Operating the CNC Generation Program

The CNC Generator is a translation utility for creating machining G-codes from a high-level source format. The primary goal is to simplify the conceptual operation of the milling machine. Instead of concentrating on individual machining strokes, parts may be defined by their geometric shapes. Surfaces and shapes are usually described in single-line commands with human-readable parameters.

Structure of a Program
A program consists of a series of lines of text. Comments begin with a ";" and everything afterwards on a line is ignored. Commands are entered one-per-line, with any parameters included on the same line. Commands and parameters are separated by one or more spaces.

Each command line consists of an (optional) single-character Option Designator, followed by a command followed by a list of parameters. Commands and parameters are not case sensitive, with one exception: the quoted literal text in a TEXT command is treated as given.

The single-character Option Designator acts as an enabling toggle for the line. This allows a single program text file to be used to mill multiple different configurations. See the description of the OPTION command for more information. The Option Designator character may be omitted from any line, but if present will be the first non-blank character on the line.

The command is a single word indicating the particular command from the following list.

The parameters in the list take the form of (1) keyword, or (2) keyword=value, or (3) "string". A value is numeric and must contain a decimal point if it is non-zero. A text string is bracketed by either single- or double-quotes. Capitalization is preserved, as are spaces. Any parameter may be omitted; if it sets a value the value zero will be implied. For example, "REF" is equivalent to "REF X=0 Y=0 Z=0".

Commands are handled in order and generate zero or more output lines of G-code.

No optimization is performed over multiple source lines. Each command is treated individually and no machine state or operating mode carries over from one command to the next. This simplifies the conceptual process of creating a program by eliminating 'sticky' modes that invisibly affect the meaning of a command.

The only commands that can affect the operation of a subsequent command are listed here:

1	GLOBAL	Sets the location of the workpiece, the height and rate of the safe traverse.
2	STABLE	Sets the coordinates of the tool change stable
3	CUTTER	Sets the diameter and length of the new tool
4	REF	Sets a new coordinate system reference for subsequent commands
5	OPTION	Sets the enable flags for optionally processing each line.

Command Reference
Here is the concise list of all commands

Command	Parameters	Description
Setup Operations		
PRAGMA	**ISCALE=**v	Default value is 100,000. Fractions of an inch for internal integer calculations.
OPTION	!@#$%^&*()<>/?-=	Select the lines to be active in this program
GLOBAL	**X=**v **Y=**v **Z=**v **SAFE=**v **FEED=**v	Set the origin of the work area at (X,Y,Z). Set the SAFE height for moving to the tool stable. Set the FEED rate for all operations. Clears any previous REF
STABLE	**X=**v **Y=**v **Z=**v	Set the tool change stable location relative to the GLOBAL origin.
REF	**X=**v **Y=**v **Z=**v	Set the offset for subsequent cutting operations within the part
CUTTER **TOOL**	**DIAM=**v **LENGTH=**v **CUT=**v **SAFE=**v	The words CUTTER and TOOL are synonyms. Goes to the tool stable and pauses for change. Sets DIAM used in any IN, OUT or FILL operation Sets CUT for amount per-pass to move in Z Sets SAFE height to use between cutting operations
Cutting Operations		
ARC **CIR**	**IN** | **OUT** | **FILL** **X=**v **Y=**v **Z=**v **R=**v | **D=**v **DEPTH=**v **B=**v **E=**v	The words ARC and CIR are synonyms. The arc will be cut from beginning angle B to ending angle E in degrees. Leaving off B= and E= will mill a complete circle. The center is (X, Y). Either Radius R= or diameter D= may be given. IN compensates for the current cutter with the tool inside the shape. OUT compensates for the tool outside the shape. FILL is like IN, but also removes all interior material.
DRILL	**X=**v **Y=**v **Z=**v **DEPTH=**v	Move to the specified point (X, Y, Z). Peck drill to the specified DEPTH.

Command	Parameters	Description
ENGRAVE	**X=**v **Y=**v **Z=**v **XX=**v **YY=**v **DEPTH=**v **"**filename**"** | 'filename'	This is not implemented.
MILL	**X=**v **Y=**v **Z=**v **XX=**v | **W=**v **YY=**v | **H=**v **DEPTH=**v	Start at (X, Y, Z). Cut back and forth in a line to (XX, YY). Alternately, the second point may be (X+W, Y+H). Use the CUTTER Cut= value for the depth of each pass. Stop at Depth below the starting Z value
RECT	**IN** | **OUT** | **FILL** **X=**v **Y=**v **Z=**v **XX=**v | **W=**v **YY=**v | **H=**v **DEPTH=**v	Cut a rectangle whose corners are (X, Y) and (XX, YY). Alternately, the corner may be specified as (X+W, Y+H). Use the CUTTER CUT= value for the depth of each pass. Stop at DEPTH below the starting Z value
SLOT	**IN** | **OUT** | **FILL** **X=**v **Y=**v **Z=**v **XX=**v | **W=**v **YY=**v | **H=**v **DEPTH=**v **WIDTH=**v	This is not implemented.
TEXT	**X=**v **Y=**v **Z=**v **DEPTH=**v **HEIGHT=**v **A=**v **"**text**"** | 'text'	Engrave the text string. Baseline starts at (X, Y, Z) and extends out at angle A. Letters are HEIGHT tall and cut to DEPTH.

Parameter Reference
The following table describes the meaning of each of the possible command parameters. Usage is intended to be as consistent as possible from for each of the commands that use a particular parameter.

Parameter	Commands	Units	Description
A=v	**TEXT**	Degrees	Specify the baseline angle that the text will be drawn on.
B=v	**ARC** **CIR**	Degrees	Specify the beginning angle for the arc.
CUT=v	**CUTTER** **TOOL**	Inches	Specify the maximum depth of cut that the tool will make on each pass.
D=v	**ARC** **CIR**	Inches	Specify the Diameter of the circle. Alternately, use **R=**v to specify the radius.
DEPTH=v	**ARC** **CIR** **MILL** **RECT**	Inches	Specify the depth below the Z= height that the cut will finish.
DIAM=v	**CUTTER** **TOOL**	Inches	Specify the diameter of the cutting tool.
E=v	**ARC** **CIR**	Degrees	Specify the ending angle for an arc.
FEED=v	**GLOBAL**	Inches per Minute	Specifies the feed rate that will be used for all operations.
FILL	**ARC** **CIR** **RECT** **SLOT**		Specifies that the interior of the shape will be milled away.
H=v	**MILL** **RECT** **SLOT**	Inches	Specify the height of the rectangular work area to mill. Alternately, specify ending position with **YY=**v.
HEIGHT=v	**TEXT**	Inches	Specifies the height of the finished text. Does not compensate for cutter diameter or descenders.
IN	**ARC** **CIR** **RECT** **SLOT**		Specifies that the tool will cut inside the specified shape.
ISCALE=v	**PRAGMA**		Specifies the accuracy to which output will be presented. The default is 100000, which means output will be in the form +0.00000
LENGTH=v	**CUTTER** **TOOL**	Inches	Specifies the maximum depth into the work that the tool may penetrate.

Parameter	Commands	Units	Description
OUT	**ARC** **CIR** **RECT** **SLOT**		Specifies that the tool will cut outside the specified shape.
R=v	**ARC** **CIR**	Inches	Specify the Radius of the circle. Alternately, use **D=**v to specify the diameter.
SAFE=v	**GLOBAL** **CUTTER** **TOOL**	Inches	
W=v	**MILL** **RECT** **SLOT**	Inches	Specify the width of the rectangular work area to mill. Alternately, specify ending position with **XX=**v.
WIDTH=v	**SLOT**	Inches	
X=v	**ARC** **CIR** **MILL** **RECT** **SLOT** **TEXT**	Inches	Specify the center of an arc or circle as (X, Y). Specify the starting point of a MILL as (X, Y). Specify the starting corner of a RECT, or SLOT as (X, Y). Specify the starting baseline of text as (X, Y).
X=v	**REF**	Inches	Specifies the reference offset to be added to the GLOBAL workpiece position for each operation.
XX=v	**MILL** **RECT** **SLOT**	Inches	Specify the ending position of the rectangular work area to mill. Alternately, specify height with **W=**v.
Y=v	**ARC** **CIR** **MILL** **RECT** **SLOT** **TEXT**	Inches	Specify the center of an arc or circle as (X, Y). Specify the starting point of a MILL as (X, Y). Specify the starting corner of a RECT, or SLOT as (X, Y). Specify the starting baseline of text as (X, Y).
Y=v	**REF**	Inches	Specifies the reference offset to be added to the GLOBAL workpiece position for each operation.
YY=v	**MILL** **RECT** **SLOT**	Inches	Specify the ending position of the rectangular work area to mill. Alternately, specify height with **H=**v.
Z=v	**ARC** **CIR** **MILL** **RECT** **SLOT** **TEXT**	Inches	Specifies the height of the work surface at the start of the operation. The tool will move to (X, Y, Z) to begin the cutting. At the end of the operation, the final cut will be to DEPTH inches below Z.

Parameter	Commands	Units	Description
Z=v	**REF**	Inches	Specifies the reference offset to be added to the GLOBAL workpiece position for each operation.

Sample Program
The sample program below illustrates the use of most of the commands.

```
GLOBAL X=1. Y=0 Z=0 Feed=10. Safe=1.
stable X=-1. Z=2.

CUTTER diam=.1250 length=0.5 cut=.1250 safe=.1        ; This is a 1/8 drill
bit
drill x=-.5000  y=0  depth=.5      ; Alignment holes
drill x=-.5000  y=2.5  depth=.5  ; Alignment holes


;-- reference machine zero lower left corner
;-- center of head circle 15650+1450=17100
ref x=.6400 y=1.7100 z=0  ; center of head

drill x=.1400  y=0 depth=.3750                         ; holes
drill x=-.1400 y=0 depth=.3750


CUTTER diam=.15625 length=.375 cut=.050 safe=.1        ; This is 5/32
ref
rect out yy=.23500 xx=.12800 depth=.3750               ; Outside of part

ref x=.6400 y=1.7100 z=0  ; Center of head
cir out r=.6400 depth=.3750 b=0 e=180.                 ; Outside of head
cir fill r=.5000 depth=.1600 b=-45. e=225.             ; Should be dee
rect fill x=-.3500 y=-.3500 xx=.3500 yy=0 depth=.1600  ; flat on dee bottom
rect fill x=-.7000 y=-.7000 xx=.7000 yy=-.5 depth=.2   ; flat on the outside
cir fill r=.3000 depth=.2050                           ; Filter cutout

            ; mill the two holes
CUTTER diam=.1250 length=.3750 cut=.0500 safe=.1       ; This is 1/8 mill
cir in x=.1400  y=0 z=-.2050 r=.1000 depth=.3750       ; holes
cir in x=-.1400 y=0 z=-.2050 r=.1000 depth=.3750

CUTTER diam=.3750 length=1.1000 cut=.0500 safe=.1      ; This is 3/8 mill
ref         ; go back to global coordinates
            ; Remove the base level from the origin
rect fill x=0 y=0 xx=1.2800 yy=1.2100 depth=.2
```

Error Messages
Each line of the source file is translated into appropriate G-codes for the output to the machine. It is possible that conflicts may arise during this generation process. The translator will stop at the error line and will place diagnostic messages in the output file.

If the translation is successful, the output will end with "**m02 (Program End)**".

If an error occurred the output will end with "**(Program Error)**" and a detail message will be embedded in the output file.

-Error: Decimal Point Required. "n" All non-zero numbers must contain a decimal point. "n" is the offending value from the current command line.

-Error: Invalid Number. "n" Numbers must contain only digits 0-9, "+", "-", and ".". "n" is the offending value from the current command line.

-Error: Bad Cut Depth Cut depth must be greater than zero.

-Error: Depth for tool. d > n Depth of cut must not be greater than the length specified in the TOOL command. "d " is the requested depth, "n" is the tool length.

-Error: Radius for tool. r <= R The diameter of an inside circle or arc is smaller than the tool diameter. "r" is the requested arc radius, "R" is the radius of the current tool.

-Error: Require IN, OUT or FILL One of the modifiers is required for CIR, ARC and RECT commands. This allows the proper tool compensation to yield the correct size finished shape.

-Error: Size for tool. The tool diameter is larger than the specified dimensions of an inside rectangle.

-Error: -SLOT NOT IMPLEMENTED The slot command is not supported in this version.

-Error: Invalid Parameter - n The specified parameter "n" is not recognized for the command.

-Error: Illegal Option Character "c" The character "c" is not legal for use as an option designator.

-Error: Unknown Command. The command is not recognized.