proLIGHT Machining Center
The Elements of an NC Part Program

Part programs generally incorporate two types of instructions: those which define the tool path (such as X, Y and Z axis coordinates), and those which specify machine operations (such as turning the spindle on or off). Each instruction is coded in a form the computer can understand.

An NC program is composed of *blocks* (lines) of code. The maximum number of blocks per program is limited by the memory (RAM) on your computer. You can, if necessary, chain programs together to form very large part programs.

Each block contains a string of *words*. An NC word is a code made up of an alphabetic character (called an *address character*) and a number (called a *parameter*). There are many categories of address characters used in NC part programs for the proLIGHT Machining Center (see Categories of NC Code).

Each block of NC code specifies the movement of the cutting tool on the Machining Center and a variety of conditions that support it. For example, a block of NC code might read:

```
NOG90GO1 X.5Y1.5Z0F1
```

If the machine is currently set for inch units, the individual words in this block translate as:

**NO** This is the block sequence number for the program. Block 0 is the first block in the program.

**G90** This indicates absolute coordinates are used to define tool position.

**GO1** This specifies linear interpolation.

**X.5** This specifies the X axis destination position as 0.5". **Y1.5** This specifies the Y-axis destination position as 1.5".

**Z0** This specifies the Z-axis destination position as 0". The cutting tool will move to the absolute coordinate position (0.5,1.5,0).

**F1** This specifies a feed rate of 1 inch per minute, the speed at which the tool will advance to the specified coordinate points.

**Categories of NC Code**

There are many categories of NC code used for programming the proLIGHT Machining Center. Each category is identified by an address character that has a special meaning. Here is a list of the address characters supported by the proLIGHT Machining Center (in order of their appearance in a block).

/ Optional skip
\ Skip
N Block number (for user reference only)
0 Subprogram starting block number
G Preparatory codes
X X axis motion coordinate
U Incremental linear axis parallel to the X axis (for absolute dimensioning)
Y Y axis motion coordinate
V Incremental Y motion dimension for absolute dimensioning
Z Primary Z motion dimension
W Incremental Z motion dimension for absolute dimensioning
A A axis coordinate (see Rotary Positioner information in Section 6)
I Arc center, X axis dimension (circular interpolation)
J Arc center, Y axis dimension (circular interpolation)
K Arc center, Z axis dimension (circular interpolation)
R Arc radius for circular interpolation, and drilling start location
Q Peck depth for pecking cycle
H Input selection number/T 001 length offset
D Compensation offset value
L Loop counter for subprograms and program cycle counter
F Feed rate in inches per minute, or dwell time in seconds
S Spindle speed
T Tool specification
M Miscellaneous functions
P Subprogram reference number/Uniform scale multiplier
; Comment (open parenthesis can also be used) "
$ Run program with absolute arc centers

Optional Skip (/)

The optional skip code allows you to optionally skip blocks of code as the NC program executes. Make sure to activate the Optional Skip parameter in the Run Settings dialog box, then place a forward slash (/) in front of the block you want to skip. With Optional Skip off, the slash is ignored and the block of code is executed. With optional skip on, the slash is recognized and the block of code is skipped. If you place a number after the optional skip code, the program executes that block every nth pass. For example: /5G28; Homes every fifth pass

Skip (/)

The skip code works like the optional skip, except that it works even if the Optional Skip parameter from the Run Settings dialog box is off.

Block Number (N Code)
The N code specifies the sequential number of the block in the NC program. Using the N code is optional; however, when you do use the N code, it must be the first character of each block in the program. Select Renumber from the Edit Menu to renumber and remove and N codes.

**Subprogram Block Number (0 Code)**

The 0 code replaces the N code in a block when a subprogram begins, and marks the start of a subprogram. *Only the first block in the subprogram should contain an 0 code.*

**Preparatory Codes (G Codes)**

G codes take effect before a motion is specified. They contain information such as the type of cut to be made, whether absolute or incremental dimensioning is being supplied, whether to pause for operator intervention, and so on.

The G codes supported by the proLIGHT Control Program fall into a number of groups: the Interpolation Group, the Units Group, the Plane Selection Group, the Wait Group, the Canned Cycles Group, the Programming Mode Group, the Preset Position Group, the Compensation Functions Group, the Coordinate System Group, the Feed Functions Group, and the Polar Programming Group.

**The Interpolation Group**

These G codes are retained until superseded in the program by another code from the same group. Four interpolation G codes are supported:

- GOO Rapid traverse
- GO1 Linear interpolation (default)
- GO2 Circular interpolation (clockwise)
- GO3 Circular interpolation (counterclockwise)

**Units Group**

There are two G codes to denote units of measure, G70 (inch) and G71 (metric). These codes tell the Machining Center which unit of measure you are using. They are placed at the beginning of the program before any moves are made. Only one of the two codes can be used per block. You can also use the Fanuc equivalents, G20 (inch) and G21 (metric).

**Plane Selection Group**

This group of codes allows you to select different planes for circular interpolation, rotation, and cutter compensation. G17 is the proLIGHT Control Program default. The supported Plane Selection Group codes are:
G 17 Select the X, Y plane for circular interpolation. Use this code if you are switching back to the X, Y plane after circular moves on the X, Z or Y, Z planes. The arc center coordinates are given by I for the X axis and J for the Y axis.

G 18 Select the X, Z plane for circular interpolation. Use this code to perform circular interpolation on the X, Z plane. The arc center coordinates are given by I for the X axis and K for the Z axis.

G19 Select the Y, Z plane for circular interpolation. Use this code to perform circular interpolation on the Y, Z plane. The arc center coordinates are given by J for the Y axis and K for the Z axis.

**Wait Group**

Wait Group codes apply only to the block in which they appear. Multiple Wait Group G codes should be placed on several different lines. The supported Wait Group codes are:

**GO4 Dwell (wait):** Equals the value of the time (F code) in seconds (used primarily for robotic operations). GO4 excludes motion commands with a new feed rate on the same line (block).

**GO5 Pause:** Use for operator Intervention.

**G25 Wait until robot input goes high.** Used in conjunction with H code, which specifies the input number. Used for robot synchronization.

**G26 Wait until robot input goes low.** Used in conjunction with H code, which specifies the input number. Used for robot synchronization.

**G31 Linear move to specified coordinate.** Stop short if specified input goes High (if H is positive) or Low (if H is negative). Move until an input is triggered or until a coordinate is reached.

**G131 Stop Z axis motion on input (INROBI ON) during digitizing.** You should only use this code with Light Machines' digitizing package.

**Canned Cycle Group**

Canned cycle codes allow you to perform a number of tool motions by specifying just one code. For detailed information on using these codes, refer *Canned Cycle Programming* in this section. Canned cycles are retained until superseded in the program by another canned cycle code. These are the canned cycle codes supported by the proLIGHT Machining Center:

G80 Canned cycle cancel.
G81 Canned cycle drilling.
G82 Canned cycle straight drilling with dwell.
G83 Canned cycle peck drilling.
G84 Canned cycle tapping.
G85 Canned cycle boring.
G86 Canned cycle boring with spindle off (dwell optional).
G89 Canned cycle boring with dwell.

**Programming Mode Group**

Programming mode G codes tell the Machining Center which programming mode to use; G90 for absolute, or G91 for incremental. These codes remain in effect until superseded by each other. The default code on program start up is G90.

With absolute programming, all X, Y and Z coordinates are relative to the 0,0 point on the Machining Center. With incremental programming, each motion to a new coordinate is relative to the previous coordinate.

**Preset Position Group**

The preset position G codes move the tool to a predetermined position. They apply only to the current programming block. These are the preset position codes supported by the proLIGHT Machining Center:

- **G27** Check reference point: Compares reported position against zero to see if any position has been lost.

- **G28** Set reference point: This code moves the machine to its home position and sets the machine position to 0,0,0. The G28 code performs an automatic calibration of the axes.

- **G92** Preset position: This code works like the Set Position function under the Setup Menu (see Section 3). The X, Y and Z coordinates following a G92 code define the new current position of the tool.

- **G98** Rapid move to initial tool position after canned cycle complete (see *Canned Cycle Programming* in this section.)

- **G99** Rapid move to point R (surface of material or other reference point) after canned cycle complete.

**Compensation Functions Group**

- **G39** Corner offset circular interpolation.

- **G40** Cancels cutter compensation.

- **G41** Left cutter compensation: Enables cutter compensation to the left of programmed tool path.

- **G42** Right cutter compensation: Enables cutter compensation to the right of programmed tool path.

- **G43** Tool length offset: Shifts Z axis in a positive direction by a value specified by an H code.
G44 Tool length offset: Shifts Z axis in a negative direction by a value specified by an H code.

G45 Increases the movement amount by the value stored in the offset value memory.

G46 Decreases the movement amount by the value stored in the offset value memory.

G47 Increases the movement amount by twice the value stored in the offset value memory.

G48 Decreases the movement amount by twice the value stored in the offset value memory.

G49 Cancels tool length offsets.

G50 Cancels scaling.

G51 Invokes scaling.

G68 Invokes rotation.

G69 Cancels rotation.

Coordinate System Group

Coordinate system codes allow you to create multiple parts by establishing multiple coordinate systems on one work piece. For instance, you can run a part program using a typical coordinate system (with the point of origin on the surface of the front left corner of the work piece), then select another coordinate system which has its origin at a different point on the surface of the work piece.

There are seven coordinate system codes. One of these codes (G53) is used to rapid to specified coordinates, while the other six allow you to make up to six individual parts on the same work piece by specifying different coordinate systems for each part. The coordinate system codes are G54 (coordinate system 1), G55 (coordinate system 2), G56 (coordinate system 3), G57 (coordinate system 4), G58 (coordinate system 5) and G59 (coordinate system 6). For a more detailed explanation on using coordinate system codes, refer to Section 3.

Polar Programming Group

These two codes, G15 and G16, allow you to perform operations based on polar coordinates. The polar coordinates are defined by X (radius) and Y (angle in degrees) when programming for the X,Y plane. For more information on using these G codes, refer to Polar Programming in this section.

X Axis Coordinate (X or U Code)

An X code specifies the coordinate of the destination along the X axis. The default measurement value is set using the Set Preferences command under the Setup Menu (refer to Section 3). You can use a U code while in absolute dimensioning to specify an incremental X motion.
Y Axis Coordinate (Y or V Code)

An Y code specifies the coordinate of the destination along the Y axis. The default measurement value is set using the Set Preferences command under the Setup menu (refer to section 3). You can use a V code while in absolute dimensioning to specify an incremental Y motion.

Z Axis Coordinate (Z or W Code)

A Z code specifies the coordinate of the destination along the Z axis (spindle axis). The default measurement value is set using the Set Preferences command under the Setup Menu. You can use a W code while in absolute dimensioning to specify an incremental Z motion.

Miscellaneous Codes (M Codes)

M codes control Machining Center functions while the part program is running. They control the spindle On/Off switch, and clamp or unclamp the air vise. M codes can also be used to chain a second program to the end of a part program, or to repeat the program. Only one M code can be specified per NC block. These are the M codes supported by the proLIGHT Machining Center:

- **M00 Pause**: Allows you to place a pause in your code. Acts like a G05 pause.
- **M01 Optional Stop**: Allows you to place 'an optional stop in your code. Place an M01 in the block of code where you would like to pause. With Optional Stop on, the M01 works like an M01 stop. With Optional Stop off, the M01 code is ignored, the other codes on the block are executed as usual.
- **M02 End of Program**: Takes effect after all motion has stopped; turns off drive motors, spindle and accessory outlets.
- **M03 Spindle Motor On**: Activated concurrently with motion specified in the program block; remains in effect until superseded by M05. M05 Spindle Motor Off Activated after the motion specified in the program block; remains in effect until superseded by M03.
- **M06 Tool Change**: Used in conjunction with a T code to perform multiple tool operations. See Section 6.
- **M08 ACCl On**: Turns on accessory ACCI outlet concurrently with the motion specified in the program block; remains in effect until superseded by M09.
- **M09 ACCl Off**: Turns off accessory ACCI outlet after the motion specified in the program block; remains in effect until superseded by M08.
- **M10 Clamp ACC2**: Closes air vise accessory concurrently with the motion specified in the program block; remains in effect until superseded by M11.
- **M11 Unclamp ACC2**: Opens air vise accessory after the motion specified in the program block; remains in effect until superseded by M10.
**M20 Chain to Next Program:** This code appears at the end of a part program and is followed on the next line by the file name of another program which is executed when all motion stops. Here's an example of a part program chain to another program:

```
N37Z.2
N38M20
PROGRAM TWO
```
If the two programs you are chaining are not in the same directory, you must specify the full path name for each file.

**M22 Write to file:** Outputs machine coordinates to a file. The proper format for using this code is: `M22(filename)`. The first time the Control Program encounters an M22 code, it opens your specified file. You're basically naming the file as you create it. You must enclose the name of the file in parentheses for the Control Program to recognize it.

If the file name is followed by ",A" (e.g., test.nc,A), the Control Program does not delete previous information from the file, rather it appends the information to the end of the existing information.

If you use more than one M22, only the first occurrence must have the file name in the parentheses. The remaining M22's may have empty parentheses, ( ).

**M25 Set robot output on (High):** Use for robot synchronization. Used in conjunction with H code to specify output number.

**M26 Set robot output off(Low):** Use for robot synchronization in conjunction with H code to specify output number.

**M30 Program stop:** Same as M02.

**M47 Rewind:** Restarts the currently running program; takes effect after all motion comes to a stop.

**M98 Call to subprogram**

**M99 Return from subprogram:** Returns you to the block following the initial M98 command.

**Go to:** Used with P code. P code defines N or 0 code destination. Goes to first occurrence of N or 0 code within the main program.

**M111 Arc as Linear Segments:** Causes the proLIGHT 2000 to make arcs as linear segments. Use with Helical Interpolation.

**M112 True Arcs:** Causes the proLIGHT 2000 to make true arcs. Opposite of M111.

**M122 Output current position to file:** Writes current X, Y, and Z position to a data file. Used with G 131 for digitizing with Light Machines' digitizing package. Similar to M22.