CHAPTER 9

AUXILIARY MACHINE CONTROL FUNCTIONS

9.1 SPINDLE SPEED (S FUNCTION)

The address S followed by a numeric value designates the spindle speed. The spindle is not under CNC control, the programmed spindle speed is primarily used to display to the operator what spindle speed is required.

9.2 TOOL FUNCTION (T FUNCTION)

The tool function consists of the address T followed by a two digit number ranging from 1 to 24. The tool function word has the following functions:

1. Store tool length offsets and diameter compensation. The format for this command is:

   T___/___/___

   Where the numeric value following the T address (T___) designates where in the tool offset table the subsequent data is to be stored, the numeric value after the first / code designates the tool length offset and the numeric value after the second / code designates the diameter offset.

   EXAMPLE:

   ```
   T1/1./.5; TLO=1.,DIA=.5
   T7//.375; TLO IS UNCHANGED, DIA=.375
   ```

2. Select the active tool number.

3. When a tool change command (M6,M26) is read by the control, the tool length offset for the active tool number will be set into the Z move offset register. Refer to Section 4.4 for details on TLOs.

4. When a cutter compensation on command (G41,G42) is read by the control, the cutter diameter for the active tool number is set into the cutter compensation register and used for subsequent calculations.

   EXAMPLE:

   ```
   N100T2M26; TOOL CHANGE @ CLR PT, TOOL=2
   N204T7G41X0Y0; USE TOOL=7 FOR DIA COMPENSATION
   ```

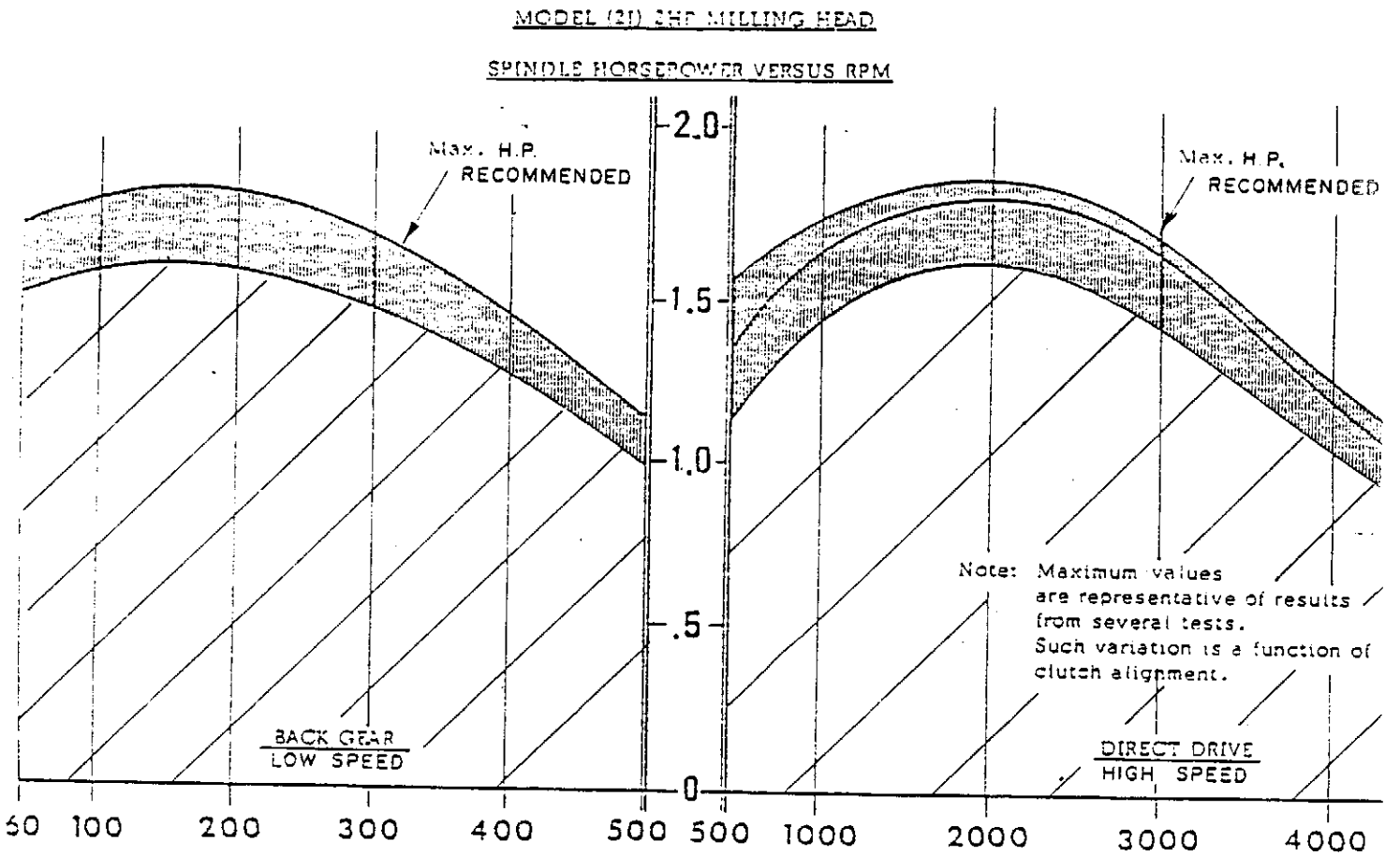

Figure 9-1: Series I CNC Spindle Drive HP vs. Speed Characteristics

9.3 MISCELLANEOUS FUNCTION COMMANDS

Programmed miscellaneous function codes (M codes) initiate various machine tool functions and establish various control conditions. The following lists the miscellaneous function codes available:

M0 PROGRAM STOP (NON-MODAL)
M1 OPTIONAL PROGRAM STOP (NON-MODAL)
M2 END OF PROGRAM (NON-MODAL)
M6 TOOL CHANGE (NON-MODAL)
M7 COOLANT/MIST (MODAL)
M8 COOLANT/FLOOD (MODAL)
M9 COOLANT OFF (MODAL)
M20 PROGRAM STOP, GOTO CLEAR PT (NON-MODAL)
M21 OPTIONAL PROGRAM STOP, GOTO CLEAR PT (NON-MODAL)
M22 END OF PROGRAM, GOTO CLEAR PT (NON-MODAL)
M25 QUILL HOME (NON-MODAL)
M26 TOOL CHANGE, GOTO CLEAR PT (NON-MODAL)
M51 ADVANCE INDEX TABLE (NON-MODAL)

NOTE

Only one M code can be used in a part program block. If more than one M code is programmed, the control will execute the last M code input.

9.3.1 PROGRAM STOP (M0)

Spindle and coolant flow will stop and program execution will be halted. To continue operation use the START/CONTINUE key.

NOTES

1. For M0 and M1 the quill will remain at its last programmed position..
2. For M0, M1, M2, M6, M20, M21, M22, M25, and M26 the system must be in the Rapid Traverse mode.

9.3.2 OPTIONAL STOP (M1)

This is the same as an M0 code except it is only executed if the OPSTOP function has been selected. Otherwise, M1 will be ignored.

9.3.3 PROGRAM REWIND (M2)

The spindle, coolant flow and program execution will be halted. Before commanded XY motion occurs (if programmed) the quill will retract to the up position. The part program text pointer will be reset to the top of the program.

NOTES

1. The part program text area may contain a number of defined part programs. Program rewind resets the text pointer to the top of the text area. To rerun a particular part program, it will be necessary for the operator to use the FIND command to move the text pointer to the desired part program.

2. The program rewind command will reset the active preparatory function (G code) commands to initialized conditions. The following G codes will be set:

   G0, G17, G30, G40, G45, G70, G72, G75, G90, G96

Hitting START/CONTINUE will repeat execution of the part program from the beginning.

9.3.4 TOOL CHANGE (M6)

This function denotes that a tool change has been requested. The following sequence occurs:

1. A control signal will be given to stop the spindle and coolant flow.

2. X and Y motion will occur, if programmed, after the Z axis retracts to the "Home" position (quill up).

3. The Z active offset register will be set to the value of the tool length offset designated by the active tool number.

4. The TOOL light on the Front Panel will indicate a tool change is to occur. In addition, a message embedded in part program text may be displayed on the LCD screen.

5. The operator restarts the spindle, sets the correct speed for the new tool and presses the START/CONTINUE key.

NOTE

The programmer should position the XY axes for part, fixture and for tool clearance prior to or in the same part program block as an M6 command.

9.3.5 COOLANT (M7,M8,M9)

An M7 code turns on mist coolant. A M8 code turns on flood coolant. M9 turns off both coolants. M7, M8, and M9 commands are modal.

NOTE

Coolant flow is interlocked with the spindle ON electrics. When the spindle goes OFF, coolant flow will be interrupted and when the spindle is turned ON, coolant flow will resume. Thus, an M9 code is not necessary within the program text and M7 and M8 need to be entered only once at the beginning of the program.

9.3.6 MOVE TO CLEAR POINT (M20,M21,M22,M26)

These commands are similar to M0, M1, M2, M6 respectively except the axes will automatically move to the clear point. The clear point is set by the operator in SETUP such that the part or fixture is cleared and the tool can be easily removed. In moving to the clear point first the quill will retract, then XY motion will occur.

NOTE

The system must be in the Rapid Traverse mode (G0) when move to clear point occurs.

EXAMPLE:

```
N230G0M26;      TOOL CHANGE @ CLEAR POINT
```

9.3.7 QUILL HOME (M25)

This will cause the quill to retract to the up posibion before XY motion occurs.

9.3.8 INDEX TABLE (M51)

This will cause the optional index table to advance one position. The system will wait until feedback from the index table switches indicate the indexing cycle is complete.

9.4 OPERATOR MESSAGES

Alphanumeric codes embedded in the part program text between ' ___ ' (quotes) will be displayed on the LCD when an M0, M1, M6, M20, M21 or M26 command occurs. This can be used to provide messages containing operator instructions.