

AD-A092 929

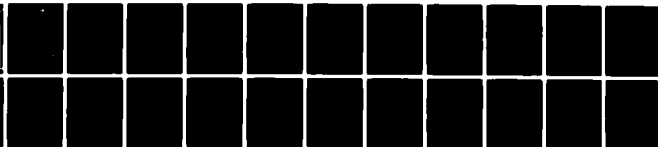
NATIONAL BUREAU OF STANDARDS WASHINGTON DC
GUIDELINES FOR EXCHANGEABLE APT DATA PACKAGES. APT PART PROGRAM--ETC(U)
JUN 80 B M SMITH
NBSIR-80-2073.2

F/G 9/2

UNCLASSIFIED

NL

100
AL
W/2X-0



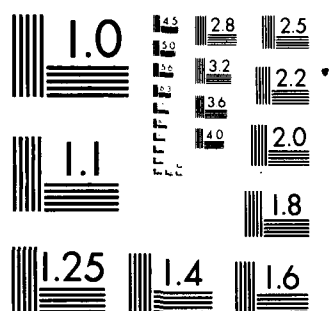
END

DATE

FILED

1 81

DTIC



MICROCOPY RESOLUTION TEST CHART
NATIONAL BUREAU OF STANDARDS 1963-A

AD A092929

REPRODUCED BY
NATIONAL TECHNICAL
INFORMATION SERVICE
U.S. DEPARTMENT OF COMMERCE
SPRINGFIELD, VA 22161

8

U.S. DEPT. OF COMM. BIBLIOGRAPHIC DATA SHEET		1. PUBLICATION OR REPORT NO. NBSIR-80-2073.2		2. Recipient's Accession No. 114696	
4. TITLE AND SUBTITLE 6 Air Force Logistics Command Guidelines for Exchangeable APT Data Packages. APT Part Programmer's Manual.				5. Publication Date 11 June 1980	
7. AUTHOR(S) 10 Mr. Bradford M./Smith				8. Performing Organ. Report No.	
9. PERFORMING ORGANIZATION NAME AND ADDRESS NATIONAL BUREAU OF STANDARDS DEPARTMENT OF COMMERCE WASHINGTON, DC 20234				11. Contract/Grant No.	
12. SPONSORING ORGANIZATION NAME AND COMPLETE ADDRESS (Street, City, State, ZIP) Air Force Logistics Center McClennan Air Force Base California 95652				13. Type of Report & Period Covered 9 Final rept.	
15. SUPPLEMENTARY NOTES This is a companion document to Air Force Logistics Command Guidelines for Exchangeable APT Data Packages, Final Report ☐ Document describes a computer program; SF-185, FIPS Software Summary, is attached.					
16. ABSTRACT (A 200-word or less factual summary of most significant information. If document includes a significant bibliography or literature survey, mention it here.) A method of APT programming and postprocessor design has been developed which permits more efficient data preparation for numerical control (NC) machine tools and then allows this data to be quickly and easily exchanged among different NC machines. This is accomplished through rigorous specification of the APT post-processor language based upon new ANSI standards for APT plus a comprehensive definition of the machining functions which should result from the use of each APT language statement. Individual post-processors are modified to process each statement in the same manner. The concept was successfully demonstrated in production by processing a single APT data package on three different milling-drilling type machine tools. A 23% increase in NC manufacturing efficiency is projected. This document describes the APT post-processor language used, including the syntax and semantics of each statement. It also defines the default conditions under which the post-processor and machine tool operator accomplish the desired function in the absence of the automatic feature.					
17. KEY WORDS (six to twelve entries; alphabetical order; capitalize only the first letter of the first key word unless a proper name; separated by semicolons) APT; Automation; Computer aided manufacturing; NC machining; NC programming; Numerical Control; Part programming					
18. AVAILABILITY ☒ Unlimited ☐ For Official Distribution. Do Not Release to NTIS ☐ Order From Sup. of Doc., U.S. Government Printing Office, Washington, DC 20402, SD Stock No. SN003-003- ☒ Order From National Technical Information Service (NTIS), Springfield, VA. 22161 U.S. Distribution Only			19. SECURITY CLASS (THIS REPORT) UNCLASSIFIED		21. NO. OF PRINTED PAGES
			20. SECURITY CLASS (THIS PAGE) UNCLASSIFIED		22. Price

S DEC 12 1980

240800

USCOMM-DC

alt

Automation Technology Program

NBSIR 80-2073.2

GUIDELINES FOR EXCHANGEABLE APT DATA PACKAGES

APT Part Programmer's Manual

Bradford M. Smith

June 1980

Prepared for the
Air Force Logistics Command
Sacramento Air Logistics Center
PRAM Office



U.S. DEPARTMENT OF COMMERCE, Philip M. Klutznick, *Secretary*

Luther H. Hodges, Jr., *Deputy Secretary*

Jordan J. Baruch, *Assistant Secretary for Productivity, Technology, and Innovation*

NATIONAL BUREAU OF STANDARDS, Ernest Ambler, *Director*

INTRODUCTION

A joint effort between the National Bureau of Standards and the Air Force Logistics Command (AFLC) has developed a method of APT part programming and postprocessor design which allows a part program to be processed for any one of a variety of NC machines with no changes needed in the part program. The concept produces savings in terms of labor content, leadtime and cost of NC part programming. Additionally, it gives shops greater flexibility in scheduling NC facilities and responding to high priority work. Details of this project are given in three companion documents.

Guidelines for Exchangeable APT Data Packages

Final Report - NBSIR 80-2073.1

APT Postprocessor Specifications - NBSIR 80-2073.3

Postprocessor Software Guide - NBSIR 80-2073.4

The approach was successfully demonstrated in production at Sacramento Air Logistics Center by processing a single APT data package on three different machine tools. Only the MACHIN selection statement and the part ORIGIN statement had to be changed. This exchangeability of part manufacturing data was made possible by modifying the postprocessor software so that each APT statement is processed in the same manner for all machine tools. Thus the original intent of the part programmer was satisfied on all three machine tools.

This Part Programmer's Manual describes the APT Postprocessor language used. For each language statement, it defines the proper order and arrangement of modifiers and scalar values. Furthermore, it defines the machining function which will result from the use of each statement. Usually these functions are performed automatically through tape controlled features on the machine tool or controller. However, the manual also specifies default conditions under which the postprocessor and machine tool operator accomplish the desired function in the absence of the automatic feature.

The work described was done at the National Bureau of Standards for the Air Force Logistics Command under Military Interdepartmental Purchase Request FD 2040 78 60013 from the Producibility, Reliability, Attainability and Maintenance Office at Sacramento Air Logistics Center. The Project Manager at Sacramento was Mr. Cleon Binyon, and software modifications were under the direction of Mr. Malcolm Allen at Ogden Air Logistics Center.

Accession For	☒	☐	☐
DTIC GR&I			
DTIC TAB			
Unannounced			
Justification			
By			
Distribution/			
Availability Codes			
Avail and/or			
Dist Special			
Dist			A

BACKGROUND

The following pages of this manual describe the APT postprocessor language chosen for this project. The syntax comes directly from the American National Standards Institute document X3.37 - 1980 on APT. However, the document contains only a subset of the APT language since lathe language is excluded.

The notation for each language statement uses square brackets to denote that one of the included words must appear. Further, it uses parentheses to denote a word that may optionally appear. An example will illustrate:

$$\text{SPINDL} / n, \left(\begin{array}{l} \text{CLW} \\ \text{CCLW} \end{array} \right), \left(\text{RANGE}, \left[\begin{array}{c} m \\ \text{LOW} \\ \text{MEDIUM} \\ \text{HIGH} \end{array} \right] \right)$$

The parentheses show that both the direction specification and the entire range couplet are optional. The square brackets show that if the range couplet is used, one of the four modifiers must appear. The small letters "n" and "m" represent numerical values.

PRECEDING PAGE BLANK-NOT FILMED

AIR / ☐ ON
☐ OFF

This statement controls the operation of air jets used for chip removal or as a coolant. Control is normally effected through the use of miscellaneous function codes on the postprocessor NC output.

Where the NC machine is not equipped with air jets under program control, this function must be implemented manually by the machine tool operator. The postprocessor brings the machine tool to a halt by placing an OPSTOP code on the NC output. A warning message is generated for the part programmer, and the operator is advised through a message on the text output to control the function of the air jets manually.

BREAK

This statement is normally used where the NC output of the postprocessor is punched paper tape. The statement is used to segment the paper tape into convenient and manageable lengths. BREAK causes the postprocessor to terminate the paper tape in an orderly fashion as with an END statement. A new length of paper tape is then started with all functions such as the spindle and coolant restored to the condition in which they were before the break.

CLAMP / ☐ ON
☐ OFF

Initiates or terminates a holding operation. Control is normally effected through the use of miscellaneous function M Codes on the postprocessor NC output.

Where the NC machine is not equipped with clamping devices under program control, this function must be implemented manually by the machine tool operator or the original statement changed by the part programmer. No warning message is generated. If an axis move is programmed on a clamped axis, the unclamping must be done automatically by the postprocessor.

PRECEDING PAGE BLANK-NOT FILMED

Preceding page blank

CLAMP / n, (ON
OFF)

Controls a holding operation on clamp number "n". The default value is "ON" if the option is not specified.

CLAMP / [AAXIS
BAXIS
CAXIS
XAXIS
YAXIS
ZAXIS
WAXIS
TLAXIS] , (ON
OFF)

Controls a clamping operation on one axis of the machine tool. The default value is "ON" if the option is not specified.

CLAMP / [ALL
HEAD,n
CHUCK
COLLET
TABLE
PALLET] , (ON
OFF)

Initiates a clamping operation on the device selected. The default value is "ON" if the option is not specified.

CLRSRF / n

The CLRSRF statement is used by the postprocessor to define a clearance level to be used in conjunction with subsequent RETRACT statements. When a RETRACT is encountered, the postprocessor will automatically retract the tool to the clearance level. The scalar "n" defines the normal distance from the part reference system origin, along the tool axis, to the clearance level.

CLRSRF /

ON
OFF

Used to invoke or suspend a previously defined clearance surface value.

COOLNT /

MIST
FLOOD
AIR
TAP

 . (

ON
OFF

)

This statement is used to control the flow and the type of coolant used. Use of the minor words FLOOD, MIST, and AIR alone constitute a request for that type of coolant to be turned ON. Control is normally effected through the use of miscellaneous function M Codes on the postprocessor NC output.

Where the NC machine is not equipped with coolant under program control, a warning message is generated. The operator is then advised through a message on the text output to control the function of the coolant manually. Only the first occurrence of a COOLNT is noted as a warning message.

COOLNT /

ON
OFF

The ON modifier is used to reinstate the coolant that was last used by the programmer. Where none has been previously specified, the default option FLOOD is used. The OFF modifier will turn off the last coolant specified to be on.

CUTCOM / n

Specifies the cutter compensation register to be used in the NC controller to adjust the programmed tool path for minor variations in cutter diameter and length from the nominal values used in the part program. These variations are usually the result of regrinding, wear, or inaccuracies of tool length adjustment. Cutter compensation is controlled by M and G codes on the NC output. The exact amount of compensation to be applied is entered by the operator into the NC controller. Where the NC machine controller is not equipped with the cutter compensation feature, this statement cannot be correctly processed. The postprocessor must then flag the use of this statement with a warning diagnostic.

CUTCOM /

ON
OFF

Turns cutter compensation ON or OFF.

CUTCOM /

RIGHT
LEFT

 , (n),

XYPLAN
YZPLAN
ZXPLAN

 , (LENGTH, m)

Determines if compensation is applied to right or left side of the cutter path. The optional scalar "n" can be used to specify the cutter compensation register to be used in the NC controller. The default register is governed by the postprocessor. The next option selects the coordinate plane to be used for the cutter compensation. Default value is the XY plane. Finally, cutter length compensation can be optionally specified by the couplet (LENGTH, m). Default length compensation is zero.

CYCLE / BORE, - OPTIONS - (options are specified on page 12)

This statement specifies a boring operation consisting of a rapid plunge, a feed-in motion and a feed-out motion. The depths for these motions are specified by the couplets RAPTO and FEDTO.

CYCLE / BORE, DEEP, (STEP, m, n, p, etc.) - OPTIONS -

This statement specifies a boring operation consisting of a rapid plunge, a series of feed motions, and a rapid retraction. The initial plunge depth is controlled by the RAPTO couplet. The feed motions are in "woodpecker" fashion; feed to depth, retract to initial plunge depth, plunge back to the previous feeddepth, feed down again, etc. The ultimate feed depth is under control of the FEEDTO modifier. Depths for the intermediate positions are under control of the STEP modifier.

CYCLE / BORE, DRAG, - OPTIONS -

This statement specifies a boring operation where the spindle is brought to a halt at the end of the cut and then withdrawn at rapid feedrate whereupon the spindle is restarted. Otherwise the parameters and operation are the same as for CYCLE / BORE.

CYCLE / BORE, NODRAG, (ORIENT, n), - OPTIONS -

This statement is similar to the DRAG option except at the end of the feed motion the spindle is stopped in a known orientation and a move is made in X or Y to back the cutter away from the work before the cutter is retracted. Where the spindle orientation capability is not available, this statement defaults to a CYCLE / BORE, MANOP.

CYCLE / BORE, MANOP, - OPTIONS -

This statement allows for a manual retraction of the tool once the feed motion has terminated. The machine tool spindle is turned off automatically.

CYCLE / BORE, RAPOUT, - OPTIONS -

This statement specifies a boring operation followed by a rapid withdrawal with spindle still in motion. Otherwise the parameters and operation are the same as for CYCLE / BORE.

CYCLE / DRILL, - OPTIONS -

This statement specifies a drilling operation consisting of a rapid plunge, a feed motion, and a rapid retraction. Depths for these motions are contained in the modifier couplets RAPTO and FEDTO.

CYCLE / DRILL, DEEP, (STEP, m1, m2 - - mn), - OPTIONS -

This statement specifies a drilling operation consisting of a rapid plunge, a series of feed motions, and a rapid retraction. The initial plunge depth is controlled by the RAPTO couplet. The feed motions are in "woodpecker" fashion; feed to depth, retract to initial plunge depth, plunge back to the previous feed depth, feed down again, etc. The final feed depth is controlled by the FEEDTO couplet. Depths for the intermediate positions are under the control of the STEP couplet.

CYCLE / CSINK, DIAMTR, m, - OPTIONS -

This statement specifies a countersinking operation consisting of a rapid plunge, a feed motion, and a rapid retraction. The feed depth is computed by the postprocessor from the scalar values of "m", the desired countersink diameter, and the tool tip angle given in the previous LOAD/TOOL statement.

CYCLE / TAP, - OPTIONS -

This statement specifies a fixed cycle of tapping operations consisting of a rapid plunge, forward spindle, feed to depth, reverse spindle, feed-out to the feed start position, and a rapid retraction.

CYCLE / MANUAL

A manually controlled operation is to be performed at each X Y point. The machine operator will restart the machine after each operation.

CYCLE / MILL, - OPTIONS -

This statement specifies a milling operation consisting of a rapid plunge, a feed motion into the workpiece, a milling sequence to successive GOTO points, and a rapid retraction after the last point. Milling feedrate is specified by a normal FEDRAT/ command. The CYCLE feedrate couplet controls only the tool axis feed motion. The milling sequence is cancelled by any other CYCLE command.

CYCLE / OFF

This statement terminates a previously defined cycle operation sequence.

CYCLE / ON, - OPTIONS -

This statement reinstates the last defined cycle operation sequence. Options may be appended to modify any aspect of the sequence desired.

For all CYCLE operations the following couplets can be used as options:

RAPTO, m	The tool will plunge at rapid feedrate to the position defined by the delta distance "m". A positive value is measured from the point back along the spindle center line. A negative value is measured from the point in the opposite direction.
FEDTO, n	The tool will plunge under feedrate control to the position defined by the delta distance "n". Positive and negative values are measured as for RAPTO.
IPM, f	The tool will move at a feedrate "f" in units of inches per minute.
IPR, r	The tool will move at a feedrate "r" in units of inches per revolution.
DWELL, s	Commands a dwell of "s" seconds.

DELAY / n

Causes a suspension of all machine operations for "n" seconds. Used to halt the machine for a given length of time.

DELAY / REV, n

Causes a suspension of all machine operations for "n" revolutions of the spindle.

Dwell times are usually initiated by an appropriate miscellaneous function, M code, or by a preparatory function, G code. The length of dwell is governed either by an associated motion register (X,Y,Z, etc.) or by activating a preset timer in the NC controller. In the case of a dwell for a specified number of revolutions, the postprocessor first calculates the time in seconds from the known spindle speed and the given number of revolutions.

For controllers without a delay feature, the postprocessor should either simulate a delay or generate a warning message indicating a delay was requested but not executed. A delay can sometimes be effected by placing leader characters on the NC output.

END

This statement indicates the logical completion of a section of programming. An orderly termination of all postprocessor output follows. Final messages are printed on the text output concerning tape length produced, machining time, number of processing errors, etc.

FEDRAT / n , $\left(\text{RANGE}, \begin{bmatrix} m \\ \text{LOW} \\ \text{MEDIUM} \\ \text{HIGH} \end{bmatrix} \right)$

Specifies the feedrate in inches per minute. Range selection is optional.

FEDRAT / $\left[\begin{array}{c} \text{IPM} \\ \text{IPR} \end{array} \right] , n , \left(\text{RANGE}, \left[\begin{array}{c} \text{m} \\ \text{LOW} \\ \text{MEDIUM} \\ \text{HIGH} \end{array} \right] \right)$

Specifies the feedrate "n" in units of inches per minute or per revolution. Range selection is optional. The postprocessor will ordinarily calculate the proper F code (feedrate number) to be placed on the NC output based upon data such as the commanded feedrate, the length of tool motion being executed, and the spindle speed if IPR has been specified.

The feedrate may have to be reduced by the postprocessor as a result of acceleration - deceleration requirements of the machine, surface feet per minute calculations, or tape reader speed limitations. All feedrates are tested by the postprocessor to lie within maximum and minimum bounds. The optional range selection will normally result in a miscellaneous M code being generated to shift gears of the spindle transmission along with a delay preparatory function G code to cause a dwell for the required length of time for gear changing.

For those NC machines without gear changing ability, the RANGE option is ignored. If by chance the commanded feedrate exceeds the capability of the machine tool, a warning message is placed on the output text listing and the maximum feedrate is assigned. If a feedrate command is encountered which cannot be executed within given range, an automatic range change is made and a warning message is generated.

GOHOME

Commands all axes of the machine tool to move to the HOME position. The statement is often used for manually changing tools, for part inspection, or for loading and unloading workpieces. Rapid feedrate is used for this move.

The HOME position is normally specified as the X,Y,Z value in the first FROM statement. The inclusion of a SET / HOMEPT, X, Y, Z command in the part program will override all previous specifications. If the HOME point is not defined in the part program, the postprocessor will assign a default value for that particular machine. However, this may not be possible with controllers offering a full floating zero capability.

INSERT - - - - - \$

Characters occurring after INSERT are inserted directly into the output control information. No check is made for validity of data. Imbedded blank characters are inserted in the same manner. Character string must be terminated with a \$.

LEADER / n

Specifies the amount of leader to be produced in units set by the postprocessor, commonly inches.

LINTOL / n

Controls the linearization function. The value "n" specifies the acceptable deviation of the tool tip from the desired cutter path.

LINTOL /

ON
OFF

Commands that the linearization function in the postprocessor be on or off. The postprocessor linearization function forces the machine tool to stay within a specified tolerance of a straight line connecting two successive cutter location points. Intermediate points are calculated to fulfill this purpose. Linearization is performed only for non-rapid movements when started by a LINTOL statement.

LOAD / TOOL, n, (TLSET, m), (TLANGL, a), (MANUAL)

Controls the loading of a tool specified by the optional parameters or by a previous SELECT / TOOL statement. "n" is the tool identification number, "m" is the set length, and "a" is the tool tip angle. These are specified in optional couplets and can occur in any order. The inclusion of the MANUAL option will disable the automatic tool change sequence to allow for special situations.

The tool to be loaded by this command is specified by the value "n". Use of just the basic statement LOAD / TOOL with no other data implies that the tool was previously specified by a SELECT / TOOL statement. Control of the tool loading operation is normally effected through the use of miscellaneous function, M Codes, preparatory function, G Codes, and tool function, T Codes, on the postprocessor NC output.

Where the NC machine is not equipped with tool changing capability under program control, this function must be implemented manually by the machine tool operator. In this situation the postprocessor will move the machine tool to the manual tool change position and will halt all machine operations by placing STOP and SPINDL / OFF codes on the NC output. The operator is then advised to load the specified tool manually through a message on the text output.

MCHTOL / n

This feature of a postprocessor checks the amount of overshoot or undershoot that will occur when a change in direction has been programmed. If the deviation from the desired cutter path exceeds the value "n", the postprocessor will compensate by inserting commands on the NC output. Dwell commands are used to correct for excessive undershoot. The commanded feedrate is reduced to correct for overshoot.

OPSKIP /

ON
OFF

This statement allows the machine tool operator to skip over optional sections of NC control information. All blocks of postprocessor NC output occurring after an OPSKIP/ON statement are flagged with slashes to allow the NC controller to ignore them whenever the machine operator uses the block delete function of the controller. Where this command is used for controllers not equipped with the block delete option, a warning message is generated.

OPSTOP

Optionally halts all machine motions under tape control and turns off the coolant, spindle, and tape reader. This command is similar to the STOP command but is under the option of the machine operator. Through a switch on the control panel he can enable or disable this feature. If the switch is off, the command is ignored.

For a machine not equipped to recognize the OPSTOP miscellaneous function M Code, the postprocessor should ignore the command and generate a warning message.

ORIGIN / X, Y, (Z), (A), (B), (C)

This statement defines the machine origin in terms of the part programming reference system. X, Y, and Z give the location of the machine tool origin in the coordinate system used for the part program. Likewise, A, B and C give the orientation of the machine's rotary axes. The values for Z, A, B and C are optional. Their default values are zero.

PIVOTZ / n

This statement is used only by multiaxis postprocessors to define the relationship of the part programming origin to the pivot plane of the machine tool. The pivot plane is the basic reference plane of the machine tool. It is generally a Z plane. All machine slide dimensions are measured from the pivot plane. The scalar value "n" specifies the distance from the part origin to the pivot plane.

PPRINT - - - - -

This statement is used to place comments in the postprocessor text output. These are used to give messages to the part programmer or machine tool operator concerning part setup, tool changes, or reasons for a programmed STOP or OPSTOP. The string of characters are inserted into the printed output of the postprocessor.

The tool to be loaded by this command is specified by the value "n". Use of just the basic statement LOAD / TOOL with no other data implies that the tool was previously specified by a SELECT / TOOL statement. Control of the tool loading operation is normally effected through the use of miscellaneous function, M Codes, preparatory function, G Codes, and tool function, T Codes, on the postprocessor NC output.

Where the NC machine is not equipped with tool changing capability under program control, this function must be implemented manually by the machine tool operator. In this situation the postprocessor will move the machine tool to the manual tool change position and will halt all machine operations by placing STOP and SPINDL / OFF codes on the NC output. The operator is then advised to load the specified tool manually through a message on the text output.

MCHTOL / n

This feature of a postprocessor checks the amount of overshoot or undershoot that will occur when a change in direction has been programmed. If the deviation from the desired cutter path exceeds the value "n", the postprocessor will compensate by inserting commands on the NC output. Dwell commands are used to correct for excessive undershoot. The commanded feedrate is reduced to correct for overshoot.

OPSKIP /

ON
OFF

This statement allows the machine tool operator to skip over optional sections of NC control information. All blocks of postprocessor NC output occurring after an OPSKIP/ON statement are flagged with slashes to allow the NC controller to ignore them whenever the machine operator uses the block delete function of the controller. Where this command is used for controllers not equipped with the block delete option, a warning message is generated.

OPSTOP

Optionally halts all machine motions under tape control and turns off the coolant, spindle, and tape reader. This command is similar to the STOP command but is under the option of the machine operator. Through a switch on the control panel he can enable or disable this feature. If the switch is off, the command is ignored.

For a machine not equipped to recognize the OPSTOP miscellaneous function M Code, the postprocessor should ignore the command and generate a warning message.

ORIGIN / X, Y, (Z), (A), (B), (C)

This statement defines the machine origin in terms of the part programming reference system. X, Y, and Z give the location of the machine tool origin in the coordinate system used for the part program. Likewise, A, B and C give the orientation of the machine's rotary axes. The values for Z, A, B and C are optional. Their default values are zero.

PIVOTZ / n

This statement is used only by multiaxis postprocessors to define the relationship of the part programming origin to the pivot plane of the machine tool. The pivot plane is the basic reference plane of the machine tool. It is generally a Z plane. All machine slide dimensions are measured from the pivot plane. The scalar value "n" specifies the distance from the part origin to the pivot plane.

PPRINT - - - - -

This statement is used to place comments in the postprocessor text output. These are used to give messages to the part programmer or machine tool operator concerning part setup, tool changes, or reasons for a programmed STOP or OPSTOP. The string of characters are inserted into the printed output of the postprocessor.

RAPID

This statement specifies that the next programmed movement is to be executed at the maximum possible feedrate. The RAPID command applies only to the next movement. All subsequent moves are at the previously defined feedrate.

The exact feedrate for Rapid motions is machine dependent but can be set by the part programmer using a SET / RAPFED, f command.

RETRACT

This statement specifies a rapid traverse movement of the tool to the clearance surface which has been previously defined by a CLRSRF statement. Motion is along the tool axis (Z axis). The command is used on positioning and milling machines.

REWIND

This command repositions the control medium to the first rewind stop code detected. The machine is brought to an orderly halt with spindle, coolant, etc. turned off.

ROTATE /

HEAD
TABLE

 , n

Commands the rotation of a head or table in the positioning mode only. Motion is not under feedrate control. The scalar "n" specifies the absolute value of the angle of rotation and is given in degrees.

The ROTATE command normally results in A or B axis movements on the postprocessor NC output. For machine tools not equipped for these movements there can be no simple manual remedy. The postprocessor should flag this condition as being beyond the capability of the selected machine tool.

ROTATE /

HEAD
TABLE

 , INCR, m , (CLW) , (ROTREF)
(CCLW)

The angular rotation from the present position is specified by the couplet (INCR,m). It is an incremental movement of "m" degrees. Where the optional direction selector is not specified, the rotation is governed by the sign of "m". Rotation is clockwise for positive "m" and counterclockwise for negative "m". Use of ROTREF insures that after rotation the part coordinate system and the machine coordinate system remain in alignment.

ROTATE /

HEAD
TABLE

 , ATANGL, n , (CLW) , (ROTREF)
(CCLW)

The couplet (ATANGL, n) commands a rotation to the angle "n" degrees. Direction of rotation is governed by the optional selector. Default direction is clockwise. As above, the option ROTREF will cause rotation of the part programming coordinate system.

SELECT / HEAD,

n
ALL
BOTH
LEFT
RIGHT
MAIN
HIGH
LOW

This statement identifies a particular head on the NC machine which will be referenced by future statements. The scalar "n" will select the head by number. Machines with a single head ignore this statement.

SELECT / TOOL, n, (TLSET, m), (TLANGL, a), (MANUAL)

This statement controls the selection of a cutting tool to be used by the next LOAD / TOOL statement. previous SELECT / TOOL statement. The scalar "n" is the tool identification number, "m" is the set length, and "a" is the tool tip angle. These are specified in optional couplets and can occur in any order. The inclusion of the MANUAL option will disable the automatic tool change sequence to allow for special situations. The tool will not be loaded into the active spindle until a LOAD/TOOL statement is encountered at a later time. Control is normally effected through the use of miscellaneous function, M codes, preparatory

function, G codes, and tool function, T codes, on the postprocessor NC output.

Where the NC machine is not equipped with automatic tool changing capability under program control, this function must be implemented manually by the machine tool operator when the LOAD/TOOL statement is encountered. The postprocessor must simply store the data given in the SELECT/TOOL statement for future use when issuing a message to the operator for loading the tool.

SEQNO / n

Specifies that the next output block will be assigned the sequence number "n". Only the next block will be affected. Normal sequencing will then resume.

SEQNO /

ON
OFF

Starts and stops the sequence numbering function. The ON modifier is used to reinstate the sequence numbering that was last used by the programmer. Where none has been previously specified, the default option is to number each block starting with the number one and incrementing by one.

SEQNO / INCR, m, (r)

Specifies that the next output block will be assigned a sequence number equal to the previous number (or zero if no previous number was assigned) plus the scalar value "m". Subsequent sequence numbers will be incremented by the value "m". If the scalar value "r" is present, a sequence number will only be assigned once every "r" output blocks.

SEQNO / n, INCR, m, (r)

Specifies that the next output block will be assigned sequence number "n" and that each following block will be incremented by "m" and "r" as above.

SET / MODE,

LINEAR
CIRCUL
PARAB

Specifies the particular mode of interpolation to be used, (when-ever possible) for all non-linear motions.

SET / RAPFED, f

Allows the specification of a feedrate for all RAPID moves including within the CYCLE commands. If "f" exceeds the normal RAPID for the machine, a default maximum value is selected.

SET / TLCHG, X, Y, (Z)

Selects the position to be used for all manual tool changes. The values X, Y, and Z are in the part coordinate system. Z is optional with the default value in the fully retracted position except with controllers using a floating zero. On these controllers, if the Z position is not specified, the Z axis machine position remains unchanged.

SET / HOMEPT, X, Y, Z

Selects the position for all GOHOME commands. The values X, Y, and Z are in the part coordinate system.

$$\text{SPINDL} / n, \begin{pmatrix} \text{CLW} \\ \text{CCLW} \end{pmatrix}, \left(\text{RANGE}, \begin{bmatrix} m \\ \text{LOW} \\ \text{MEDIUM} \\ \text{HIGH} \end{bmatrix} \right)$$

Commands the spindle to be turned on at the specified angular velocity "n" revolutions per minute. The default of the optional angular direction is clockwise. The default of the optional range is determined by the postprocessor, and "m" specifies a range by number.

Control of spindle speed is accomplished by varying the speed of the drive motor and by changing the gear ratio between the motor and the spindle. The desired gear range can be specified by the words HIGH, MEDIUM, and LOW or can be selected by number. m = 1 specifies the lowest range; m = 2 is the next higher range; m = 3 next, etc. For transmissions with three ranges, LOW, MEDIUM and HIGH correspond to m = 1, 2, 3. But for transmissions with only two ranges, MEDIUM and HIGH both select the m = 2 range. Control is accomplished through the use of appropriate miscellaneous function, M Codes, and preparatory function, G Codes.

If a range selection is necessary but not specified, it shall be accomplished by the postprocessor in the best way possible. Where the NC machine is not equipped with spindle functions under program control, this function must be implemented manually by the machine tool operator. The postprocessor brings the machine tool to a halt by placing a STOP code on the NC output. The operator is then advised through a message on the text output to manually set the speed direction, and range of the spindle drive system to the proper values.

$$\text{SPINDL} / \text{ON}, \begin{pmatrix} \text{CLW} \\ \text{CCLW} \end{pmatrix}, \left(\text{RANGE}, \begin{bmatrix} m \\ \text{LOW} \\ \text{MEDIUM} \\ \text{HIGH} \end{bmatrix} \right)$$

Commands the spindle to be turned on. Previous spindle speed, direction, and range are reinstated unless modified by the optional parameters. If SPINDL / ON appears as the first specification, a default spindle speed is assigned by the postprocessor.

SPINDL / OFF

Turns off the spindle motor and coolant.

SPINDL / NEUTRL

Places the spindle motor transmission in the neutral position.

SPINDL / LOCK

Fixes the spindle shaft so that it cannot be turned.

SPINDL / KEYLCK

Causes the spindle to be fixed in the key lock position.

SPINDL / ORIENT, (n)

Causes the spindle to be oriented into an absolute angular position specified by the optional scalar "n" or in the absence of "n" in a position determined by the machine hardware or the post-processor.

STOP

Halts all machine motions under tape control and turns off the coolant, spindle, tape reader, etc.

TMARK

This command causes a rewind stop code to be placed on the post-processor NC output. It is used by the NC controller to terminate the rewind sequence of the control tape and restart machining from a known point in the program.

TRANS / X, Y, (Z)

Defines the translation parameters used to modify tool position coordinates. The X, Y and Z values are those to be added to the part coordinates in order to obtain the machine tool coordinates. Z is optional with the default value being zero.