SECTION III

TOOLS AND METHODS CONSIDERATIONS

3.1 CUTTING TOOLS

It is important to give attention to the cutting tool and its relationship to the machine spindle since it is the tool that makes the chips which causes the machine tool to be considered a productive manufacturing system. What good therefore is a precision spindle with .0005" TIR runout 6" from the spindle nose, if a drill point is going to walk .005"!

3.1.1 The Drill

The characteristics of the standard twist drill will be discussed by illustrating the problems faced in making an accurate hole.

The twist drill is the most commonly used and most effective metal removing tool, however, its complex geometry (Figure 3-1) leads to problems in making a true hole of the correct size and location. The drill must have the web mass and rigidity to withstand the torsional forces generated deep within the hole; but the long overhang, the necessary web thickness and blunt point make it a poor candidate to break the workpiece surface to start a hole. The drill point may even locally work - harden the workpiece surface then "walk" off center before penetrating the material.

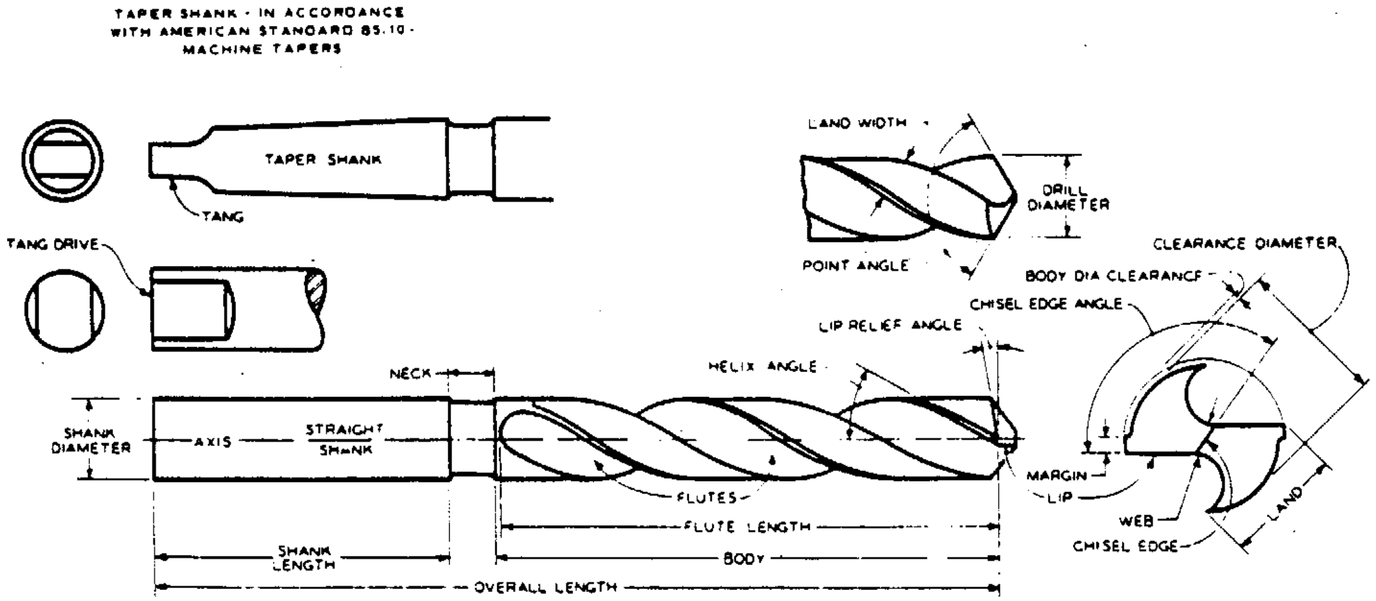


Figure 3-1. Identity of Twist Drill Features

Geometry: There are allowable errors in the geometry of a twist drill defined in USAS B94.11-1967 examples of which are shown below:

1/4" to 1/2" Dia. Drills	
Lip Height	.004"
Web Concentricity	.005"
Flute Spacing	.005"

At premium cost from the supplier or with proper drill pointing and web thinning equipment, each of the above tolerances can be reduced by a factor of 3.

Length: Stub length drills should be used where possible or longer drills held in a collet gripping on the margin of the drill so as to expose minimum length. Stub length drills are also known as the Screw Machine Drills (Figure 3-2). They permit faster feeds and still generate good quality holes - this is typical of Screw Machine Operations with short, rigid tooling. If a deep hole is required, jobbers length or taper length drills should be used only after first using a screw machine drill to about 75 percent of its flute length.

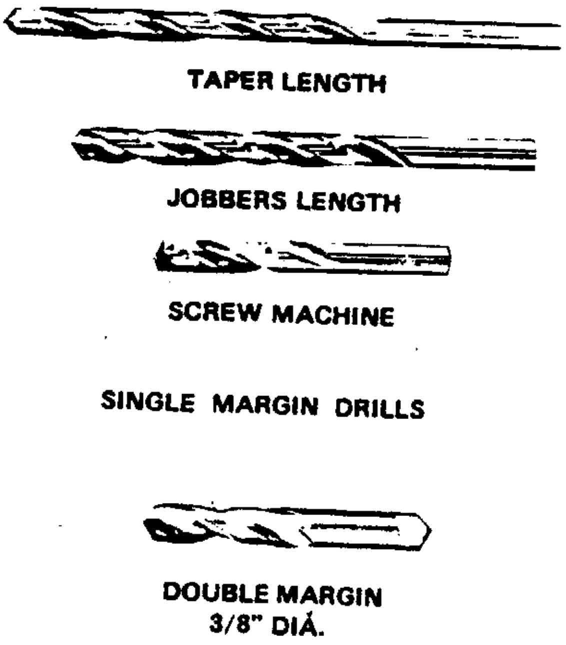


Figure 3-2. Twist Drills

Sub Land Drills: The small drill size at the top of the Sub Land Drill has additional rigidity due to the larger diameter of the shank. Additional benefit of dual operations with one tool is also evident (Figure 3-3). Production and regrind facilities must warrant the use of this tool.

Spade Drills: Half the cost of twist drills but require internally fed coolant if the depth is greater than 1.5 times the diameter. Very high horsepower is required to use a spade drill.

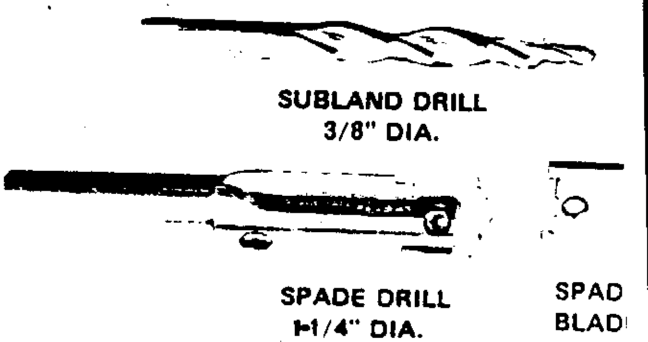


Figure 3-3. Subland and Spade Drills

Center Drills: A combination drill and countersink (Figure 3-4) that is of greatest benefit in lathes ONLY. This tool is not recommended for spot drilling because speeds and feeds are not compatible with both the small drill point and the countersink diameter.

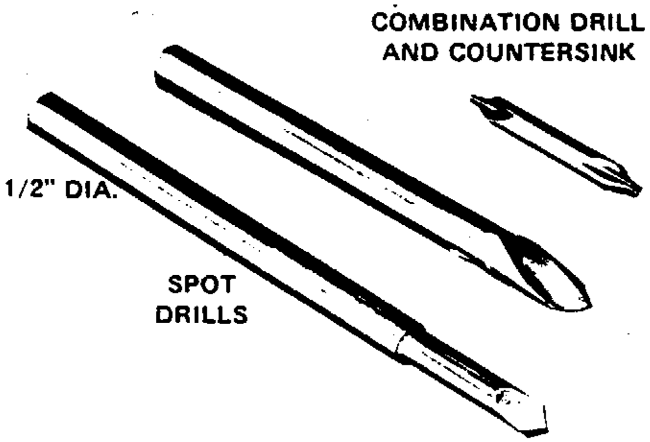


Figure 3-4. Spot Drills

Spot Drill: A special purpose tool with very sharp point and very thin web to penetrate the surface of a workpiece without "walking." (Figure 3-4) It is never used beyond the depth of the point itself (working depth about 95 percent of point depth) and it therefore is generally designed with very short flute length on a shank of any suitable length. Since most twist drills have a 118 degree angle (depth of point 0.3 diameter) the best spot drill is made with a 90 degree angle for four reasons.

1. Sharper Angle for surface penetration.
2. The cone produced forces the following drill to engage at the cutting edge lip, not the blunt chisel point (Figure 3-5). Essential when hole location tolerances are better than ± 1/64" and the drill length to diameter ratio exceeds 4:1.
3. The programming depth equals one-half the desired cone diameter - easy calculation (Figure 3-6) and one spot drill covers many drill sizes.
4. The cone diameter will produce a natural chamfer at the top of the hole.
5. Sometimes used to drill through thin sheet metal when tolerances permit.

NOTE: For holes with location tolerances of ± .005" or better, a boring operation is recommended.

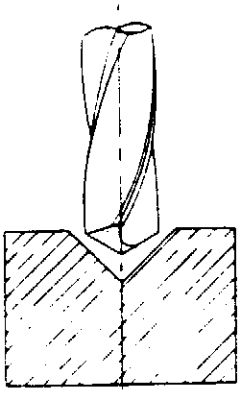
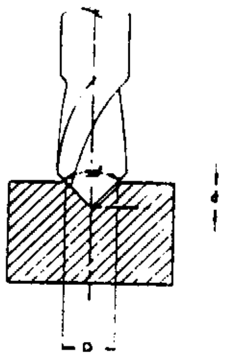


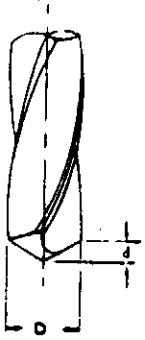
Figure 3-5. Geometry at Hole Location

90° SPOT DRILL



DEPTH OF HOLE d = D/2
Where
D = DIAMETER OF HOLE + CHAMFERED EDGE

118° STD. DRILL POINT



DEPTH OF DRILL PT. d =.3D
Where
D = DIAMETER OF DRILL

Figure 3-6. Drill Point

3.1.2 The Tap

Numerical Control frees the operator from the skill necessary to produce a good tapped hole. The main consideration is the tap itself.

Spiral Point: (Figure 3-7): Also called a "gun tap," drives the chip ahead of the tap and permits additional strength by having a heavier web and prevents chips from packing in the flutes. Excellent for thru holes or blind holes with adequate chip clearance at the bottom of the hole.

Spiral Flute: (Figure 3-7): The opposite of the spiral point tap. The spiral flute tap lifts the chip out of the hole instead of pushing the chip ahead - it is therefore of greatest use in blind holes where its action is much like that of a twist drill. This tap is highly recommended for ductile materials. The bottoming tap is recommended over a plug tap because the heavier chip it generates will be pulled out of the hole easier than the thinner, more abundant chips of the plug tap.

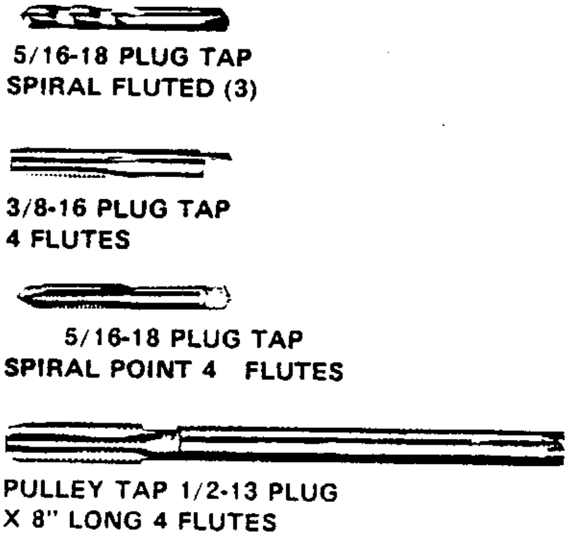


Figure 3-7. Machine Taps

Pipe Taps: The tapered pitch diameter of these threads provides a challenge to the spindle drive system and to the wear on the tap itself. Some applications require taper-reaming.

Detailed Considerations:

Speed and Feed relationship is fixed by the lead of the tap.
Chip Clearance - particularly in blind holes.
Adequate Lubrication at the cutting edge - lard oil is best.
Coarse threads in small diameters.
High thread engagement - 55 to 65% on diameter is sufficient.
Long thread engagement - 1.5 x diameter is sufficient.
Tough materials - probably requires a high sulphur lubricant.

3.1.3 Boring

Multi-function NC Machines have taken over much of the jig boring market where the following tolerances are required:

Positioning Accuracy ± .0005"
Hole Roundness within .0002"

To achieve these figures, tools must be short and rigid.
(See discussion on Tool Deflection in Paragraph 3.1.6)

An offset boring bar creates spindle unbalance.

Cartridge type inserts for single point boring are best for production.

3.1.4 Points to Watch

We have referred to a precision spindle on an NC machine with a maximum runout of .0005" TIR 2"-4" from the nose. All this can be lost through lack of CONCENTRICITY in the means of securing the cutting tool to the spindle:

a) Between Spindle ID and Holder OB.
b) Between Holder ID and Collet OD.
c) Between Collet ID and Tool Shank OD.
d) Between Tool Shank OD and cutting edge.

Dirt is one source of error but another is the stamped legend on the shank of a tool - the raised characters (Figure 3-8) can throw a tool off as much as .003". Legend should be etched, not stamped.

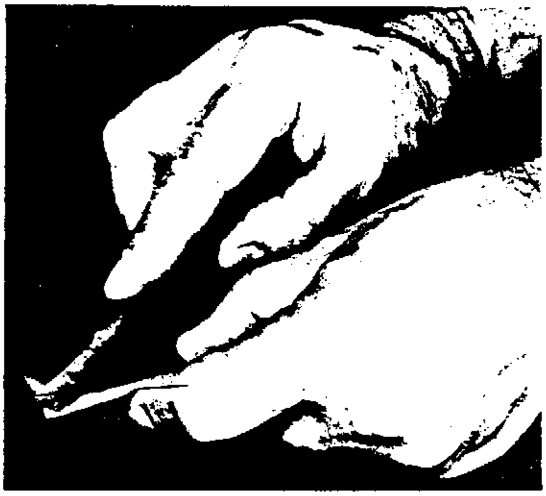


Figure 3-8. Removing Legend

3.1.5 The End Mill

The End Mill is second only to the drill as a metal removing tool in NC Machining. It might be noted in passing that, although the arbor-mounted milling cutter will remove more chips per dollar in production, the End Mill with its extreme versatility is the real key to NC milling. Because of its superior construction, the NC machine has the capability of bringing the cutting tool to the workpiece with more accuracy, more horsepower, and greater rigidity than a conventional machine tool. There are limiting factors however: the clamping, banking and fixturing setup, the tool holder and the End Mill itself.

The fact that the programmer works to an assumed cutter diameter introduced some significant features in peripheral end milling practice:

1. Deflection can be controlled by using the shortest possible end mill.

2. Wear caused by chatter can be limited by using the shortest possible flute length (Figure 3-9). For example: In tests on a superalloy, an end mill with 2" flute length traveled 2" before a regrind was necessary whereas a similar tool with 1" length of flute traveled 85".

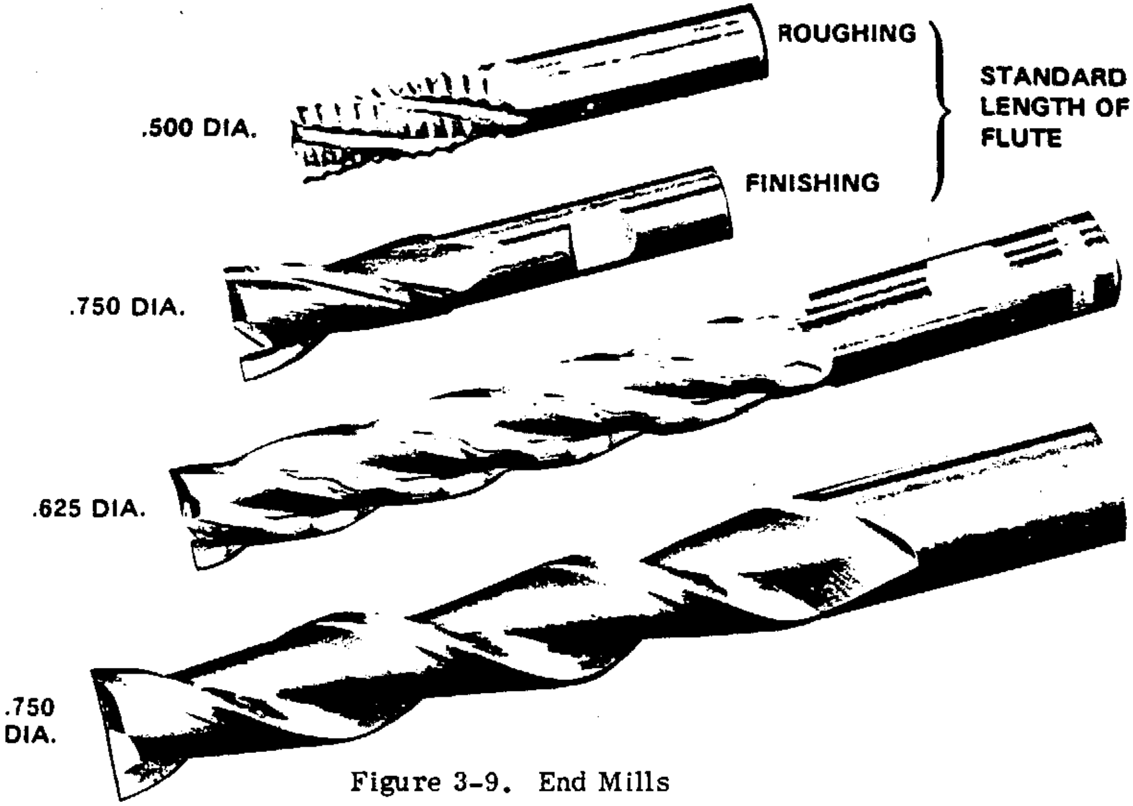


Figure 3-9. End Mills

3. When a tool is reground, it should be used for NC roughing and elsewhere in the plant's general production.

4. If the NC machine is to use reground cutters, proceed as follows:

   a) Purchase all end mills .003" to .005" oversize - this is a general manufacturer's standard - see Specification: USAS B94.19-1968.

   b) Program nominal diameter plus .004".

   c) When worn, regrind to .005" increments.

   This will permit the doubling of EM life for ± .010" type parts. Note also that when regrinding:

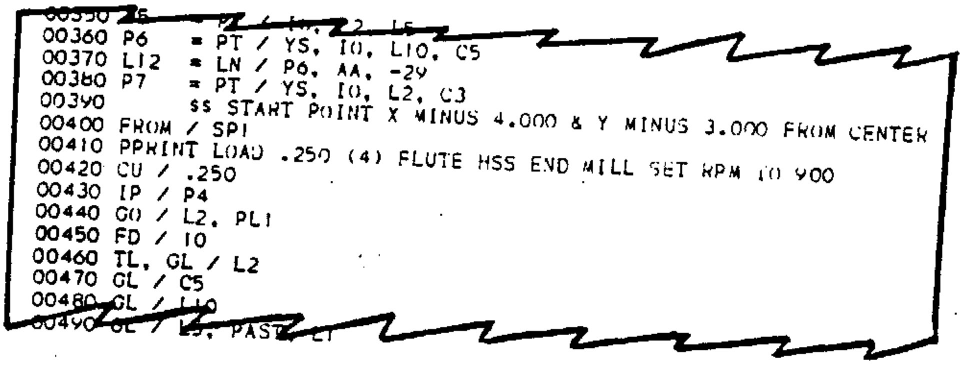
   d) Regrind the flute face to minimize diameter change with possible loss of radial rake. However, be aware that this might generate a taper if the helix angle is incorrect.

e) All regrinds to be fixed increments less than nominal (.005", .010", .020").

f) These regrinds can be sorted and color coded.

g) Color code the tapes that match the special diameters.

NOTE: When preparing tapes using computer assist, another tape is readily available after a one line change in the part program input listing (Figure 3-10).



```
00360 P6    = PT / YS, IO, L10, C5
00370 L12   = LN / P6, AA, -29
00380 P7    = PT / YS, IO, L2, C3
00390         $$ START POINT X MINUS 4.000 & Y MINUS 3.000 FROM CENTER
00400 FROM / SP1
00410 PPRINT LOAD .250 (4) FLUTE HSS END MILL SET RPM TO 900
00420 CU / .250
00430 IP / P4
00440 GO / L2, PL1
00450 FD / 10
00460 TL, GL / L2
00470 GL / C5
00480 GL / L10
```

Figure 3-10. Computer-Assisted Program Input Listing

5. The web thickness of end mills has been increased to 60 percent from 50 percent in the last decade. This will generate the need to compromise the regrind diameter against the reduced chip clearance. The major requirement remains - that of a stiffer end mill and more rigid setup.

6. The recent additions of inserted carbide blades, either straight or with a helix, have considerable merit along with those types with square or triangular inserts.

The following operations can be performed with an end mill:

Slot Milling
Peripheral End Milling
Flat Facing
Face Grooves
Recesses
Pockets
Thread Milling

Boring (peripheral)
Counterboring
Spot Facing
Deburring
Chamfering
Boring (single point)

The following notes are made to explain certain of these operations:

Thread Milling: This operation is similar to peripheral end milling with a multi-tooth milling cutter on an arbor. Since it is not possible to obtain a true thread form of 60° in a helix, no better than Class 2 threads can be expected. Such applications will be confined to 2" diameter threads and larger where the cost of a tap would be prohibitive. Three axis contouring is programmed (see page 9-22).

Boring: Single or multiple point with an end mill is used to straighten a drilled hole, if required, prior to reaming. Peripheral end milling of a drilled hole to open it to a bore size is practical for roughing and semi-finishing where excessive tool deflection is not encountered (Depth = 75 percent of tool diameter). See Figure 3-11.

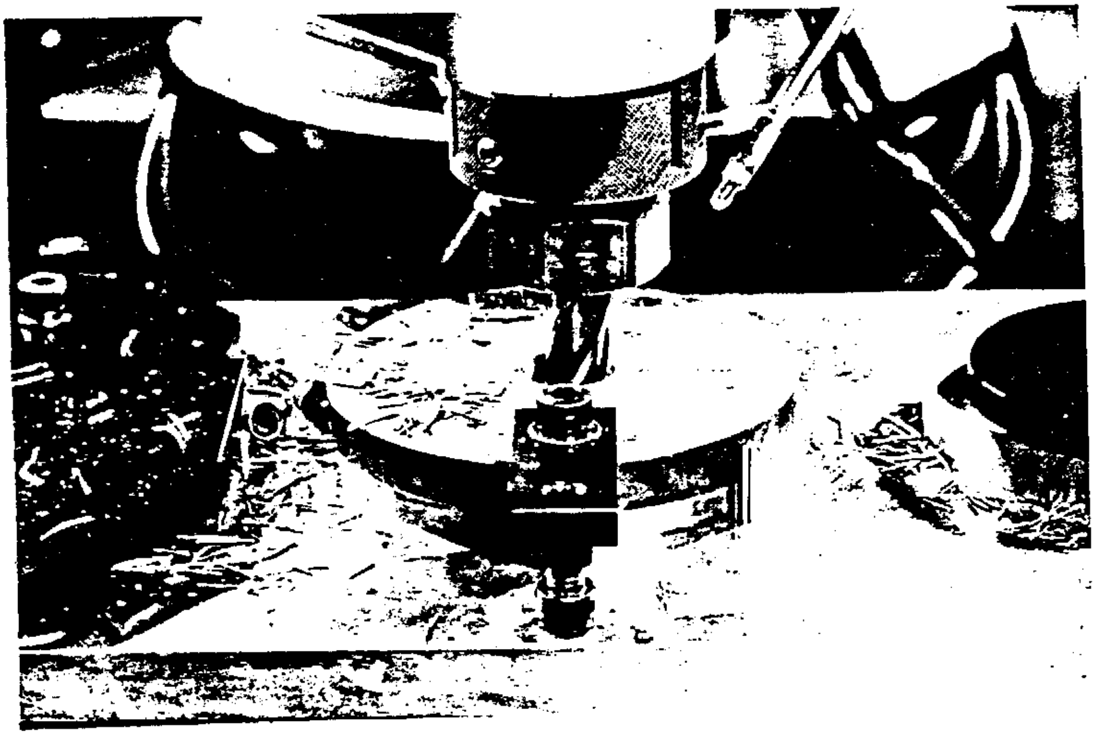


Figure 3-11. Boring With End Mill

3.1.6 Tool Deflection

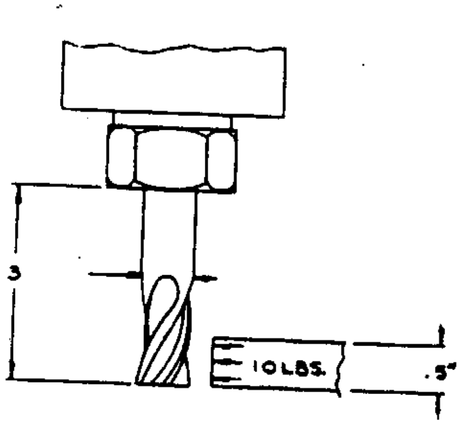
There are countless reminders in all good machining practice to use short, large diameter, rigid tools to minimize deflection (Figures 3-12).

UNDER UNIFORM 10 LB. LOAD OVER 0.5" AS SHOWN. AT TIP OF TOOL **MAX DEFLECTION = 0.001"**.

NOTE: IF A 10 LB. LOAD IS CONCENTRATED AT THE TIP (BORING). AT TIP OF TOOL **MAX DEFLECTION = 0.002"**.

IN BOTH CASES:

*DEFLECTION $\propto$ SIDE LOAD



ASSUME EM TO
A UNIFORM CYLI
DRICAL ROD OF
0.5" DIA.

MOMENT OF INER
TIA = 0.003 ins^4.

Figure 3-12. Tool Deflection

Tool Deflection is proportional to L^3/D^4.

Thus an end mill or boring bar 4x longer than another of the same diameter has to be 64x less rigid, i.e., will result in 64x the deflection. Similarly, the larger the diameter the less deflection.

Also, Tool Deflection is proportional to 1/E, the modulus of elasticity of the cutting tool material. For example:

E = 30,000,000 psi for Steel
E = 94,000,000 psi for Carbide

Obviously a carbide end mill is 3x more rigid, a fact which may become important in maintaining straight perpendicular walls when peripheral end milling.

3.2 TOOL HOLDERS

3.2.1 General Description

All cutting tools are adapted to the machine spindle with tool holders which have common features as shown in Figure 3-13.

---

*$\propto$ = varies as

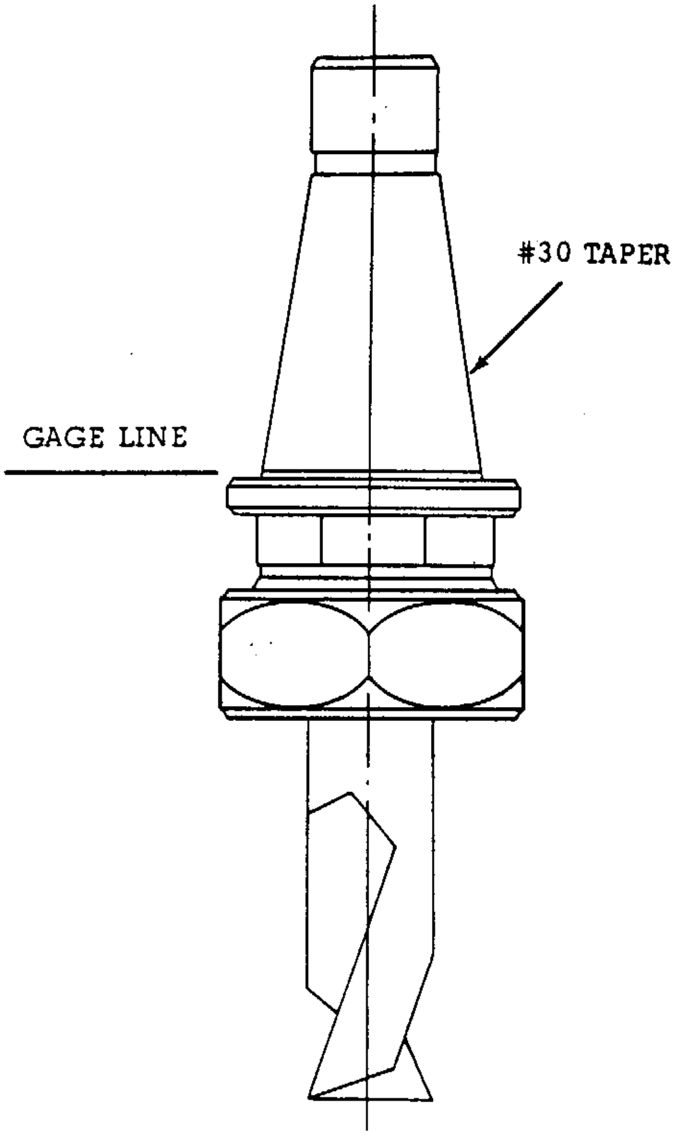


Figure 3-13. Typical Tool and Tool Holder

3.2.2 Tool Setting

When a cutting tool is inserted into its tool holder, the cutting edge and/or tip is located at an imprecise distance from the gage line of the holder. Subsequent programmed Z axis motions with that tool installed are equally inaccurate. Two versatile methods of control of Z axis depth are available with the Series I CNC, both of which are simple and accurate. The next section on OFFSET discusses two methods but it is necessary in this paragraph to discuss the two approaches as far as the tool holder is concerned.

a. Random Set Tools

If a series of cutting tools is selected for a job (partially completed form F115, Figure 3-14) they can be assembled in their holders in random manner as far as length is concerned. It is, however, advisable after assembly to examine these in a row on a bench and to readjust the extremely long or short to approximate the length of the others. By doing this, all tools will tend to extend the same distance in the spindle and with the proper height of the knee (controlled by the longest tool) minimum quill travel will be obtained with each tool and, therefore, minimum cycle time.

b. Preset Tools

Extending the above, one can complete the form F115 using all the columns available. The information for this calculation originates in the Tooling Manual, M122 for the tool holders and in catalog data for the cutting tool itself.

It is also possible that under certain circumstances a greater degree of precision is required in the presetting of tools for several reasons: length of cut must be a certain minimum; the possibility of interference between a portion of the holder and the workpiece or its clamp; or the need to standardize tool-tool holder combinations. In this event, forms F-125 are available for this purpose. An extract is shown in Figure 3-15, showing a picture of a tool holder for a spindle having a power drawbar - the setting information is the same.

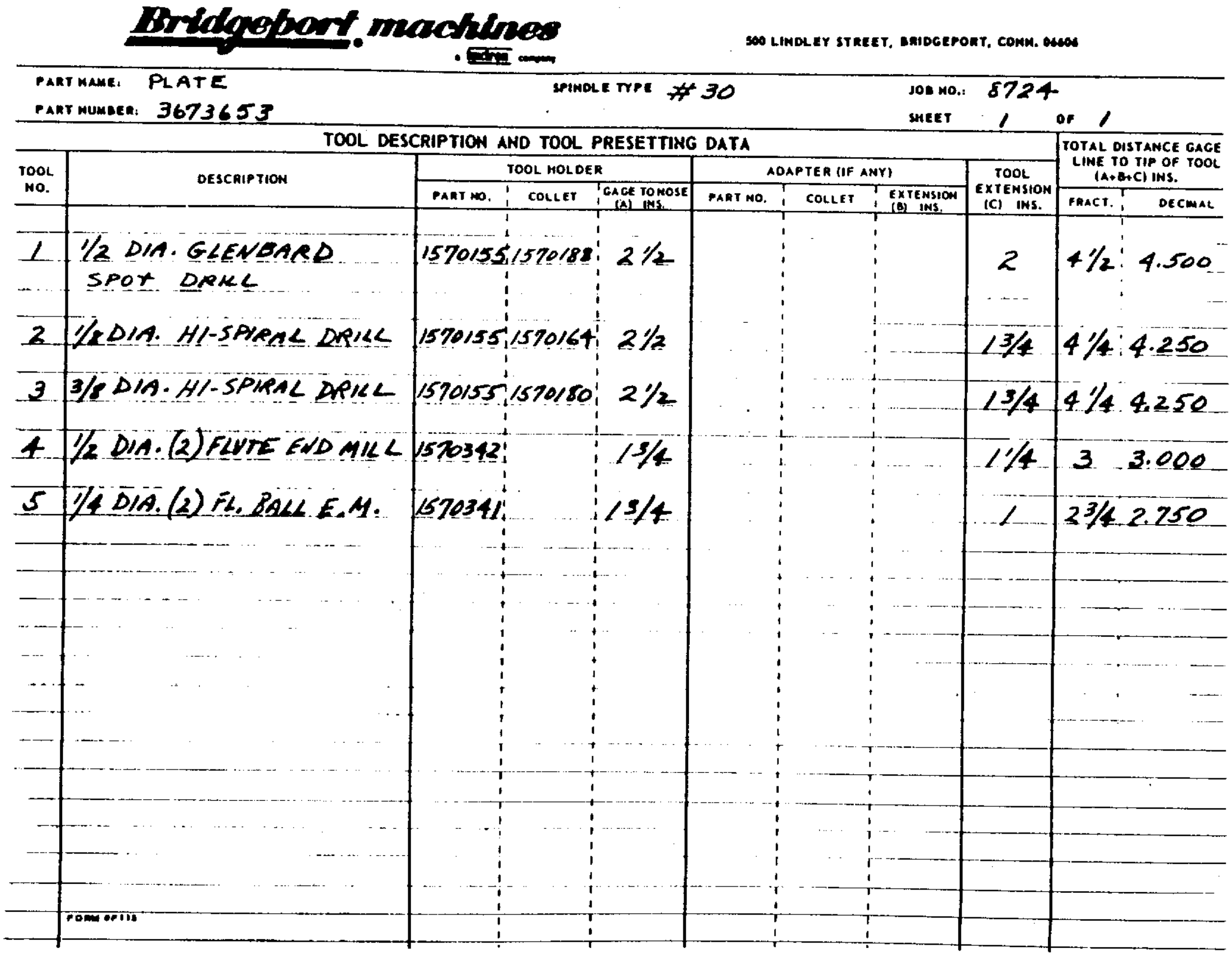
Suggested methods and procedures for tool setting are made available to reduce the time to make the necessary adjustments and to systematize the actions of all those involved. The forms used in several examples in this manual are, therefore, the important means of communication between Departments.

3.2.3 Tool Length Offset

The Electronic Industries Association (EIA) described OFFSET as a correction parallel to a controlled axis; OFFSET is not to be confused with COMPENSATION which is defined as a displacement normal to the cutter path.

Tool Length Offset on the Series I CNC has the capability (within reasonable limits) of counteracting the effects of a change in pre-programmed tool length without alteration to the corresponding pre-punched tape program. Tool Length Offset is unique to the tool in the spindle.

The circumstances under which Tool Length Offset is found most valuable are as follows:



Bridgeport machines

a [illegible] company

500 LINDLEY STREET, BRIDGEPORT, CONN. 06606

PART NAME: PLATE
SPINDLE TYPE #30
JOB NO.: 8724
PART NUMBER: 3673653
SHEET 1 OF 1

TOOL DESCRIPTION AND TOOL PRESETTING DATA

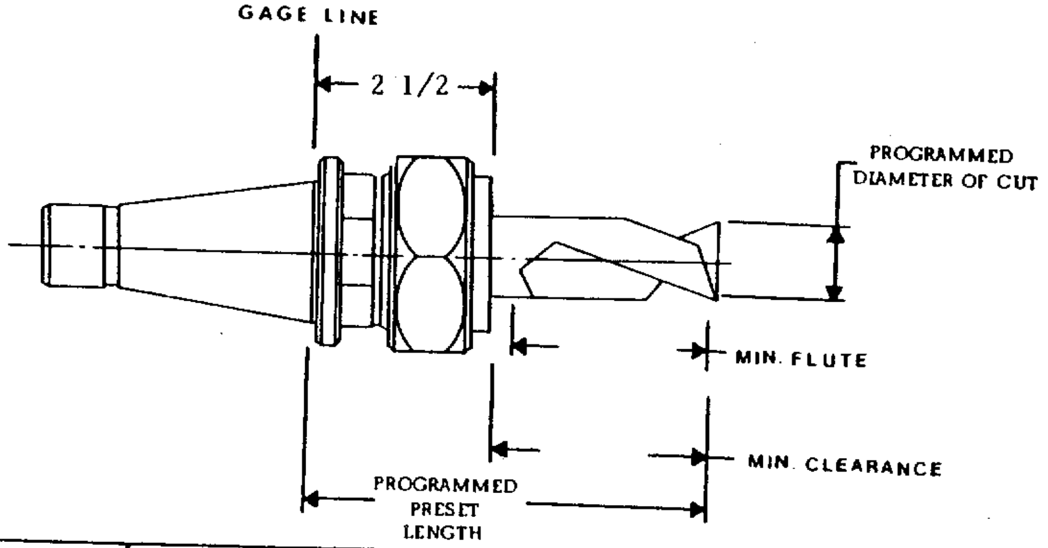
TOOL NO.	DESCRIPTION	TOOL HOLDER			ADAPTER (IF ANY)			TOOL EXTENSION (C) INS.	TOTAL DISTANCE GAGE LINE TO TIP OF TOOL (A+B+C) INS.	
		PART NO.	COLLET	GAGE TO NOSE (A) INS.	PART NO.	COLLET	EXTENSION (B) INS.		FRACT.	DECIMAL
1	1/2 DIA. GLENBARD SPOT DRILL	1570155	1570188	2 1/2				2	4 1/2	4.500
2	1/8 DIA. HI-SPIRAL DRILL	1570155	1570164	2 1/2				1 3/4	4 1/4	4.250
3	3/8 DIA. HI-SPIRAL DRILL	1570155	1570180	2 1/2				1 3/4	4 1/4	4.250
4	1/2 DIA. (2) FLUTE END MILL	1570342		1 3/4				1 1/4	3	3.000
5	1/4 DIA. (2) FL. BALL E.M.	1570341		1 3/4				1	2 3/4	2.750

FORM OP115

Figure 3-14. Tool Description and Presetting Data

Bridgeport machines • textron company 500 LINDLEY STREET, BRIDGEPORT, CONN 06606

CHANGE	DATE	TOOL IDENTIFICATION END MILL



DURABLE TOOLING			PART NO.	I.D.NO	PART NO.	I.D.NO	PERISHABLE TOOLING
HOLDER	TYPE	SIZE					

NOTES

MANUFACTURER

TOOL CODING

TOOL SETTING DRAWING

PURCHASE ORDER NO

DRAWN BY

DATE

F-143

Figure 3-15. Tool Setting Drawing

- Cutter preset dimension is not exact. This is particularly effective with end mills in end mill holders and also with tools which draw up into a collet.
- Change of reference plane.
- Broken cutter replacement.
- Dull cutter reground and replaced.

As an example (Figure 3-16) assume that Tool #1 has a travel of -2.000" from the upstop (Z axis home) and that Tool #2 has a travel of -(2.000" + 1.500") from the same upstop. Further, consider the fact that the programmer requires both tools to do work on the piece part at the same reference plane. It is, therefore, evident that the programmer must know exactly where each tool tip is with reference to the only item that is common to both - the gage line. The programmer, therefore, must call for a specific Z axis motion on Tool #2 greater than Tool #1 if accuracies are to be achieved. Alternatively, a length offset capability in the control can take care of the differences.

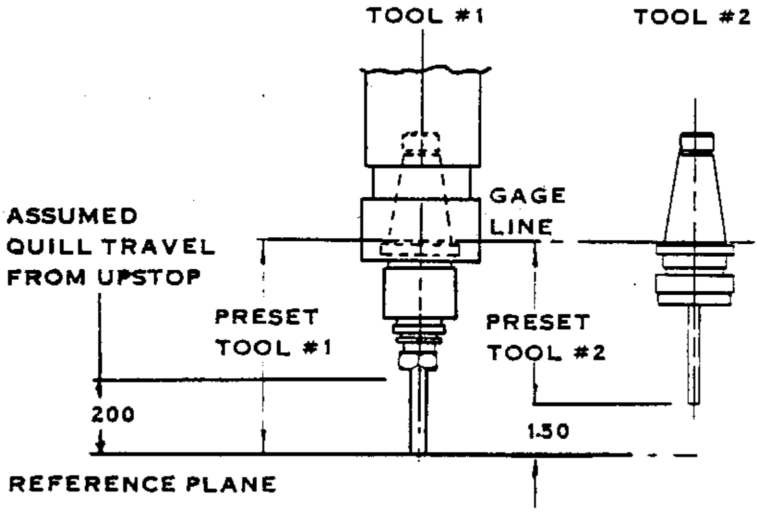


Figure 3-16. Tool Length Differences

<u>First Method:</u> The Z axis can be jogged into position and a button depressed which will automatically store the amount moved (Figure 3-17).

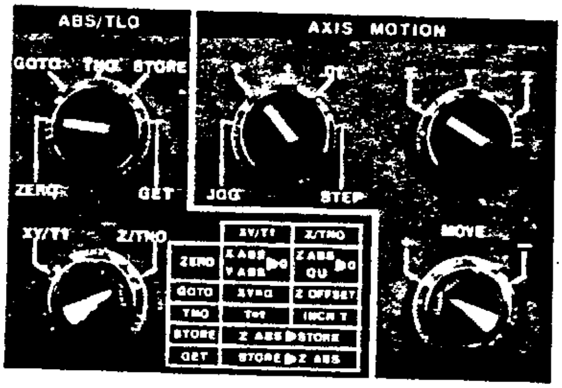


Figure 3-17. Tool Length Offset Controls

Second Method: The measurement of the actual preset tip of the tool from a gage line (Figure 3-18) can be compared with the required preset distance and the difference (THE OFFSET) input to the machine. Such method of input is by keyboard to the storage within the control.

NOTE: The stored amount is volatile; that is to say, it disappears if power is disconnected. It is recommended, therefore, that a record of offsets be made.

The OFFSET value should be checked before operating the machine. The stored TLO data can be retrieved and displayed on the Universal Data Readout.

Tool Length Offset when properly used will achieve an accurate reference plane, within the resolution of the machine and ultimately control the finished depths in a machined part. Tool length precision is obtained by measurement on the machine tool, by mechanical height gage (Figure 3-19) or by optical methods.

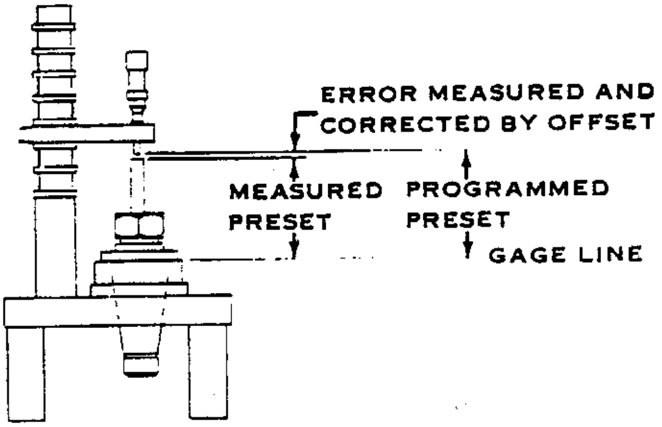


Figure 3-18. Preset

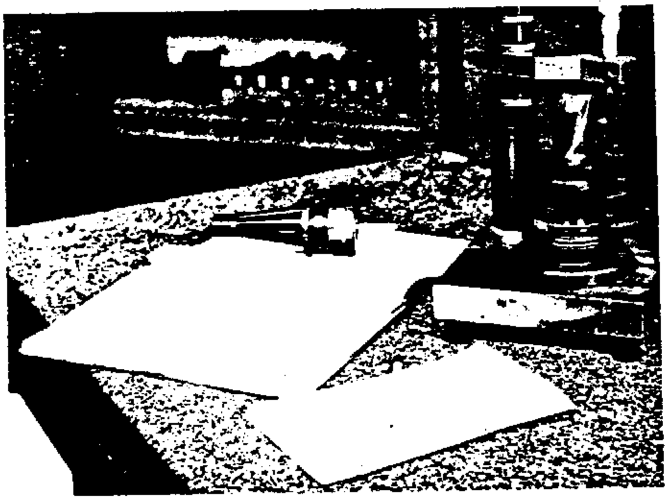


Figure 3-19. Measuring Actual Preset

Presetting is still important despite the convenience and accuracy obtainable with this offset. Some degree of preset is required to ensure that the chosen tool is used with the precise diameter of cut and geometry, and also that the correct length of cut and clearance with holder prescribed by the programmer is used to perform the task appointed for it (Figure 3-20).

Accumulation of the Preset Summary Data can now be made to establish a listing of all the TLO values for the tools on the job to be run (Figure 3-21).

3.2.4 Tool Length Offset Usage

The intent of TOOL LENGTH OFFSET is to provide a means of adjustment for the difference between the programmed setting length and the actual set length of the tool being used. However, this does not mean that proper set distances as required in the program need not be adhered to; but rather assist the combined efforts of both the programmer and operator in their effort to meet exceptional tolerances which without the aid of TOOL LENGTH OFFSET would be difficult to obtain or maintain due to circumstances of tool wear, regrind, etc.

The following is a list of conditions describing the use of TOOL LENGTH OFFSET:

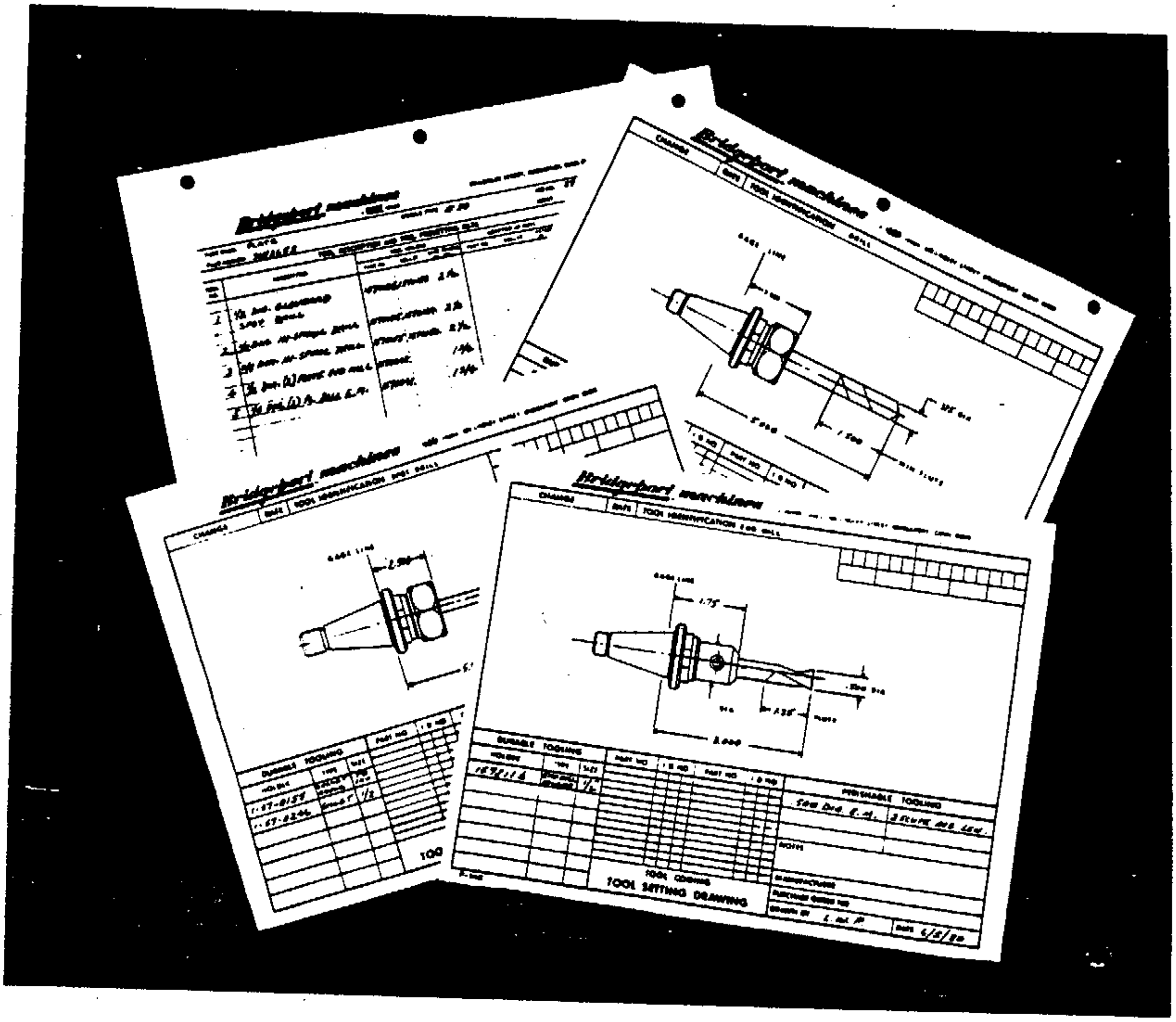
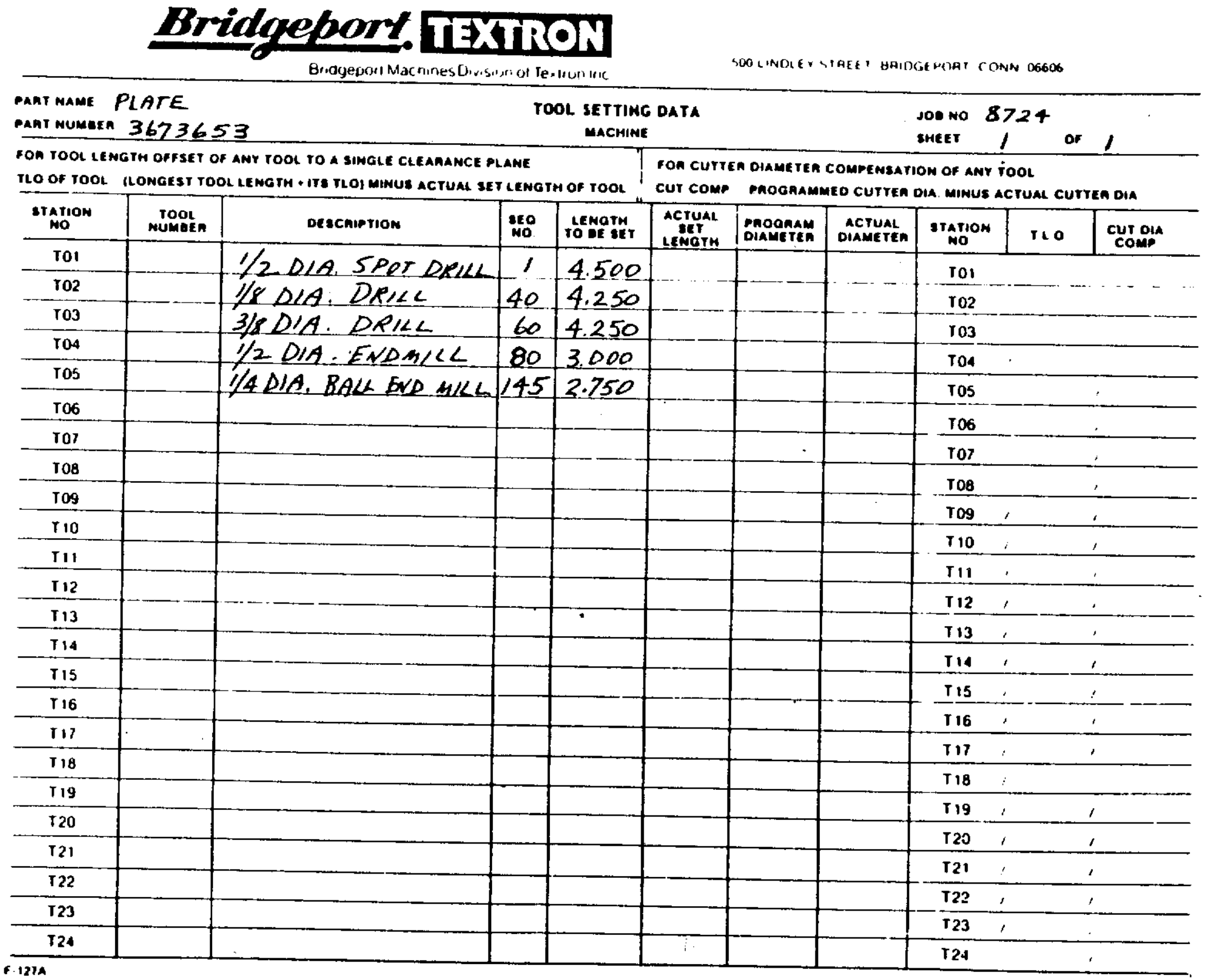


Figure 3-20. Tool Setting Drawings

a. Tool length offset is automatically called out with the coded tool select number and M6 as a specific designator at the end of a tool transfer. The Z axis absolute register is set to the tool length offset value.

b. The tool length offset is the distance from the bottom of the tool in the Z axis home position to a convenient clearance plane. This establishes the TLO reference plane. The reference system automatically established by the control is that Z absolute=0" at the TLO reference plane. Thus, the absolute Z axis register is preset to a positive value.

NOTE: The Z axis home (quill retracted) position is set by a mechanical switch on the machine and is independent of the contents of the Z absolute register.



Bridgeport TEXTRON

Bridgeport Machines Division of Textron Inc

500 LINDLEY STREET BRIDGEPORT CONN 06606

PART NAME PLATE

PART NUMBER 3673653

TOOL SETTING DATA

MACHINE

JOB NO 8724

SHEET 1 OF 1

FOR TOOL LENGTH OFFSET OF ANY TOOL TO A SINGLE CLEARANCE PLANE

TLO OF TOOL (LONGEST TOOL LENGTH + ITS TLO) MINUS ACTUAL SET LENGTH OF TOOL

FOR CUTTER DIAMETER COMPENSATION OF ANY TOOL

CUT COMP PROGRAMMED CUTTER DIA. MINUS ACTUAL CUTTER DIA

STATION NO	TOOL NUMBER	DESCRIPTION	SEQ NO.	LENGTH TO BE SET	ACTUAL SET LENGTH	PROGRAM DIAMETER	ACTUAL DIAMETER	STATION NO	T L O	CUT DIA COMP
T01		1/2 DIA. SPOT DRILL	1	4.500				T01		
T02		1/8 DIA. DRILL	40	4.250				T02		
T03		3/8 DIA. DRILL	60	4.250				T03		
T04		1/2 DIA. ENDMILL	80	3.000				T04		
T05		1/4 DIA. BALL END MILL	145	2.750				T05		
T06								T06		
T07								T07		
T08								T08		
T09								T09		
T10								T10		
T11								T11		
T12								T12		
T13								T13		
T14								T14		
T15								T15		
T16								T16		
T17								T17		
T18								T18		
T19								T19		
T20								T20		
T21								T21		
T22								T22		
T23								T23		
T24								T24		

F-127A

Figure 3-21. Tool Setting Data, F-127

c. Offsets are executed on the next Z axis block after a tool change command providing that the next Z motion block is programmed in the G90 (absolute) mode.

For example: G0G90Z0 would rapid traverse the quill to the TLO Reference plane.

or: G0G90Z-1.0 would rapid traverse the quill to a plane 1.0" below the TLO reference plane.

d. Once at the TLO reference plane, the programmer can, if desirable, insert another Z axis reference coordinate system by using a G92 (preset absolute register) command.

e. A total of 24 offsets are provided. The range of entry is .001 to 9.999 inches.

f. Offset values are inserted either via tape, MDI keyboard or the TOOL OFFSET operator's controls. Offset values can also be inserted via CRT or teleprinter. The format Tn/Z is used.

NOTE: Tool offsets embedded in the program text will override any MDI input values.

g. An offset once inserted is continuously associated with that tool until a new entry is made.

3.3 MACHINABILITY

3.3.1 Rigidity

To utilize the Torque/HP characteristics of the Series I CNC machine in conjunction with the specified accuracies, it is essential that total rigidity of all items be maintained.

- Workpiece supported for maximum stiffness
- Part surfaces flat and parallel
- Tool deflection held to a minimum (refer to paragraph 3.1.6)
- Correct tool geometry

3.3.2 Speeds and Feeds

In addition to the selection of proper tooling for the part, it is essential that the programmer have a basic knowledge of feed and speed requirements. It is often that special materials will give rise to question on feeds and speeds - a recommended reference work in this all important subject is:

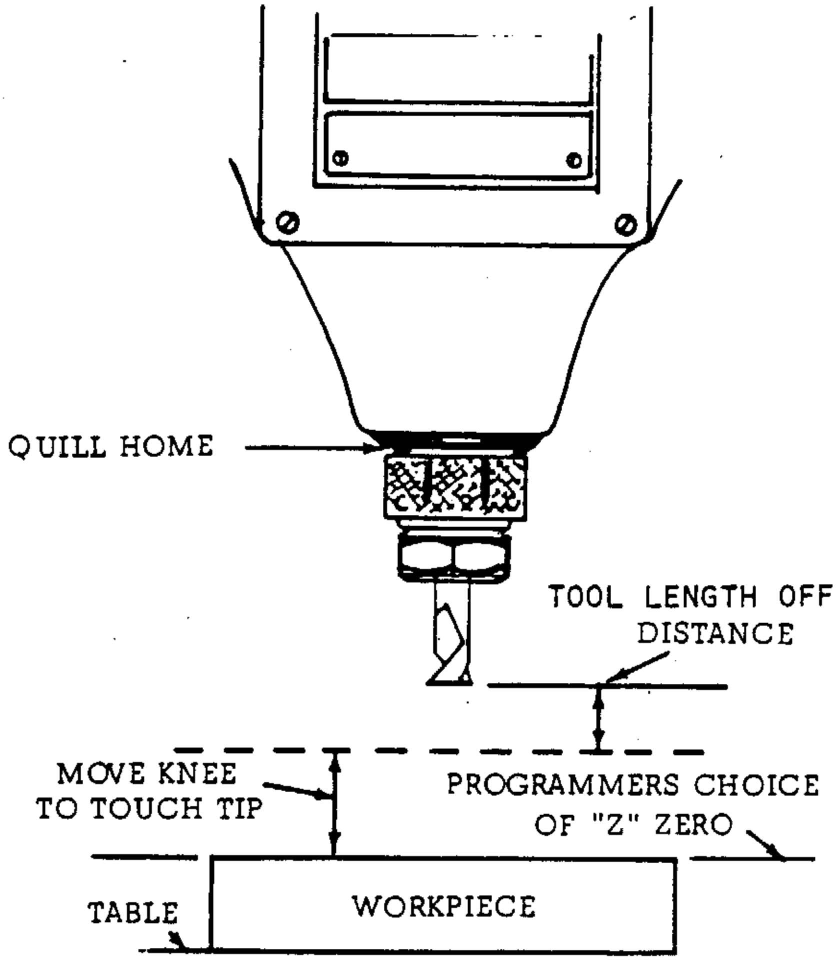


Figure 3-22. Tool Length Offset

Tool Length Offset for a Particular Tool:

This is the distance from the tip of that selected tool with the spindle in the home position, to the selected "Z" Zero Plane, wherever the "Z" Zero Plane has been selected for that job. See Figure 3-22.

To Set the Knee to the Zero Plane:

Place the longest tool to be used in the current job in the spindle. Call this tool out with its TLO value (which must be a minimum of .500"). Depress GOTO. The quill will move down .500". Loosen the clamps on the knee and raise it to touch the "Z" Zero Plane on the workpiece to the tip of the tool. Lock the knee again; the machine Z axis and knee are now set for the job.

MACHINING DATA HANDBOOK. (METCUT Research Associates, Inc., 3980 Rosslyn Drive, Cincinnati, Ohio 45209). The machining data contained in this handbook provides starting recommendations for all types of material removal operations and materials. Data for the handbook was collected with the help of the U.S. Army Weapons Command Metal Cutting Advisory Committee and approximately 250 companies in industry.

To assist the programmer's task, Bridgeport Machines has convenient forms available to record the basic decisions by the programmer before commencing the actual program manuscript. See form F114 (Figure 3-23).

3.3.3 Machine Capabilities

The following operation with feedrates and spindle speeds are examples of the performance of the Series I CNC. The contrast that is evident is an illustration of the versatility of the machine.

	IPM	RPM
Drill #80 holes in Mild Steel	2.7	4200
Drill 3/4 dia. holes in Mild Steel	3.0	440
Tap 1/4-20 holes in Alum. & Cast Iron	25.0	500
Mill 4.5 in^3 /min. in Aluminum (1" dia. x 1/2 dp x 3/4 wide)	18.0	1200
Mill 1.25 in^3 /min. in Mild Steel (1/2" dia. x 1/2 dp x 1/4 wide)	14.0	900
Mill .75 in^3/min. in AISI 4140 Rc 25 (3/4 dia. X 1/2 dp slot)	2.0	350
Pocket Mill with 5/8 in. dia. Serrated Tooth 4 Flute EM AISI 4140 Rc 25 3/4 Deep X .300 stepover	2.5	380

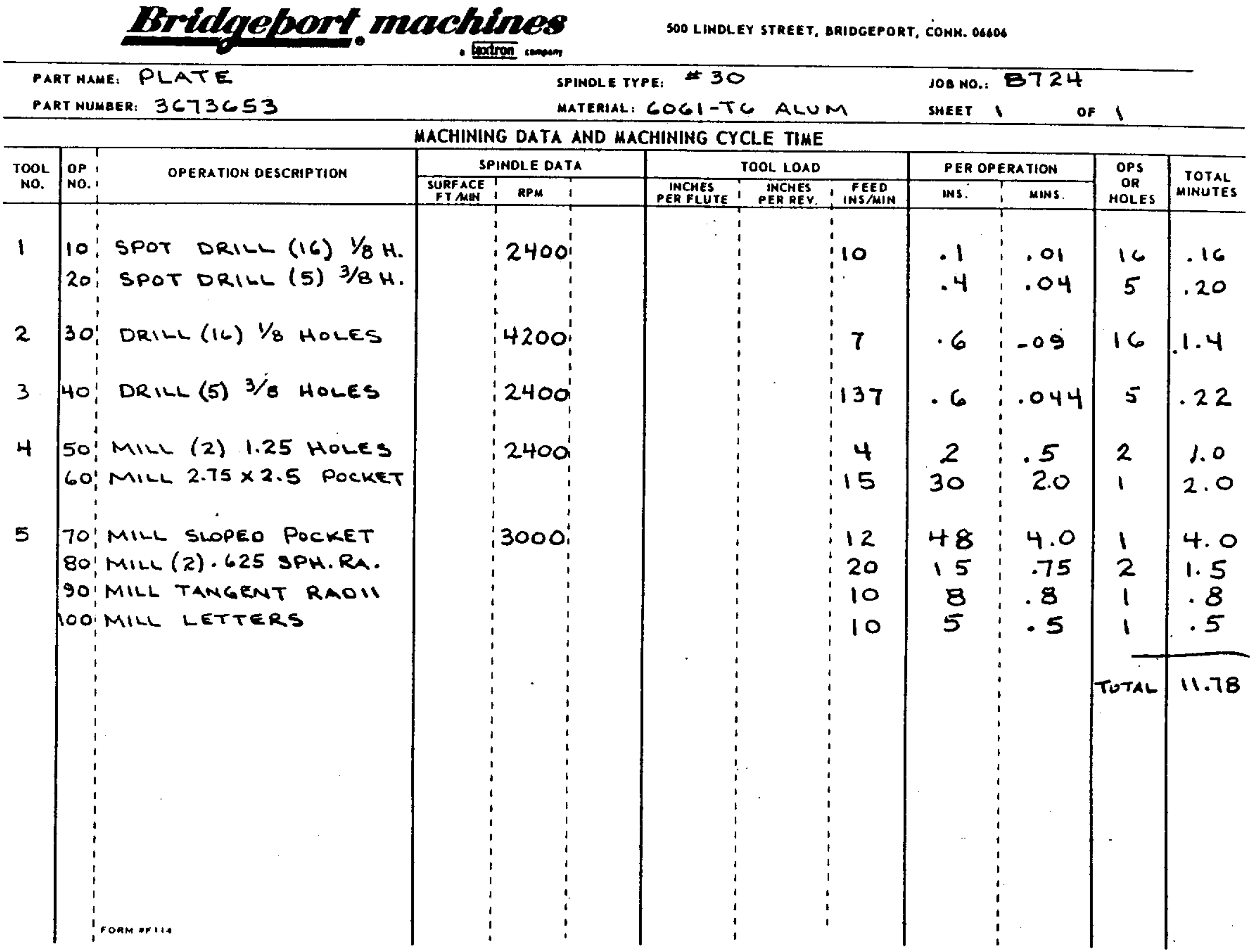
	TYPE OF	SURFACE SPEED		POWER	FEED PER TOOTH	
MATERIAL	MILL CUT	HSS	CARB.	FACTOR	HSS	CARB.
Alum.alloys wrought.Solut. treated aged	SLOT	250-280	---	3.0	.003-.004	---
	PERIPH.	600-800	1000-1300		.007-.010	.007-.010
	FACE	800-1200	MAX		.010-.022	.010-.020
Alum.alloys as cast.	SLOT	300-350	---	3.0	.004-.005	---
	PERIPH.	800-1000	1000-1300		.007-.010	.007-.010
	FACE	1000-1500	2400-2800		.010-.022	.010-.020
Magnesium alloys all	SLOT	450-500	---	4.0	.004-.005	---
	PERIPH.	800-1000	1300		.008-.011	.009-.012
	FACE	900-1500	MAX		.010-.022	.010-.020
Brass	SLOT	165-185	---	2.0	.004-.005	---
	PERIPH.	275-350	575-750		.006-.008	.007-.009
	FACE	325-425	600-800		.008-.014	.008-.014
Soft Carbon Stl. 1029 also Leaded	SLOT	100-110	---	.7	.0025-.003	---
	PERIPH.	160-210	400-520		.005-.006	.007-.008
	FACE	160-210	560-725		.008-.012	.008-.014
1030-1055 32Rc 102,000 PSI 275-325 BHN	SLOT	60-70	---	.6	.002-.0025	---
	PERIPH.	70-90	275-360		.004-.005	.005-.006
	FACE	90-110	360-460		.005-.009	.006-.010
1060-1095 42Rc 136,000 PSI 325-375 BHN	SLOT	35-45	---	.5	.002-.0025	---
	PERIPH.	60-70	225-250		.004-.005	.005-.006
	FACE	65-85	290-330		.005-.008	.006-.007
Med.& High Car. 49Rc 168,000 PSI	SLOT	25-30	---	.4	.001-.001	---
	PERIPH.	45-55	95-120		.001-.0015	.001-.002
	FACE	20-25	130-170		.002-.003	.004-.005
High Car. Mart. 54Rc & UP. 180,000 PSI up.	SLOT	---	20-25	.3	---	.001-.002
	PERIPH.	---	55-70		---	.001-.0015
	FACE	---	60-75		---	.004-.005
STAINLESS STEEL Free mach.anne. or cold drawn.	SLOT	75-85	---	.8	.002-.0025	---
	PERIPH.	125-160	350-455		.004-.005	.006-.007
	FACE	135-175	475-610		.006-.008	.010
St.Stel.,300 ser. Cold Drawn,Aust. 225-275 BHN	SLOT	65-75	---	.5	.002-.0025	---
	PERIPH.	110-145	325-420		.004-.005	.006-.007
	FACE	90-115	330-440		.005-.006	.008-.010
St.Stl.,17-4PH Annealed 150-200 BHN	SLOT	55-60	---	.7	.0015-.002	---
	PERIPH.	90-100	275-360		.003-.004	.005-.006
	FACE	85-110	375-390		.005-.006	.008-.010
St.Stl.,17-4PH Hardened 325-375 BHN	SLOT	40-45	---	.5	.0015-.002	---
	PERIPH.	65-80	190-260		.002-.003	.003-.004
	FACE	55-70	220-260		.004-.005	.005-.007

$$\text{Required HP} = \frac{\text{CU. IN./min.}}{\text{P}} \qquad \text{Max. Cu. In./min.} = \text{Available H.P.} \times \text{P}$$

MATERIAL	TYPE OF MILL CUT	SURFACE SPEED		POWER FACTOR	FEED PER TOOTH	
		HSS	CARB.		HSS	CARB
Titanium,Commer. pure,Annealed 140-200 BHN	SLOT	85-95	---	.7	.005-.006	---
	PERIPH.	140-170	350-400		.007	.007-
	FACE	110-145	330-440		.004-.006	.004-
Titanium, 5Al-2.5 SN Annealed	SLOT	50-55	---	.4	.004-.005	---
	PERIPH.	100-110	275-300		.005-.006	.007-
	FACE	55-70	240-295		.004-.006	.004-
Titanium, 6Al-4V Annealed 310-350 BHN	SLOT	55-60	---	.4	.004-.005	---
	PERIPH.	90-100	260-290		.005-.006	.007-.
	FACE	45-55	145-185		.004-.006	.004-.
Titanium,6Al-4V Solution treated 320-380 BHN	SLOT	45-50	---	.3	.004-.005	---
	PERIPH.	75-90	200-225		.004-.006	.006-.
	FACE	150-550	120-160		.004-.006	.004-.
Titanium,Sol.Tr. 6Al-6V-2SN 375-420 BHN	SLOT	45-50	---	.3	.004-.005	---
	PERIPH.	70-80	175-200		.004-.006	.006-.
	FACE	30-40	90-115		.004-.007	.004-.
Hasteloy X,Inconel Nickel base,wrought Cold Drawn	SLOT	13-15	---	.4	.003-.004	---
	PERIPH.	20-30	60-90		.002-.004	.002-.
	FACE	20-25	60-65		.004-.005	.005-.
Inconel X Inconel 718,High temp.alloys	SLOT	7-8	---	.3	.002-.003	---
	PERIPH.	12-15	50-60		.002-.003	.002-.
	FACE	25-30	60-70		.004-.006	.005
Monel,Nickel all. wrought & cast 115-240 BHN	SLOT	30-33	---	.5	.004-.005	---
	PERIPH.	65-85	200-250		.004-.005	.004-.0
	FACE	65-85	130-175		.006-.008	.006-.0
Tool St.,S1-S7 L2,L3,L6,Annealed 175-225 BHN	SLOT	50-65	---	.5	.002-.0025	---
	PERIPH.	85-110	365-485		.004-.005	.005-.0
	FACE	120-145	385-510		.006-.010	.008-.0
Tool St.,A2-A10 01-07,ANNEALED 200-250 BHN	SLOT	45-50	---	.5	.0015-.002	---
	PERIPH.	75-90	300-390		.003-.004	.004-.0
	FACE	95-130	330-440		.005-.007	.006-.0
Tool St.,D2-D7 Annealed 200-250 BHN	SLOT	35-40	---	.4	.0015-.002	---
	PERIPH.	55-65	200-260		.003-.004	.004-.0
	FACE	55-70	195-255		.006-.008	.008-.01
Cast Iron,Gray As Cast,A48 Class 30,35,40,pearlitic	SLOT	55-60	---	1.0	.002-.003	---
	PERIPH.	90-125	325-500		.004-.005	.007-.00
	FACE	90-120	385-510		.010-.013	.010-.01
	SLOT					
	PERIPH.					
	FACE					

Required HP = $\frac{\text{CU. IN./min.}}{P}$

Max. Cu. In./min. =Available H.P. x P



Bridgeport machines a textron company

500 LINDLEY STREET, BRIDGEPORT, CONN. 06606

PART NAME: PLATE
PART NUMBER: 3673653
SPINDLE TYPE: # 30
MATERIAL: 6061-T6 ALUM
JOB NO.: B724
SHEET 1 OF 1

MACHINING DATA AND MACHINING CYCLE TIME

TOOL NO.	OP NO.	OPERATION DESCRIPTION	SPINDLE DATA			TOOL LOAD			PER OPERATION		OPS OR HOLES	TOTAL MINUTES
			SURFACE FT/MIN	RPM		INCHES PER FLUTE	INCHES PER REV.	FEED INS/MIN	INS.	MINS.		
1	10	SPOT DRILL (16) 1/8 H.		2400				10	.1	.01	16	.16
	20	SPOT DRILL (5) 3/8 H.							.4	.04	5	.20
2	30	DRILL (16) 1/8 HOLES		4200				7	.6	.09	16	1.4
3	40	DRILL (5) 3/8 HOLES		2400				137	.6	.044	5	.22
4	50	MILL (2) 1.25 HOLES		2400				4	2	.5	2	1.0
	60	MILL 2.75 x 2.5 POCKET						15	30	2.0	1	2.0
5	70	MILL SLOPED POCKET		3000				12	48	4.0	1	4.0
	80	MILL (2) .625 SPH. RA.						20	15	.75	2	1.5
	90	MILL TANGENT RADII						10	8	.8	1	.8
	100	MILL LETTERS						10	5	.5	1	.5
											TOTAL	11.78

FORM #F114

Figure 3-23. Machining Data Summary, F-114