius

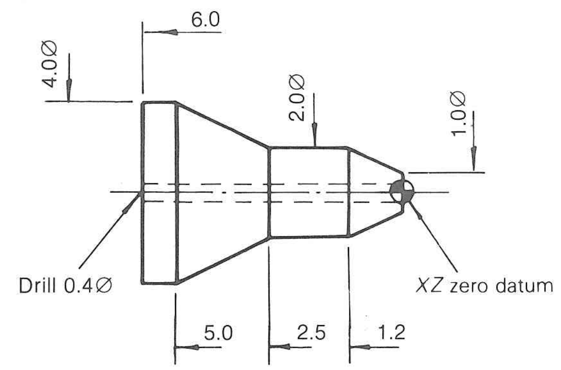
Allocate offset numbers

Flood coolant

Axes indicated assume a rear-mounted turret



Material: mild steel
(Metric Programming
Exercise; see Example
Program 10)



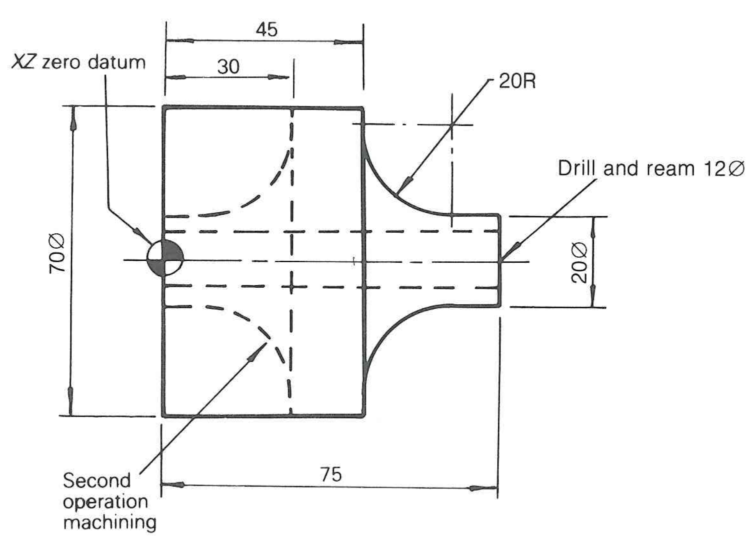
Material: brass
(Inch Programming Exercise)

EXERCISE NO.	11

Machining
Stock material 4.0∅ hand loaded and positioned
Face one end
Center drill
Drill 0.4∅
Rough turn to profile
Finish turn with continuous cut
Part off

Tooling
Light turning and facing tool, cemented carbide insert
HSS No. 2 center drill
HSS drill .4∅
Parting–off tool, cemented carbide insert 0.125 wide

Programming
Absolute dimensions
Linear interpolation
Feed inch/rev varied for roughing and finishing cuts
Surface cutting speed feet/min varied for roughing and finishing cuts
Flood coolant
Peck drill cycle
Ignore tool-tip radius
Allocate tool offset numbers



Material: medium carbon steel
(Metric Programming Exercise)

EXERCISE NO.	12

Machining
Material pre–faced billet 70∅ hand loaded and located against back face of chuck
Face second end to length
Center drill
Drill in two stages
Ream
Rough turn to profile
Finish turn profile

Tooling
Light turning and facing tool, cemented carbide insert, tip radius 2.0mm
HSS No. 3 center drill
HSS drill 6∅
HSS drill 11.5∅
HSS reamer 12∅

Programming
Absolute dimensions
Linear interpolation
Circular interpolation
Cutter radius compensation
Feed mm/rev, varied for roughing and finishing cuts
Surface cutting speed m/min varied for roughing and finishing cuts
Mist coolant
Peck drill cycle for initial drilling
Allocate tool offset numbers

EXERCISE NO.	13

Machining

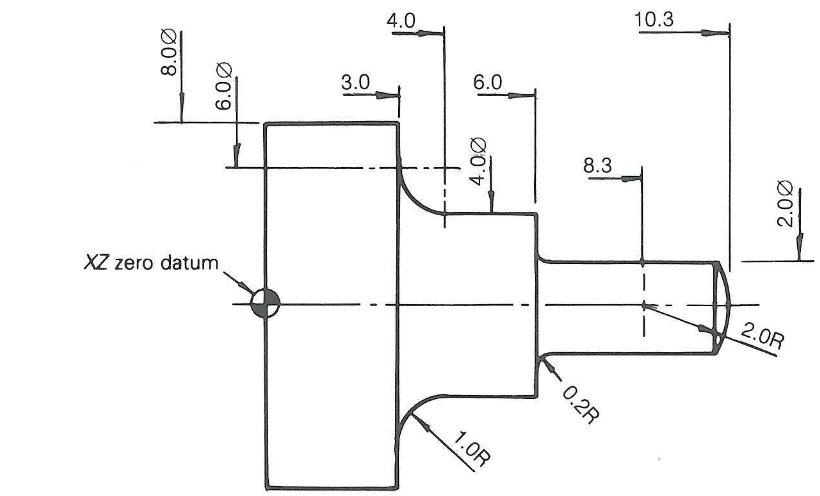
Material pre-faced billet 8.0∅ hand loaded and located against back face of chuck
Face second end to length
Rough turn 4.0∅ and 2.0∅
Finish turn complete profile in one pass

Tooling

Roughing tool, cemented carbide insert, tip radius 0.200
Light turning and facing tool, cemented carbide insert, tip radius 0.100

Programming

Absolute dimensions
Linear interpolation
Circular interpolation
Cutter radius compensation
Feed inches/rev varied for roughing and finishing cuts
Surface cutting speed feet/min varied for roughing and finishing
Mist coolant
Allocate tool offset numbers



Material: mild steel
(Inch Programming Exercise)

EXERCISE NO.	14

Machining

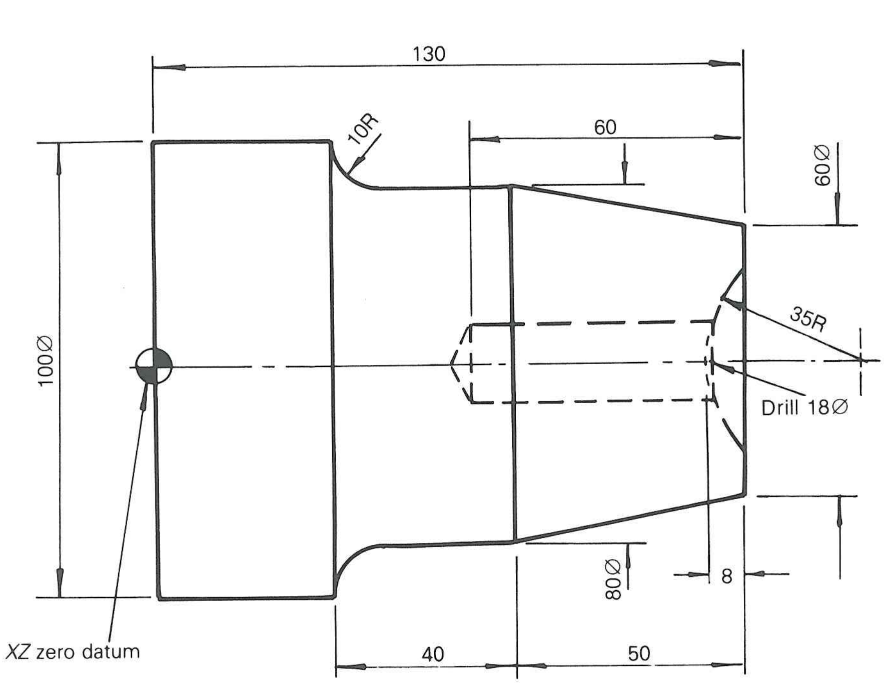
Material pre-faced billet 100∅,hand loaded and located against back face of chuck
Face second end to length
Center drill
Drill 18∅ in two stages
Machine end radius in three passes
Rough machine to profile
Finish machine profile with continuous cut

Tooling

Light turning and facing tool, cemented carbide insert, tip radius 0.8mm
Roughing tool, cemented carbide insert, tip radius 2mm
HSS No. 4 center drill
HSS drill 8∅
HSS drill 18∅
Internal turning tool, tip radius 1mm

Programming

Absolute dimensions
Linear interpolation
Circular interpolation
Cutter radius compensation
Feed mm/rev varied for roughing and finishing cuts
Surface cutting speed m/min varied for roughing and finishing cuts
Flood coolant
Allocate tool offset numbers



Material: brass
(Metric Programming Exercise)

EXERCISE NO.	15

Machining

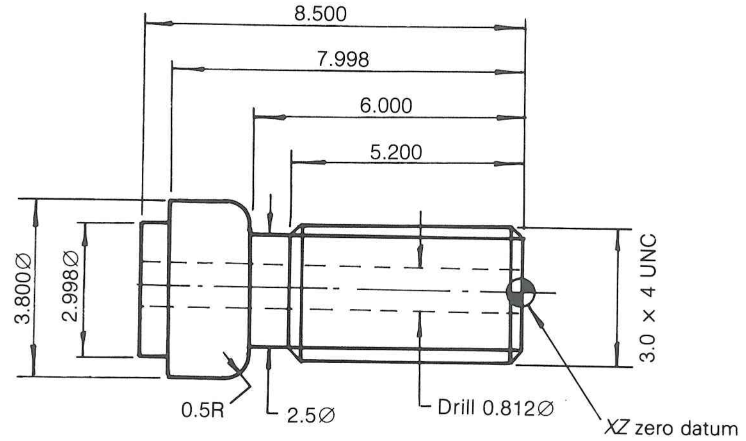
Stock material 4.0∅, hand loaded and positioned
Face end
Turn 3.8∅, 3.0∅, 2.5∅
Cut screw thread 3.0 × 4 UNC
Center drill
Drill 0.812∅
Part off to length

Tooling

Light turning and facing tool, cemented carbide insert, tip radius .031
Grooving tool, cemented carbide insert, 0.250 wide
Parting-off tool, cemented carbide insert, 0.125 wide
Screw cutting tool, cemented carbide insert, unified national form
HSS No. 3 center drill
HSS 0.812 drill

Programming

Absolute positioning
Linear interpolation
Circular interpolation
Cutter radius compensation
Screw cutting cycle
Peck drilling cycle
Feed inches/rev
Surface cutting speed feet/min
No coolant
Allocate tool offsets



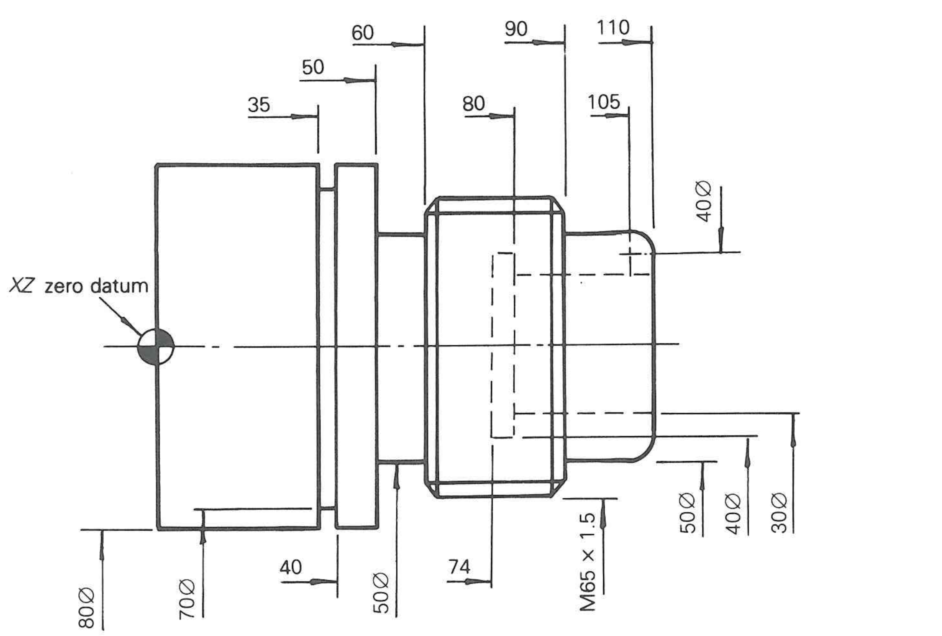
Material: brass
(Inch Programming Exercise)

EXERCISE NO.	16

Machining

Material pre-faced billet 80∅
Machine to drawing
Complete an operation schedule

Detail tooling to be used
Detail work-holding and setting arrangements
Determine appropriate speeds and feeds
Complete a part program
Prove the part program
Machine the component



Material: medium carbon steel
(Metric Programming Exercise)

Assigned miscellaneous functions:	M plus two digits
M00	Programmed stop. Stops all slide movement, spindle rotation and coolant
M01	Optional stop. Ignored unless activated manually from control console.
M02	End of program
M03	Spindle on clockwise
M04	Spindle on counter-clockwise
M05	Spindle off
M07	Coolant on (mist)
M08	Coolant on (flood)
M09	Coolant off
M10	Quill clamp on
M11	Quill clamp off
M30	End of program. Rewind tape.
Axis movement commands:	End point in *X*, *Y* and *Z*
	Start point of arcs relative to circle center *I*, *J* and *K*
	Dimensional values in 3/2 format metric, that is, three digits before and two digits after the decimal point; 2/3 format inch. Do not program the decimal point and omit leading zeros.
	Plus (+) signs are not required but (−) signs must be included.
Other functions:	
Feed	F plus four digits
	Feed/min in 1 mm or 0.1 in. steps
	Feed/rev in 0.001 mm or 0.001 in. steps
	Do not program decimal point or leading zeros
Spindle speed	S plus four digits
Tool identification	T plus two digits
	T01 to T16. With offsets use four digits, first two tool number, second two offset number, offsets 00 to 32.
Dwell	D plus three digits in 0.1 s (Do not program decimal point.)
	Slash delete. Messages ignored when 'slash delete' switch on the control unit console is activated. They are obeyed when switch is off.
%	Rewind stop
*	End of block

PROGRAMMING NOTES RELATING TO MILLING AND DRILLING EXERCISES

Pocket Clearance Cycle (G28)

1. Position the cutter over the center point of the pocket.
2. Program the appropriate movement in the *Z* axis.
3. Program G28 with *X* and *Y* values indicating pocket dimensions. Cutter radius compensation and the step-over value will be automatically determined and implemented.
4. Cancel the cycle by programming G80.

(Note this code is not available or standard on all machines.)

Mirror Image (G31)

1. Program G31 together with the axis or axes to be mirrored. For example, G31 *X* will mirror in the *X* axis only; G31 *XY* will mirror in the *X* and *Y* axes. No other data are to be included in this block.
2. Program the original axes commands.
3. Cancel by programming G30.

(Note this is not a standard code that is available on all machines, some machines will use a manual switch.)

Cutter Radius Compensation (G41, G42)

1. Program G41 or G42 in the same block as G01 or G00 when making the approach move to the profile to be machined. The cutter will offset by the appropriate radius.
2. Cancel G41 or G42 by programming G40 in the withdrawal move.

(Note some machines will require this code to be programmed in a separate block before the move.)

Drill Cycle (G81)

1. Program G81 when making the approach move to the first hole position. The block must also contain the *Z* depth to be drilled and *Z* clearance position. The drill cycle will activate at the end of the positional move and will be repeated at the end of each subsequent positional move until cancelled.
2. Cancel G81 by programming G80 in the withdrawal move.

Peck Drill Cycle (G83)

1. Program G83 when making the approach move to the first hole position. The block must also contain the *Z* depth to be drilled, the *Z* clearance position and the peck distance as a *W* or *K* increment. The drill cycle will activate at the end of the positional move and will be repeated at the end of each subsequent positional move until cancelled.
2. Cancel G83 by programming G80 in the withdrawal move.

ALTERNATIVE MACHINE SPECIFICATION AND CONTROL SYSTEM FOR PROGRAMMING EXERCISES 1 TO 8: MILLING AND DRILLING

Machine Type and Specification

Vertical machining center with three-axis control

Traverse:	*X* longitudinal 600 mm (24 in.)
	Y transverse 400 mm (16 in.)
	Z vertical 450 mm (18 in.)
Spindle speed:	10–3300 rev/min infinitely variable
Working surface:	1000 mm × 300 mm (40 in. × 12 in.)
Feeds:	1–5000 mm/min (0.1–198 in/min)
Tool changing:	Manual

Control System

Program format:	Word address
Axes controlled:	*X*, *Y* and *Z* singly or simultaneously
Interpolation:	Linear *X*, *Y* and *Z*
	Circular *XY*, *YZ* and *ZX* planes
Command system:	Incremental or absolute
Data input:	MDI or perforated eight-track tape
Tape code:	ISO or EIA via tape sensing

Programming Information

Block numbers:	N plus one to three digits
Preparatory functions:	G plus two digits
G00	Rapid traverse at machine maximum (modal)
G01	Linear interpolation, programmed feed (modal)
G02	Clockwise circular interpolation (modal)
G03	Counter-clockwise circular interpolation (modal)
G08	Ramp down inhibit (modal)
G09	Cancels G08
G28	Pocket clearance cycle (modal)

G30	Cancels mirror image
G31	Mirror image with axis command (modal)
G40	Cutter offset cancel
G41	Cutter radius compensation left (modal)
G42	Cutter radius compensation right (modal)
G70	Inch programming (modal)
G71	Metric programming (modal)
G80	Cancels all fixed cycles
G81	Drill cycle (modal)
G83	Peck drill cycle (modal)
G90	Absolute programming (modal)
G91	Incremental programming (modal)
G93	Inverse time feedrate (V/D) (modal)
G94	Feed inch (mm)/min (modal)
G95	Feed inch (mm)/rev (modal)
G96	Constant surface speed
G97	Spindle speed rev/min (modal)

ALTERNATIVE MACHINE SPECIFICATION AND CONTROL SYSTEM FOR PROGRAMMING EXERCISES 9 TO 16: TURNING

Machine Type and Specification

Precision turning center with two-axis control

Traverse:	*X* transverse 160 mm (6.3 in.)
	Z longitudinal 450 mm (17.7 in.)
Spindle speed:	50–3800 rev/min
Feed:	0–400 mm/min, 0–200 inch/min
Tooling:	Indexable turret providing eight tool stations

Control System

Program format:	Word address
Axes controlled:	*X* and *Z* singly or simultaneously
Interpolation:	Linear *X* and *Z* axes
	Circular *XZ* plane
Command system:	Absolute
Data input:	MDI and magnetic tape

Programming Information

Block numbers:	N plus one to three digits
Preparatory functions:	G plus two digits
G00	Rapid traverse at machine maximum (modal)
G01	Linear interpolation, programmed feed (modal)

G02	Clockwise circular interpolation (modal)
G03	Counter-clockwise circular interpolation (modal)
G08	Ramp down inhibit (modal)
G09	Cancels G08
G33	Threading cycle
G40	Cutter offset cancel
G41	Cutter radius compensation left (modal)
G42	Cutter radius compensation right (modal)
G70	Inch programming (modal)
G71	Metric programming (modal)
G80	Cancels all fixed cycles
G81	Drill cycle (modal)
G83	Peck drill cycle (modal)
G90	Absolute programming (modal)
G91	Incremental programming (modal)
G92	Preset axes command
G93	Feed inverse time (V/D)
G94	Feed inch or mm/min (modal)
G95	Feed inch or mm/rev (modal)
G96	Surface cutting speed feet or meters/min
G97	Surface speed rev/min
Assigned miscellaneous functions:	M plus two digits
M00	Programmed stop. Stops all slide movement, spindle rotation and coolant.
M01	Optional stop. Ignored unless activated manually from control console.
M02	End of program
M03	Clockwise spindle rotation
M04	Counter-clockwise spindle rotation
M05	Spindle off
M06	Tool change
M07	Coolant on (mist)
M08	Coolant on (flood)
M09	Coolant off
M30	End of program. Tape rewind.
Axis movement commands:	End point in *X* and *Z* Start point of arcs relative to circle center/and *K* (see pp. 128–136) Dimensional values in 3/2 format metric, that is, three figures before and two after the decimal point; 2/4 format inch. Program the decimal point but no leading or trailing zeros. Plus (+) signs are not required but (−) signs must be included. *X* values to be programmed as a diameter.
Other functions:	
Feed	F plus four digits Feed/min in 1 mm or 0.1 in. steps Feed/rev in 0.001 mm or 0.001 in. steps Do not program decimal point
Spindle speed	S plus four digits

Surface speed	S plus three digits
Tool identity	T plus two digits, T01 to T10. With offsets program four digits, offsets 00 to 24, first two digits tool number/second two offset number.
%Rewind stop *End of block	Slash delete. Messages ignored when 'slash delete' switch on the control unit console is activated. They are obeyed when switch is off.

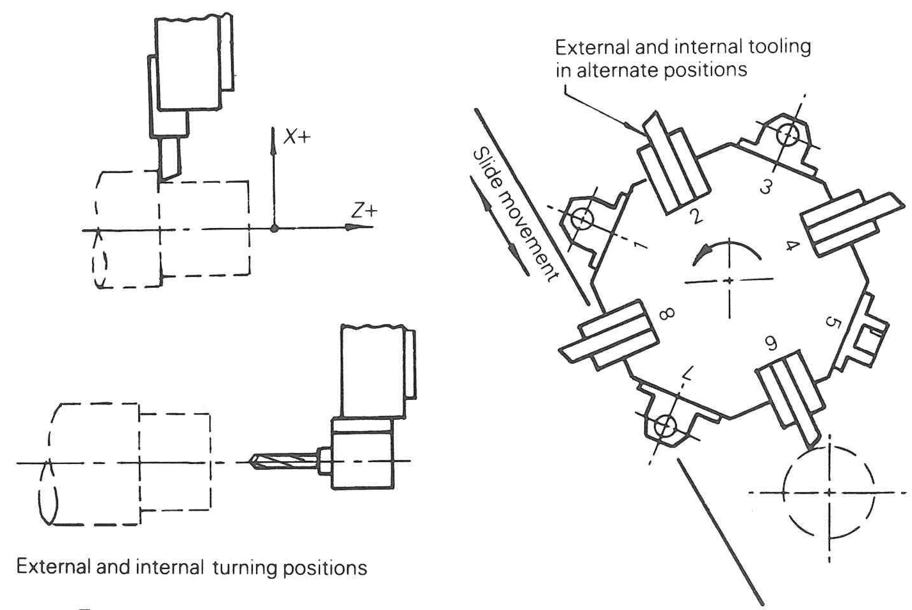
PROGRAMMING NOTES RELATING TO TURNING EXERCISES

Tooling System and Turret Indexing

The tooling system comprises an eight-station turret mounted at 90° to the machine spindle axis, the tooling positions being numbered 1 to 8. (See following figure.) It illustrates the way the tooling is arranged in relation to the workpiece.

Turret indexing is achieved as follows:

1. In the block following the end of a machining sequence program M06 (tool change). No other data are to be included in this block. The turret will withdraw to a pre-set safe indexing position.
2. In the next block program the required tool number and, if applicable, its related tool offset number. The turret will index by the shortest route to that tool position.



Turret arrangement

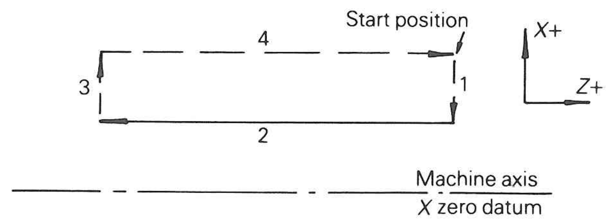
3. In the next block program a rapid move to the next pre-cutting position, which should be approximately 2mm or 0.100 in. clear of the workpiece.
4. Finally, continue to program moves at a controlled feedrate.

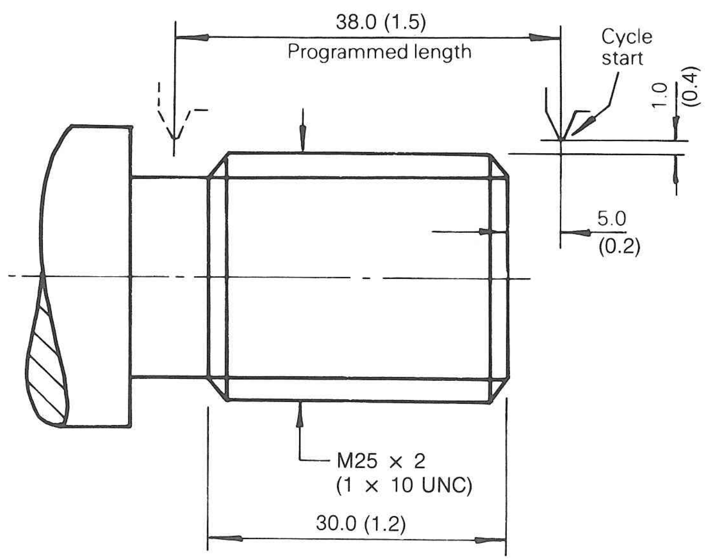
Note tool changes for various machines will program differently and machinery manuals should be consulted.

Screw-cutting Cycle (G33)

The movement sequence of the screw-cutting cycle is illustrated in the following figure.

1. Tool moves rapidly to programmed *X* value.
2. Thread is cut to programmed *Z* value.
3. Rapid traverse to initial position in *X* axis.
4. Rapid traverse to initial position in *Z* axis.





Component detail. (Inch units are given in parentheses.)

The final thread depth is reached by a series of rough and finish pass cuts as indicated in the part program below, which refers to the preceding figure.

The pitch is programmed as a *K* value.

Allow an approach distance of 0.250 inches in th *Z* axis when cutting threads.

(Inch)	N	G	X	Z	I	K	F	S	T	M	*
	35	0	2.7	.25							*
	40	33	2.48	−3.8		2		490	0404		*
	45		2.46								*
	50		2.44								*
	55		2.42								*
	60		2.4								*
	65		2.38								*
	70		2.36								*

Allow an approach distance of 5 mm in the *Z* axis when cutting threads.

(Metric)	N	G	X	Z	I	K	F	S	T	M	*
	35	0	26	5							*
	40	33	24.8	−38		2		490	0404		*
	45		24.6								*
	50		24.4								*
	55		24.2								*
	60		24								*
	65		23.8								*
	70		23.6								*

The *X* values are continually reduced until the full thread depth is reached. Two passes of the final cut should be made to reduce tool pressure for finishing.

The *Z* axis move is programmed as an incremental value from the start position.

The cycle is cancelled by programming G80.

Drill Cycles (G81 and G83)

1. Position drill at start point in the *Z* axis on *X* zero datum.
2. Program G81 or G83 together with *Z* depth to be drilled. G83 requires the peck distance to be programmed as a special coded increment. (Refer to machine manual for address used.)
3. Cancel G81 or G83 by programming G80.

Cutter Radius Compensation (G41 and G42)

Common to both turning and milling exercises.

PARTIALLY COMPLETED EXERCISES USING ALTERNATIVE CONTROL SYSTEMS

Exercise 1 (Inch Program)

PART PROGRAM EXERCISE NO. 1					MACHINE As book / COMPILED BY A. Programmer						CONTROL As book / DATE 28-3-84			
N	G	X	Y	Z	I	J	K		F	S	T	M	*	REMARKS
01								%					*	Rewind stop
05	70												*	Inch units
10	90												*	Absolute
15	94												*	Feed inch/min
20	97												*	Speed rev/min
25	00	8000	4500	100							0101		*	Rapid to Z clear/P1
30										2000		03		Spindle/coolant on
	01			-1300					200			08	*	Feed to Z depth
35	00			100									*	Rapid to Z clear
40			1300										*	Rapid to P2
45	01			-1300									*	Feed to Z depth
50	00			100									*	Rapid to Z clear
55		4000											*	Rapid to P3
60	01			-1300									*	Feed to Z depth
65	00			100									*	Rapid to Z clear
70			3000										*	Rapid to P4
75	01			-1300									*	Feed to Z depth
80	00			100									*	Rapid to Z clear
85			4500										*	Rapid to P5
90	01			-1300									*	Feed to Z depth
95	00			100									*	Rapid to Z clear
100		0	0	2000										Rapid to base datum
				(Manual tool change)								00	*	Program stop

Exercise 2 (Metric Program)

PART PROGRAM EXERCISE NO. 2					MACHINE As book / COMPILED BY A. Programmer						CONTROL As book / DATE 8-5-84			
N	G	X	Y	Z	I	J	K		F	S	T	M	*	REMARKS
01								%					*	Rewind stop
02	71												*	Metric units
03	91												*	Incremental
04	95												*	Feed mm/min
05	97												*	Speed rev/min
06	00	6000	-1500								0101		*	Rapid to P1/Z0
07										750		03		Spindle/coolant on
	01			-1000					225			07	*	Feed to Z depth
08								D20				10	*	Dwell, Quill clamp
09			-4500										*	Feed in Y axis
10		2500											*	Feed in X axis
11			4500										*	Feed in Y axis
12												11	*	Quill clamp off
13	00			20000										Rapid lift
												00	*	Program stop
				(Manual tool change)										

Exercise 9 (Inch Program Example, Decimal Point Programming)

PART PROGRAM EXERCISE NO. 9 | MACHINE As book | CONTROL As book
COMPILED BY A. Programmer | DATE 20-6-84

N	G	X	Y	Z	I	J	K		F	S	T	M	*	REMARKS
01								%					*	Rewind stop
05	70												*	Inch units
10	95												*	Feed inches/rev.
15	90												*	Absolute
20	97												*	Speed rev/min.
25											0202		*	Tool/offset
30	00	5.1·		0									*	Rapid to start
										2300		03		Spindle on
35	01	0							.007			07	*	Feed to X0 coolant on
40	00			.1·									*	Clear face
45		4.5·											*	Return in X axis
50	01			-7.625						.010			*	Feed in Z axis
55	00	4.6·											*	Clear diam.
60				.1·									*	Return to start
65		4.0·											*	Rapid in X axis
70	01			-5.5·						.010			*	Feed in Z axis
75	00	4.1·											*	Clear diam.
80				.1·									*	Return to start
85		3.5·											*	Rapid in X axis
90	01			-5.5·						.010			*	Feed in Z axis
95	00	3.6·											*	Clear diam.
100				.1·									*	Return to start

Exercise 10 (Metric Programming Example)

PART PROGRAM EXERCISE NO. 10 | MACHINE As book | CONTROL As book
COMPILED BY A. Programmer | DATE 5-6-84

N	G	X	Y	Z	I	J	K		F	S	T	M	*	REMARKS
01								%					*	Rewind stop
05	71												*	Metric
10	95												*	Feed mm/rev.
15	96												*	Speed m/min.
20											0202		*	Tool and offset
25	00	104·		0									*	Rapid to start
30												03		Spindle/on
	01	0							250	170		08	*	Feed to X0, coolant on
35	00			2·									*	Clear face
40												06	*	Tool change
45											0101		*	Index to c/drill
50		0		2·									*	Rapid to start
55	01			-5·					80	28			*	Center drill
60	00			2·									*	Return to start
65												06	*	Tool change
70											0303		*	Index to drill 10ϕ
75		0		2·									*	Rapid to start
80	83			-85·				w 20	180				*	Peck drill 10ϕ
85	80												*	Cancel cycle
90												06	*	Tool change
95											0505		*	Index to drill 20ϕ
100	00	0		2·									*	Rapid to start
101	81			-85·					300				*	Drill 20ϕ

APPENDIX D

GDT SYMBOLS

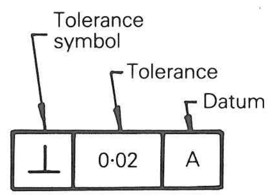
Geometric tolerances

Geometrical tolerance symbols

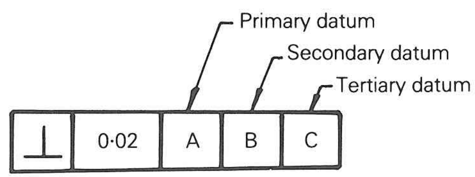
SYMBOL	CHARACTERISTIC	APPLICATION
—	STRAIGHTNESS	Applied to an edge line or axis. For an edge or line the tolerance zone is the area between two parallel straight lines containing the edge or line. The tolerance value is the distance between the two lines.
⏥	FLATNESS	Applied to a surface. The tolerance zone is the space between two parallel planes. The tolerance value is the distance between the two planes.
○	ROUNDNESS	Applied to the cross-section of a cylinder, cone or sphere. The tolerance zone is the angular space between two concentric circles lying in the same plane. The tolerance value is the distance between the two circles.
⌭	CYLINDRICITY	Applied to the surface of a cylinder. Combines roundness, straightness and parallelism. The tolerance zone is the angular space between two coaxial cylinders. The tolerance zone is the radial distance between the two cylinders.
⌒	PROFILE OF A LINE	Applied to a profile. The tolerance zone is an area defined by two lines that have a constant width normal to the stated profile. The tolerance is the diameter of a series of circles contained between the two lines. The tolerance may be unilateral or bilateral.
⌓	PROFILE OF A SURFACE	Applied to a surface. The tolerance zone is a space contained between two surfaces normal to the stated surface. The tolerance value is the diameter of a series of spheres enveloped by the two surfaces. The tolerance may be unilateral or bilateral.
//	PARALLELISM	Applied to a line, surface or cylinder. The tolerance zone is the area between two parallel lines or planes, or the space between two parallel cylinders, which must be parallel to the datum feature. The tolerance is the distance between the two lines or planes or, in the case of a cylinder, the diameter of the cylinder.
⊥	SQUARENESS	Applied to a line, surface or cylinder. For a line or surface the tolerance zone is the area between two parallel lines or planes which are perpendicular to the datum surface. The tolerance is the distance between the lines or planes. For a cylinder the tolerance zone is the space within a cylinder equal in diameter to the tolerance value and perpendicular to the datum plane.
∠	ANGULARITY	Applied to a line, surface or cylinder. For a line or surface the tolerance is the area or space between two parallel lines or planes inclined at a specified angle to the datum feature. For a cylinder the tolerance zone is the space within a cylinder equal in diameter to the tolerance value and inclined at a specified angle to the datum feature.
⌖	POSITION	Applied to a circle or cylinder. The tolerance zone is the space within a cylinder equal in diameter to the tolerance value and coaxial with the datum axis. The tolerance limits the deviation of the datum axis from its true position.
◎	CONCENTRICITY	Applied to parallel lines or surfaces. The tolerance zone is the area of space between the lines or surfaces symmetrically disposed in relation to a datum feature. The tolerance value is the distance between the lines or planes.
⌯	SYMMETRY	Applied to a point, axis, line or plane. The tolerance zone definition varies according to the feature. The tolerance value will limit the positional deviation from the specified true position.
↗	RUN OUT	Applied to the surface of a solid of revolution or to a face perpendicular to the axis. The tolerance value indicates the permissible indicator movement during one revolution.
○	MAXIMUM MATERIAL CONDITION	MMC exists when the component or feature contains the maximum amount of material permitted by its dimensional tolerances. When M is included in a tolerance frame the tolerance value need only be applied rigorously when the component or feature is in that condition. When not in that condition the geometric tolerance may be increased up to the difference between the MMC limit and the actual finished size.

Geometric tolerance frames

1. Tolerance relating to a single datum:



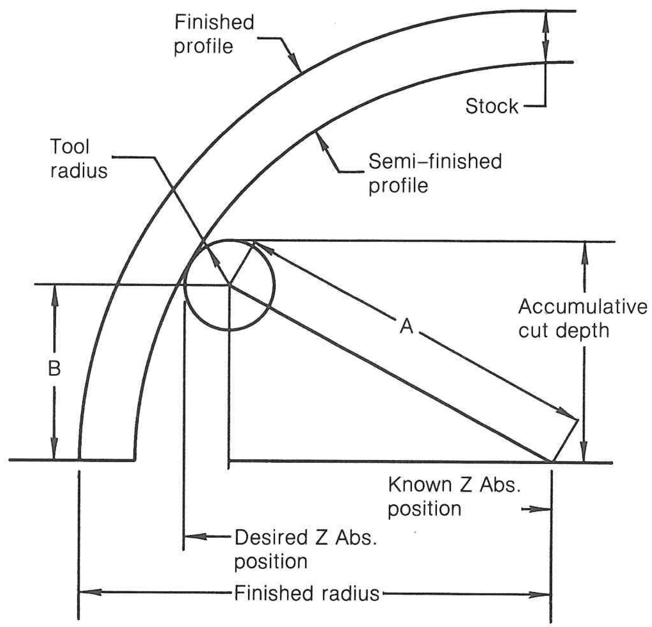
2. Tolerance relating to more than one datum:



APPENDIX E

FORMULAS

FORMULA #1

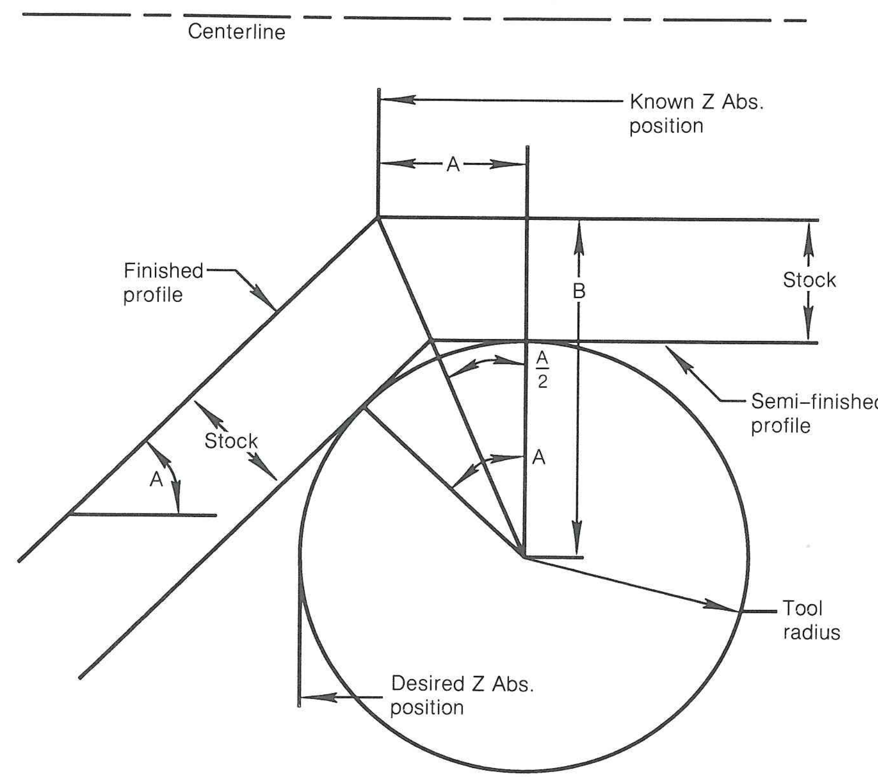


A = Finished radius–stock–tool radius

B = Accumulative cut depth–tool radius

Desired Z Abs. = Known Z Abs. – $\sqrt{A^2 - B^2}$ – tool radius

FORMULA #2

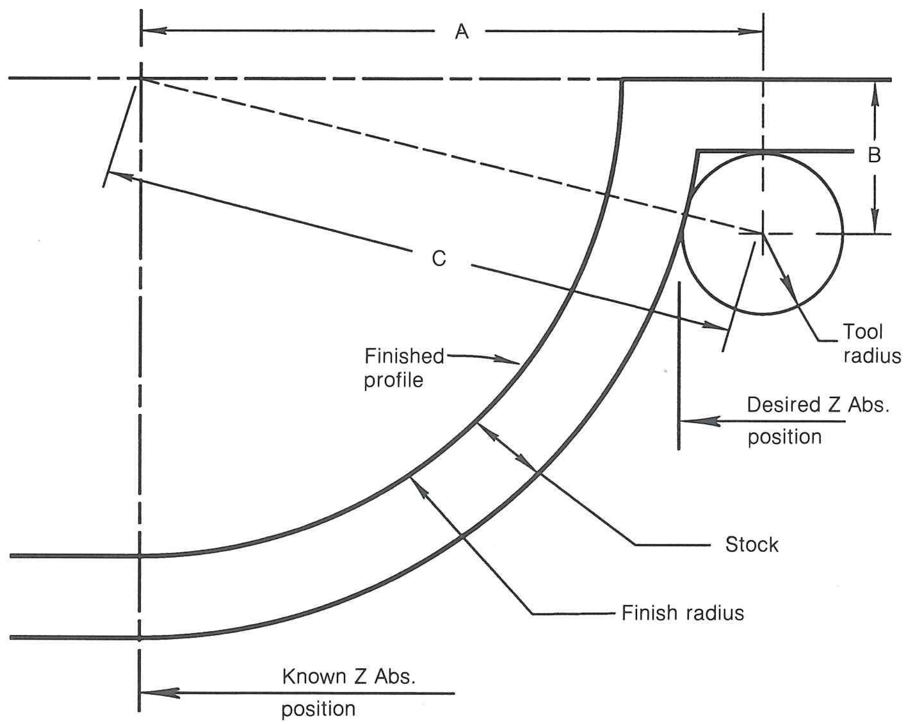


B = Stock + tool radius

A = Tan $\frac{A}{2}$ × B

Desired Z Abs. = Known Z Abs. – (tool radius – A)

FORMULA #3



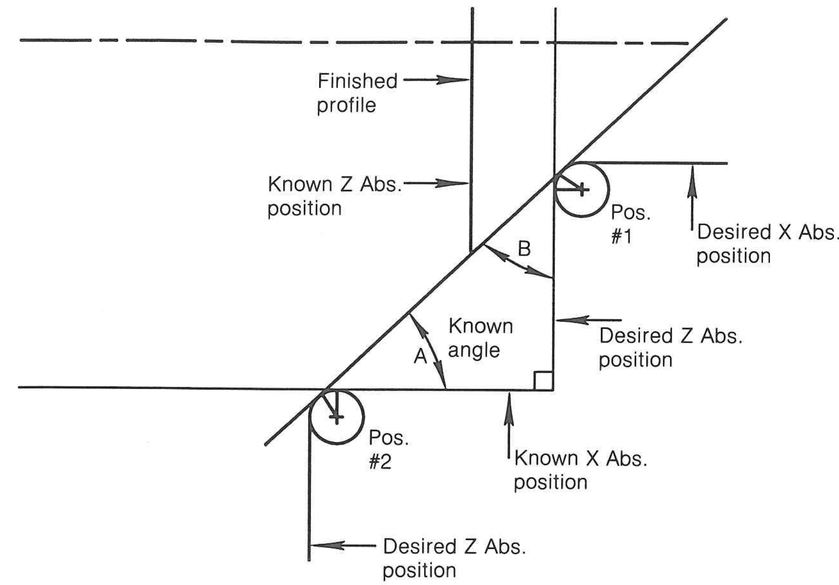
B = Tool radius + stock

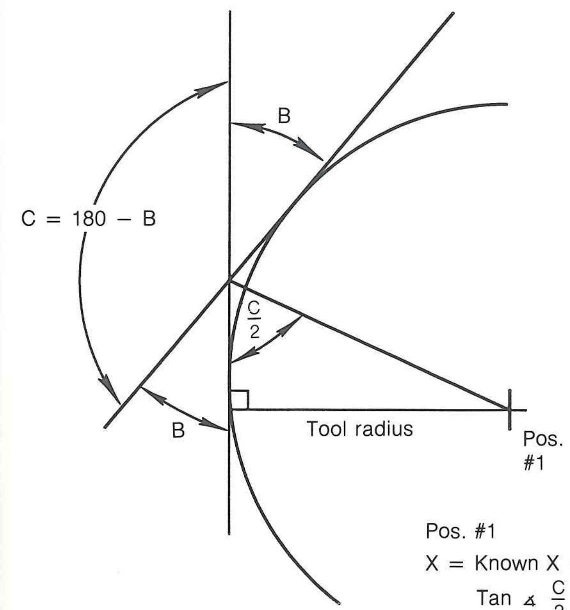
C = Finish radius + stock + tool radius

$A = \sqrt{C^2 - B^2}$

Desired Z Abs. = Known Z Abs. + A – tool radius

FORMULA #4





Pos. #1

X = Known X – tan ∡ A (chamfer length + clearance) + Tan ∡ $\frac{C}{2}$ (tool radius) – tool radius

Z = Known Z + clearance

Pos. #2

X = Known part radius

Z = Known Z – chamfer length + tan ∡ $\frac{C}{2}$ (tool radius) – tool radius

INDEX