The Official
Generative Machining
ASCII CLDATA Output Specification
August 9, 2000
Version 8.1.1

```
MODE/MILL
MULTAX/ON
GOHOME
LOADTL/0, IN, 1, LENGTH, 0.000000, OSETNO, 0
CUTTER/6.000000
LINTOL/0.050000
SPINDL/1910.000, RPM, CLW
RAPID
GOTO/308.427225, -75.000001, 143.449146, 0.819150, 0.000000, 0.573580
RAPID
GOTO/308.427225, -75.000001, 86.310234, 0.819150, 0.000000, 0.573580
COOLNT/FLOOD
CYCLE/DRILL, 14.802582, MMPR, 0.127000, 10.000000, RAPTO, 5.000000
GOTO/267.453123, -75.000001, 57.619609, 0.819150, 0.000000, 0.573580
CYCLE/OFF
RAPID
GOTO/308.427225, -75.000001, 86.310234, 0.819150, 0.000000, 0.573580
RAPID
GOTO/308.427225, -75.000001, 143.449146, 0.819150, 0.000000, 0.573580
RAPID
GOTO/318.379825, -112.505078, 143.449146, 0.819152, 0.000000, 0.573576
RAPID
GOTO/318.379825, -112.505078, 72.095689, 0.819152, 0.000000, 0.573576
CYCLE/DRILL, 14.802582, MMPR, 0.127000, 10.000000, RAPTO, 5.0000
GOTO/318.379825, -112.505078, 43.405482, 0.819152, 0.000000, 0.573576
CYCLE/OFF
RAPID
GOTO/318.379825, -112.505078, 72.095689, 0.819152, 0.000000, 0.573576
RAPID
GOTO/318.379825, -112.505078, 143.449146, 0.819152, 0.000000, 0.573576
RAPID
GOTO/300.503903, -57.868905, 143.449146, 0.819152, 0.000000, 0.573576
RAPID
GOTO/300.503903, -57.868905, 97.625152, 0.819152, 0.000000, 0.573576
CYCLE/DRILL, 14.802582, MMPR, 0.127000, 10.000000, RAPTO, 5.000000
GOTO/259.530040, -57.868905, 68.934944, 0.819152, 0.000000, 0.573576
CYCLE/OFF
COOLNT/OFF
RAPID
GOTO/300.503903, -57.868905, 97.625152, 0.819152, 0.000000, 0.573576
RAPID
GOTO/300.503903, -57.868905, 143.449146, 0.819152, 0.000000, 0.573576
GOHOME
END
FINI
```

1. ASCII CLDATA Vocabulary

This section specifies the format of the Generative Machining ASCII cldata output. This output is based on the ISO-4343 1978 standard. Parameters enclosed in brackets [,aaa | ,bbb | ,ccc] indicate a choice of parameters. The vertical bar | divides the choices. Parameters enclosed in braces {xxx, yyy, zzz} indicate optional parameters. Optional parameters with choices are enclosed in braces and brackets {[,xxx | ,yyy | ,zzz]}. The ~ symbol preceding items in brackets or braces indicates that the item can be repeated one or more times. Parameters specified in upper case are used literally in the format. Valid entries for all other parameters are detailed following the command format.

1.1. Motion Definition and Control Commands

1.1.1. GOTO

This statement produces a linear motion at the present feedrate (given by FEDRAT).

GOTO/ x, y, z {,i, j, k}

Variable	Type	Description
x	Real	X Axis position
y	Real	Y Axis position
z	Real	Z Axis position
i	Real	X Axis component of tool axis vector
j	Real	Y Axis component of tool axis vector
k	Real	Z Axis component of tool axis vector

Usage

The IJK values are only output in MULTAX mode, which must be declared by the MULTAX/ON statement.

1.1.2. MOVARC

The MOVARC statement defines a circular move.

MOVARC/ x1, y1 ,z1 ,i ,j ,k ,r, ANGLE, a
GOTO/ x2, y2, z2 {, i2, j2,k2 }

Variable	Type	Description
x1	Real	X Axis position of the arc center
y1	Real	Y Axis position of the arc center
z1	Real	Z Axis position of the arc center
i	Real	X Axis component of arc (circle) axis vector
j	Real	Y Axis component of arc (circle) axis vector
k	Real	Z Axis component of arc (circle) axis vector
r	Real	Radius of the arc
a	Real	subtended (included) angle of the arc
x2,y2,z2	Real	arc end point

Usage

The GOTO or MOVARC statement immediately preceding the MOVARC statement is the start point of the circle. For example:


```
                      GOTO/0,2,0
                      MOVARC/0,0,0,0,0,-1,2,90
                      GOTO/2,0,0
```
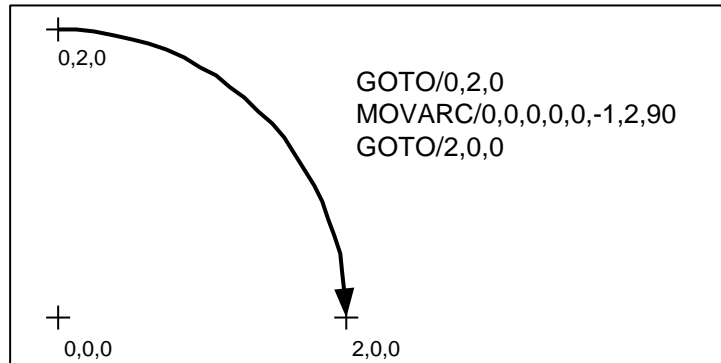
In this example, the start point of the arc is 0,2,0. The arc center is at 0,0,0 and the cylinder axis is 0,0,-1 (the -1 signifying that the cutting direction is clockwise; a positive vector signifies counter-clockwise direction, i.e. right hand rule). The arc radius is 2 and the end point is 2,0,0.

The MOVARC must be followed by a single GOTO statement specifying the coordinates of the end point.

MOVARCs are output in XYZIJK (multax) mode as well as the XYZ mode.

MOVARCS are output for all circular motion in all planes.

1.1.3. GOHOME

This statement is used to move the tool to a programmer specified home position. The actual position can be derived from the values in the FROM statement, or determined by the postprocessor.

GOHOME

Usage

The GOHOME statement is not output by Generative Machining. It can be added by the user in Transition moves or by editing the toolpath.

1.1.4. RAPID

This statement is used to indicate that the next linear motion is to be at rapid feedrate. Note that it affects only the next motion. Subsequent motions are at the previously programmed FEDRAT value.

RAPID

Usage

RAPID is output on a line by itself immediately preceding the GOTO statement that it modifies.
RAPID only effects the subsequent GOTO statement.

Generative Machining will always output the RAPID statement immediately before the effected GOTO statement that it.

1.1.5. FROM

This statement specifies the "home" position for the tool.

FROM/ x, y, z, {i, j, k}

Variable	Type	Description
x	Real	X Axis position
y	Real	Y Axis position
z	Real	Z Axis position
i	Real	X Axis component of tool axis vector
j	Real	Y Axis component of tool axis vector
k	Real	Z Axis component of tool axis vector

Usage

The FROM statement is only output when there is a machine defined (which only affects milling programs. Turning programs must always have a machine defined).

In milling programs, the FROM statement is output once at the beginning of the program before the first tool change. The XYZ coordinates in the FROM statement are those specified by the user on the machine form for the HOME POINT.

In turning programs, a FROM statement is output immediately after the first HEAD statement for each turret (presently a maximum of two). The XYZ coordinates in the FROM statements are those specified by the user on the machine form for the HOME POINT for the TURRET being used.

1.1.6. MULTAX

This statement indicates that output in the form of XYZIJK will follow. MULTAX/ ON will precede any GOTO or FROM statement which contains i, j, k values. MULTAX statements will be re-issued after each tool change.

MULTAX/ [ON, OFF]

1.1.7. CUTTER

1.1.7.1. Milling Format

For Milling, the CUTTER statement is now specified using a seven parameter statement containing the values which define the cutter configuration used to compute the CLDATA locations.

CUTTER/d, r, e, f, a, b, h

Variable	Type	Description

d	Real	Tool diameter
r	Real	Tool corner radius
e	Real	Radial distance from tool center to center of corner radius
f	Real	Axial distance from tip of tool to center of corner radius
a	Real	Angle of tool tip
b	Real	Tool side angle
h	Real	Tool length
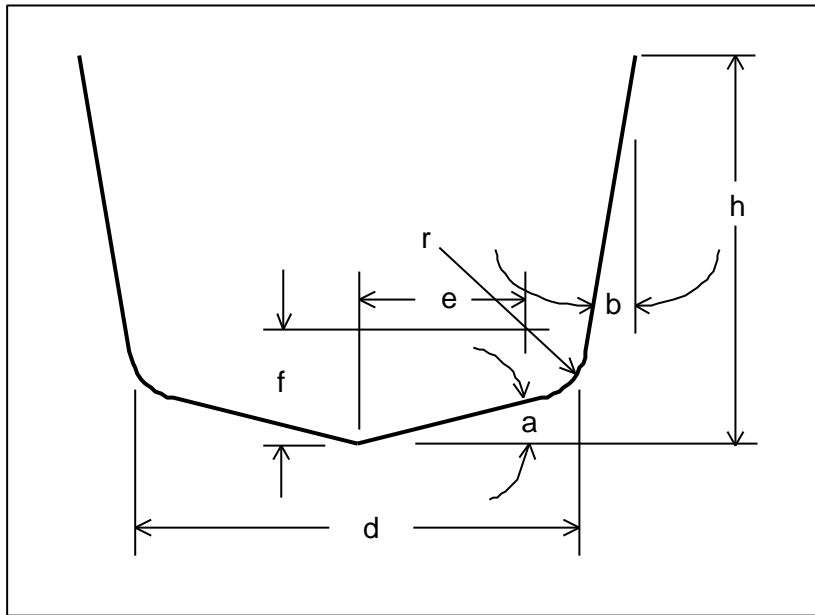
1.1.7.2. Turning Format

For Turning, the CUTTER statement specifies the tool nose diameter (the nose radius x 2).

CUTTER/d

Variable	Type	Description
d	Real	Tool nose diameter (twice the tool nose radius)

Usage

The CUTTER statement is output after every tool change (LOADTL or TURRET) statement. Only the tool diameter and length are specified by Generative Machining.

1.1.8. ORIGIN

This statement defines the relationship of the machine coordinate system to the part coordinate system. Its primary purpose is to support the use of fixture offsets and localized toolpath output.

ORIGIN/ x, y, z {, I, J, K }

Variable	Type	Description
x	Real	The distance from the operation origin to the machine origin, measured along the machine X axis
y	Real	The distance from the operation origin to the machine origin, measured along the machine Y axis
z	Real	The distance from the operation origin to the machine origin, measured along the machine Z axis
I	Real	The X axis component of the tool axis (Operation Coordinate System Z) vector for the operation
J	Real	The Y axis component of the tool axis (Operation Coordinate System Z) vector for the operation
K	Real	The Z axis component of the tool axis (Operation Coordinate System Z) vector for the operation

Usage

The ORIGIN statement is used to specify the distance **from** the operation coordinate system **to** the machine coordinate system and is output whenever this relationship changes. The ORIGIN XYZ values used are based upon the operation coordinate system used in Generative Machining.

The ORIGIN statement is not used for Turning or MIll/Turn machines.

For MULTAX (IJK) output, the X, Y, Z values in the ORIGIN statement are the distance **from** the Operation Coordinate System **to** the Machine Tool Coordinate System, in the Machine Tool Coordinate system frame of reference.

The IJK values are only output in MULTAX mode. Otherwise, only the translation (XYZ) values are output.

For ROTABL output, Generative Machining transforms the toolpath data (and hence, the operation coordinate system) to the location it should be in after the tables are indexed. The ORIGIN values are then specified **from** the transformed location **to** the Machine Tool Coordinate System, in the Machine Tool Coordinate system frame of reference.

Direction and Orientation of ORIGIN statement values (MULTAX Output Example)
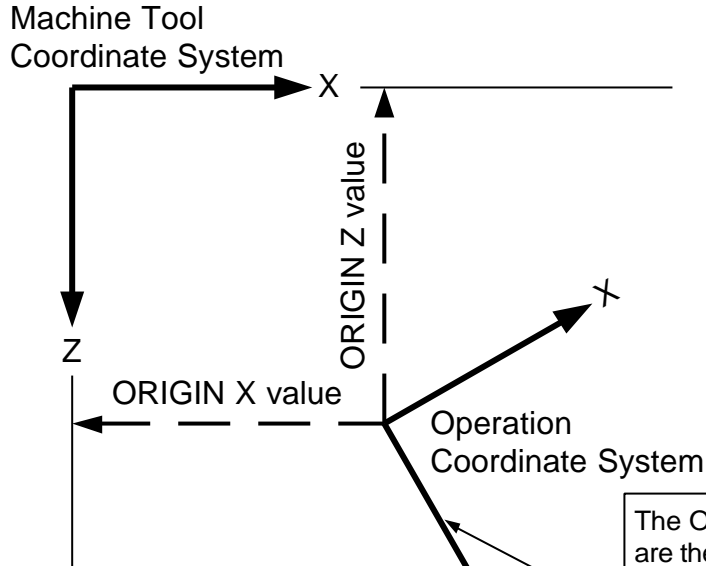
Machine Tool
Coordinate System

X

ORIGIN Z value

Z

ORIGIN X value

X

Operation
Coordinate System

Z

The ORIGIN IJK values are the XYZ components of the Operation Coordinate System Z axis specified relative to the Machine Coordinate system

The X, Y, Z values in the ORIGIN statement are the distance **from** the Operation Coordinate System **to** the Machine Tool Coordinate System, in the Machine Tool Coordinate system frame of reference.

Direction and Orientation of ORIGIN statement values
(ROTABL Output Example)

For ROTABL output, Generative Machining transforms the toolpath data to the location it should be in after the tables are indexed. The ORIGIN values are then specified from this position to the machine coordinate system.

The X, Y, Z values in the ORIGIN statement are the distance **from** the **transformed** Operation Coordinate System **to** the Machine Tool Coordinate System, in the Machine Tool Coordinate system frame of reference.

1.2. Cycle Statements

1.2.1. CYCLE - General

Cycle statements are used to indicate a machine tool cycle mode for axial machining operations such as drilling and tapping. Cycle statements are always followed by GOTO commands indicating the x,y,z {I,j,k} locations at which the cycles are to be performed. The coordinates of the GOTO statement are the hole selected by the user. The cycle is in effect until another cycle statement is programmed changing the cycle type it is canceled with a CYCLE/OFF CLDATA record.
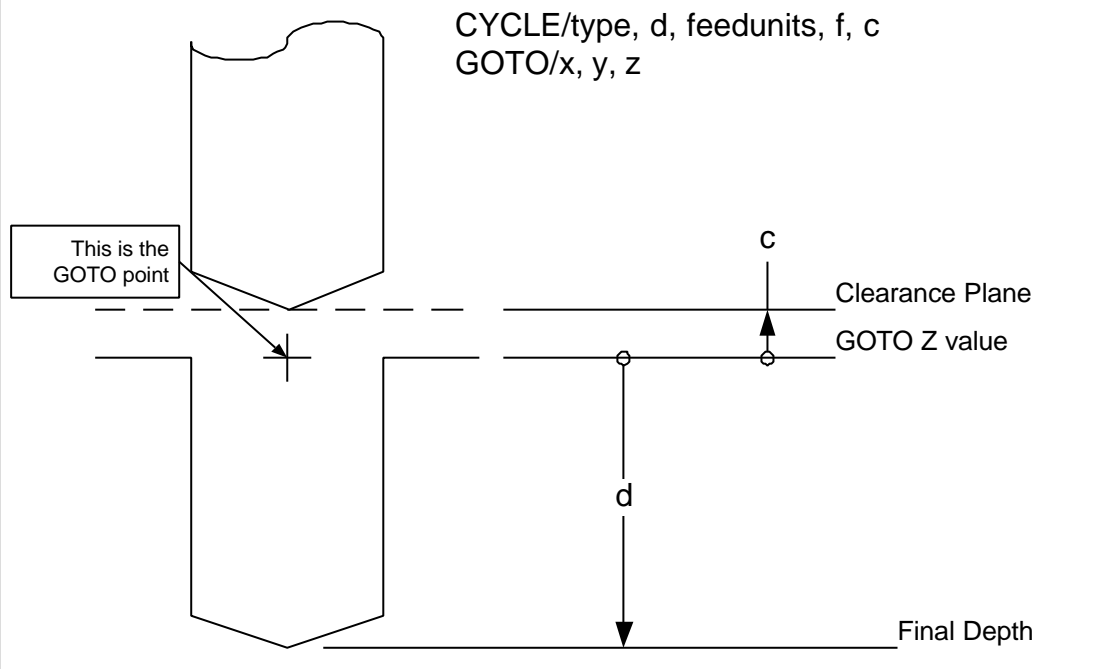
CYCLE statements in Generative Machining have the general form:

CYCLE/type, d, feedunits, f, c {RAPTO, r,}
GOTO/x, y, z {,I,j,k}
.

.
GOTO/x, y, z {,I,j,k}
CYCLE/OFF

Variable	Type	Description
type	Text	Cycle Type - BORE, BRKCHP, DEEP, DRILL, FACE, REAM, TAP
d	Real	Depth - measured along the hole axis from the GOTO Z value to the final depth
feedunits	Text	Feedrate type - IPM, IPR, MMPM, MMPR
f	Real	Feedrate value
c	Real	Clearance Distance - the distance from the GOTO Z value to the clearance plane, measured along the hole axis
r	Real	Rapid Advance Distance - Measured along the hole axis from the clearance plane value to the R plane (where the tool starts moving at feedrate).

Basic Cycle Example

CYCLE/type, d, feedunits, f, c
GOTO/x, y, z

This is the GOTO point

c

Clearance Plane

GOTO Z value

d

Final Depth

RAPTO Cycle Example

CYCLE/type, d, feedunits, f, c {,RAPTO, r} {,DWELL, h}
GOTO/x, y, z

This is the
GOTO point

Clearance Plane

r

c

Engage Plane

GOTO Z value

d

Final Depth
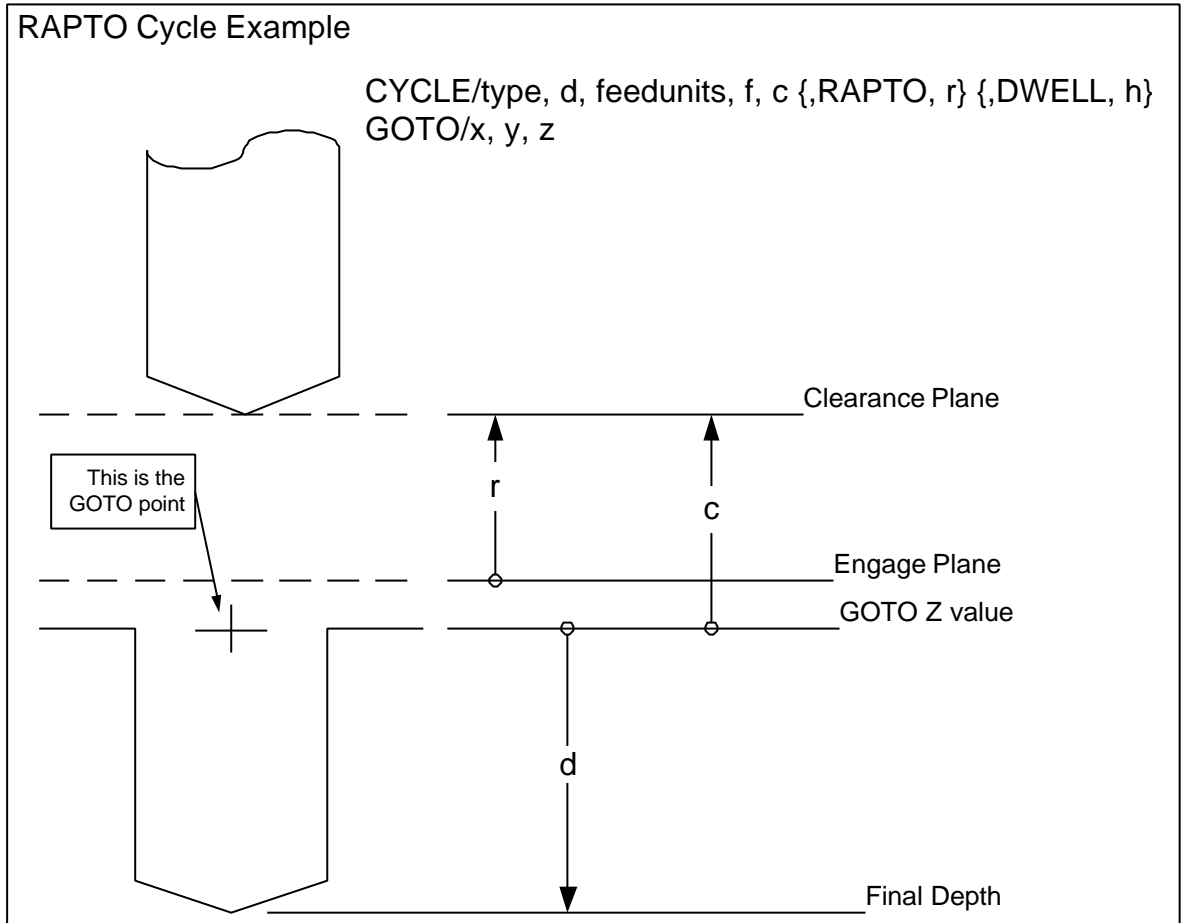
Usage

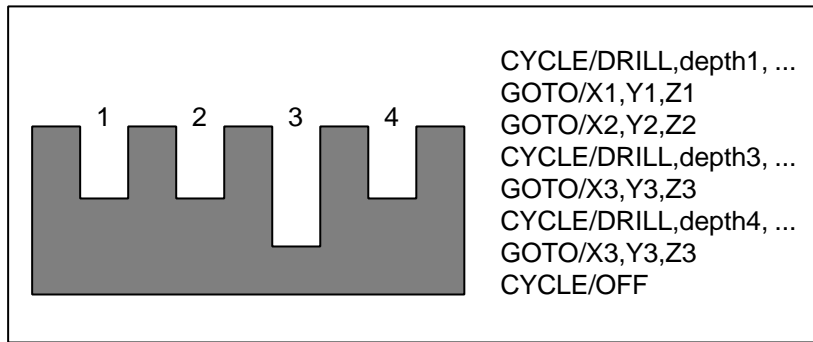In any cycle, if the dwell value is 0 or is not set, the DWELL couplet must not be output.
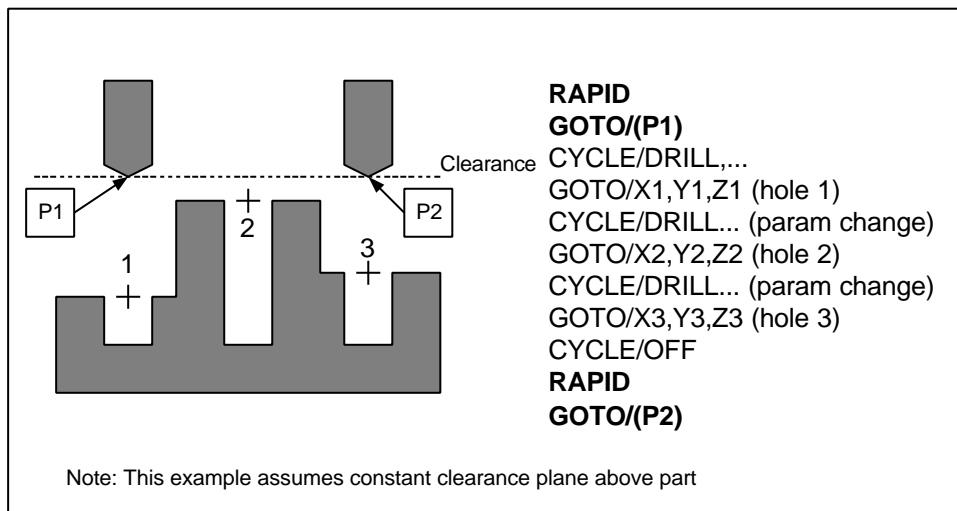
When the CYCLE parameters are the same for multiple holes in a single operation, a single CYCLE statement followed by several GOTO statements will be output.

1   2   3

CYCLE/DRILL...
GOTO/X1,Y1,Z1
GOTO/X2,Y2,Z2
GOTO/X3,Y3,Z3
CYCLE/OFF
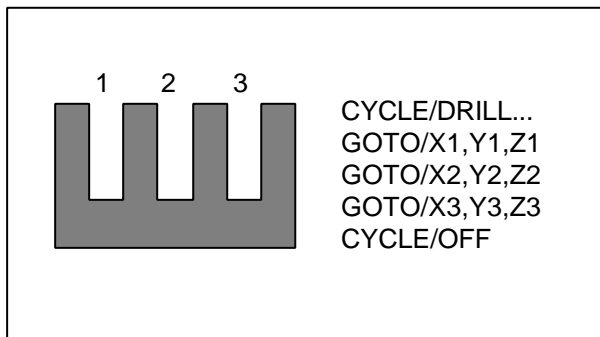
When a hole parameter changes, the CYCLE statement is restated with the changed parameters:

```
CYCLE/DRILL,depth1, ...
GOTO/X1,Y1,Z1
GOTO/X2,Y2,Z2
CYCLE/DRILL,depth3, ...
GOTO/X3,Y3,Z3
CYCLE/DRILL,depth4, ...
GOTO/X3,Y3,Z3
CYCLE/OFF
```

Each operation, prior to machining a series of holes, first positions to the clearance plane above the first hole. At the end of the series of holes, the CYCLE is canceled, and and a RAPID move is output to move the tool to the clearance plane above the final hole (it should be there anyway. This is done as a safety measure).


```
RAPID
GOTO/(P1)
CYCLE/DRILL...
GOTO/X1,Y1,Z1
GOTO/X2,Y2,Z2
GOTO/X3,Y3,Z3
CYCLE/OFF
RAPID
GOTO/(P2)
```



```
RAPID
GOTO/(P1)
CYCLE/DRILL,...
GOTO/X1,Y1,Z1 (hole 1)
CYCLE/DRILL... (param change)
GOTO/X2,Y2,Z2 (hole 2)
CYCLE/DRILL... (param change)
GOTO/X3,Y3,Z3 (hole 3)
CYCLE/OFF
RAPID
GOTO/(P2)
```

Note: This example assumes constant clearance plane above part
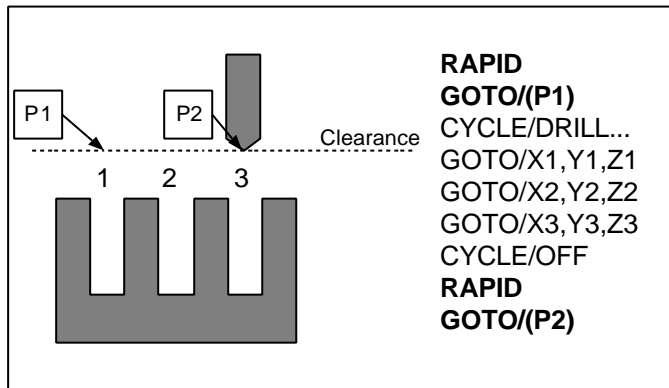
In Generative Machining holemaking operations, the user has a choice of using a single clearance plane that the tool retracts to after machining every hole, or having the tool retract only as far as necessary to clear an obstacle by the clearance value.

Multiple Height and Depth Example, Retract to Clearance Between Each Hole

Engage Distance = .1
(3 places)

P1

Clearance Plane Z = 5.0

P2

Z=4

Z=2

3   4
Depth = 1.25

1   2

Depth = .75

Depth = .5 **5**
Depth = .625 **6**

Z=1

→ Feed Move
----→ Rapid Move

```
RAPID
GOTO/ (P1)
CYCLE/DRILL,.75, IPR,.01, 3,RAPTO,2.9
GOTO/x,y,2   (Hole 1)
GOTO/x,y,2   (Hole 2)
CYCLE/DRILL,1.25,IPR,.01,1,RAPTO,.9
GOTO/x,y,4   (Hole 3)
GOTO/x,y,4   (Hole 4)
CYCLE/DRILL,.5,IPR,.01,4,RAPTO,3.9
GOTO/x,y,1   (Hole 5)
CYCLE/DRILL,.625,IPR,.01,4,RAPTO,3.9
GOTO/x,y,1   (Hole 6)
CYCLE/OFF
RAPID
GOTO/ (P2)
```
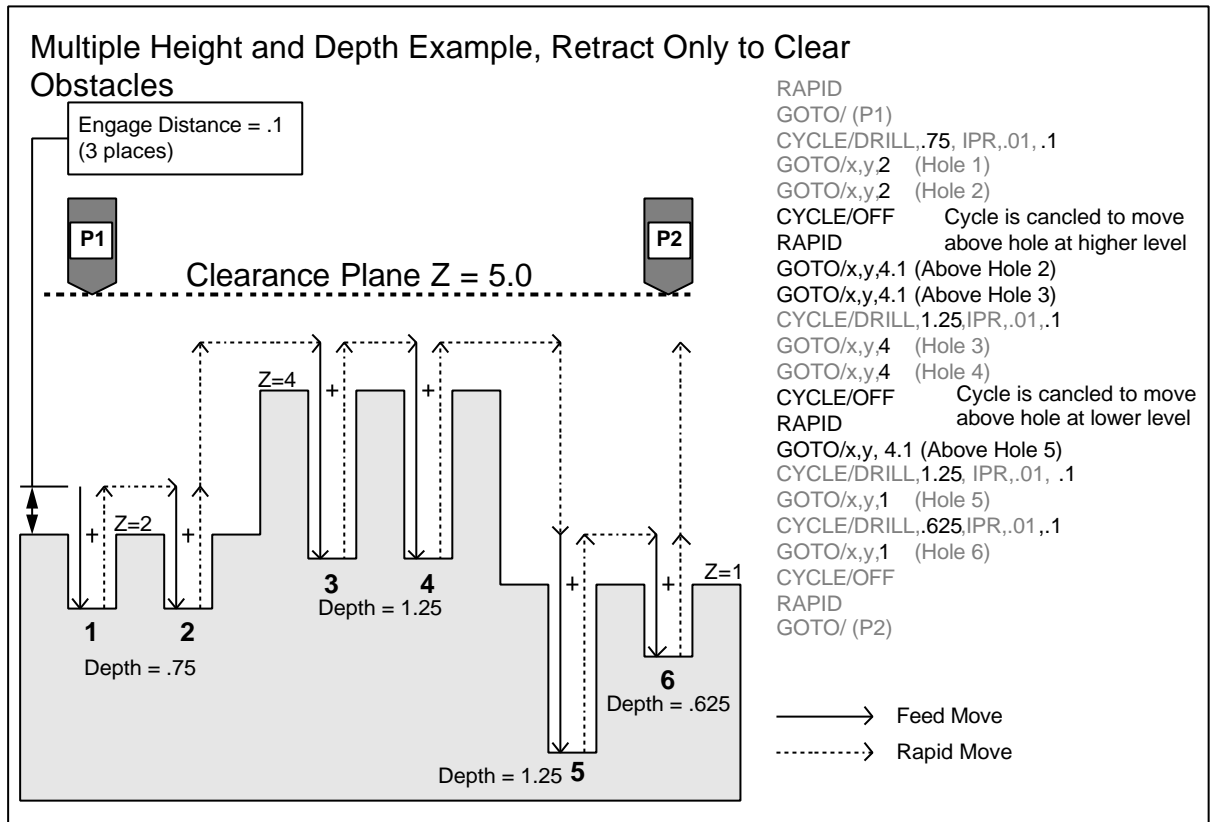
The option to have the tool retract only to clear obstacles is called "Retract to Minimum Z plus Clearance". When this option is used, the user specifies a clearance height, which is typically a small value. the tool will move at the user specified level above the holes unless there is an obstacle to be cleared or the next hole is at a different level.

When there is an obstacle to be cleared, the cycle is canceled, A RAPID move along the tool axis occurs to move the tool to the clearance required to clear the obstacle, followed by another RAPID move to position the tool above the next hole.

When the next hole in a series is higher, the cycle is canceled, A RAPID move along the tool axis occurs to move the tool to the clearance plane for the higher hole, followed by another RAPID move to position the tool above the next hole. This is identical to the action for clearing an obstacle.

When the next hole in a series is lower, the cycle is canceled and a RAPID move is inserted to move the tool above the next hole at the clearance plane level from the previous hole.

Multiple Height and Depth Example, Retract Only to Clear Obstacles

Engage Distance = .1
(3 places)

P1

Clearance Plane Z = 5.0

P2

Z=4

Z=2

3   **4**
Depth = 1.25

1   **2**

Depth = .75

6
Depth = .625

Z=1

Depth = 1.25 **5**

```
RAPID
GOTO/ (P1)
CYCLE/DRILL,.75, IPR,.01,.1
GOTO/x,y,2    (Hole 1)
GOTO/x,y,2    (Hole 2)
CYCLE/OFF        Cycle is cancled to move
RAPID                above hole at higher level
GOTO/x,y,4.1 (Above Hole 2)
GOTO/x,y,4.1 (Above Hole 3)
CYCLE/DRILL,1.25,IPR,.01,.1
GOTO/x,y,4    (Hole 3)
GOTO/x,y,4    (Hole 4)
CYCLE/OFF        Cycle is cancled to move
RAPID                above hole at lower level
GOTO/x,y, 4.1 (Above Hole 5)
CYCLE/DRILL,1.25, IPR,.01,  .1
GOTO/x,y,1    (Hole 5)
CYCLE/DRILL,.625,IPR,.01 ,.1
GOTO/x,y,1    (Hole 6)
CYCLE/OFF
RAPID
GOTO/ (P2)
```

——————→ Feed Move

----------> Rapid Move

1.2.2.   CYCLE/ OFF

This statement cancels the previously programmed cycle.

CYCLE/ OFF

1.2.3.   CYCLE/ DRILL

CYCLE/ DRILL,d, [IPR | IPM | MMPR | MMPM], f,c {,RAPTO,r} {,DWELL,[q | REV, p]}

Variable	Type	Description
d	Real	Depth - measured along the hole axis from the GOTO Z value to the final depth
f	Real	Feedrate value
c	Real	Clearance Distance - the distance from the GOTO Z value to the clearance plane, measured along the hole axis
r	Real	Rapid Advance Distance - Measured along the hole axis from the clearance plane value to the R plane (where the tool starts moving at feedrate).
q	Real	Dwell time in seconds
p	Real	Dwell time in spindle revolutions

1.2.4. CYCLE/ TAP

CYCLE/ TAP,d, [IPR | IPM | MMPR | MMPM], f,c {,RAPTO,r}

Variable	Type	Description
d	Real	Depth - measured along the hole axis from the GOTO Z value to the final depth
f	Real	Feedrate value
c	Real	Clearance Distance - the distance from the GOTO Z value to the clearance plane, measured along the hole axis
r	Real	Rapid Advance Distance - Measured along the hole axis from the clearance plane value to the R plane (where the tool starts moving at feedrate).

1.2.5. CYCLE/ BORE

This cycle generates several boring output cycle statements depending on the options specified.

CYCLE/ BORE,d, [IPR | IPM | MMPR | MMPM],f,c {,j } {,RAPTO,r} {,ORIENT {,a }} {,DWELL,[q | REV, p]}

Variable	Type	Description
d	Real	Depth - measured along the hole axis from the GOTO Z value to the final depth
f	Real	Feedrate value
c	Real	Clearance Distance - the distance from the GOTO Z value to the clearance plane, measured along the hole axis
j	Real	Jog Distance (the ORIENT minor word must be specified)
r	Real	Rapid Advance Distance - Measured along the hole axis from the clearance plane value to the R plane (where the tool starts moving at feedrate).
a	Real	Orient Angle (the ORIENT minor word must be specified)
q	Real	Dwell time in seconds
p	Real	Dwell time in spindle revolutions

CYCLE/ REAM

The REAM cycle performs a "feed in - feed out" sequence of operations.

CYCLE/ REAM,d, [IPR | IPM | MMPR | MMPM], f,c {,RAPTO,r} {,DWELL,[q | REV, p]}

Variable	Type	Description
d	Real	Depth - measured along the hole axis from the GOTO Z value to the final depth
f	Real	Feedrate value
c	Real	Clearance Distance - the distance from the GOTO Z value to the clearance plane, measured along the hole axis
r	Real	Rapid Advance Distance - Measured along the hole axis from the clearance plane value to the R plane (where the tool starts moving at feedrate).
q	Real	Dwell time in seconds

| p | Real | Dwell time in spindle revolutions |

1.2.6. CYCLE/ DEEP

The DEEP cycle is a "chip clearing" cycle normally used for deep holes. The tool retracts to the R plane between depth increments to clear the chips Both constant and varying depths are supported.

CYCLE/ DEEP,d,INCR,i1,i2,....,in, [IPR | IPM | MMPR | MMPM], f,c {,RAPTO,r} {,DWELL,[q | REV, p]}

Variable	Type	Description
d	Real	Depth - measured along the hole axis from the GOTO Z value to the final depth
i1, i2, …	Real	Incremental depth values. The last increment value is repeated until the final hole depth is reached. A maximum of 10 depth increments is output.
f	Real	Feedrate value
c	Real	Clearance Distance - the distance from the GOTO Z value to the clearance plane, measured along the hole axis
r	Real	Rapid Advance Distance - Measured along the hole axis from the clearance plane value to the R plane (where the tool starts moving at feedrate).
q	Real	Dwell time in seconds
p	Real	Dwell time in spindle revolutions

1.2.7. CYCLE/ BRKCHP

The BRKCHP cycle is a "chip break" cycle normally used for machining holes in materials that produce stringy, continuous chips. The tool retracts a small amount (determined by the postprocessor or a machine tool setting) between depth increments to break the chips Both constant and varying depths are supported.

CYCLE/ BRKCHP,d,INCR,i1,i2,....,in, [IPR | IPM | MMPR | MMPM], f,c {,RAPTO,r} {,DWELL,[q | REV, p]}

Variable	Type	Description
d	Real	Depth - measured along the hole axis from the GOTO Z value to the final depth
i1, i2, …	Real	Incremental depth values. The last increment value is repeated until the final hole depth is reached. A maximum of 10 depth increments is output.
f	Real	Feedrate value
c	Real	Clearance Distance - the distance from the GOTO Z value to the clearance plane, measured along the hole axis
r	Real	Rapid Advance Distance - Measured along the hole axis from the clearance plane value to the R plane (where the tool starts moving at feedrate).
q	Real	Dwell time in seconds
p	Real	Dwell time in spindle revolutions

1.3. Machine / Postprocessor Control Commands

1.3.1. UNITS

The UNITS command specifies the CLDATA file units.

UNITS/ [INCHES | MM]

Usage

The UNITS command is output once to the CLDATA file.
It is the second command in the file immediately following the PARTNO command.

1.3.2. AUXFUN

This statement is used to output a machine controller "M" code directly to the part program.

AUXFUN/ m ~[,m]

Variable	Type	Description
m	Integer	The M code (auxiliary function) code number.

Usage

AUXFUN is not output automatically by Generative Machining, but can be added by the user via Transition moves or toolpath editing.

1.3.3. COOLNT

This statement is used to turn coolant on and off.

COOLNT/ [ON | OFF | MIST | FLOOD | TAP | THRU]

Usage

The COOLNT statement is output once at the beginning of an operation if the coolant mode is any of the "on" conditions (ON, MIST, FLOOD, TAP, THRU). A COOLNT/OFF statement is output at the end of the operation to turn off the coolant.

If coolant is specified to be off for the operation, no COOLNT commands are output (the coolant is assumed to be turned off at the beginning of the operation since any operation specifying coolant should have concluded with a COOLNT/OFF statement).

1.3.4. CUTCOM

This statement is used to turn cutter compensation on and off. It is also used to select the side, with respect to the following motion vector, that compensation will be applied to. It can optionally specifiy the cutter compensation control register.

CUTCOM/ [RIGHT | LEFT] {,d}

Variable	Type	Description
d	Integer	The cutter radius compensation register number.

CUTCOM/ON

This turns CUTCOM on, using the settings (plane, register, side, etc.) that were last used. Generative Machining does not create this form of the command, but it's possible to add it manually via the toolpath editing capability.
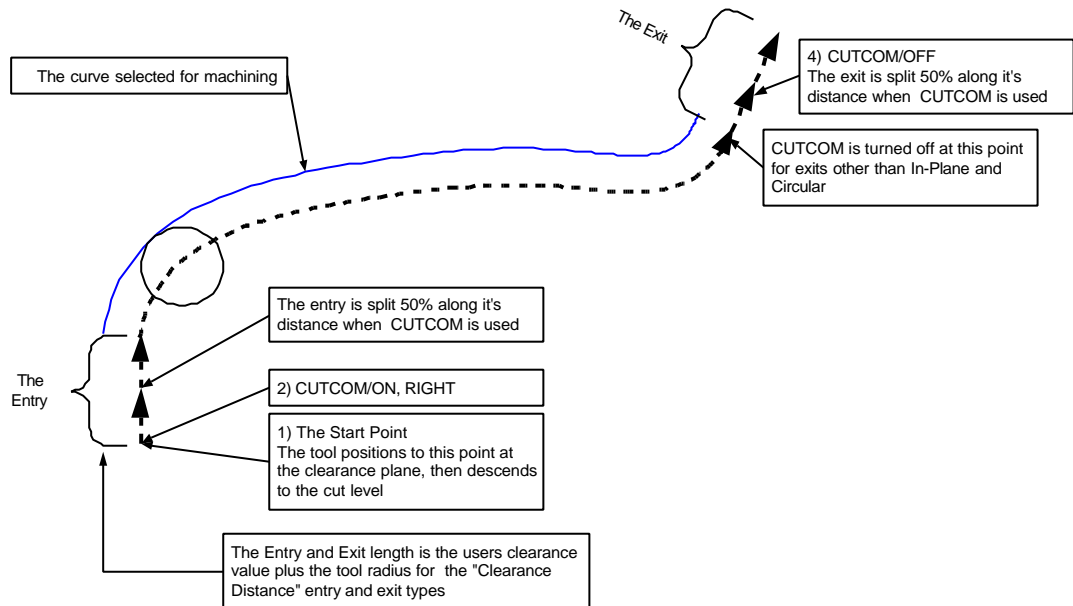
CUTCOM/OFF

This form turns CUTCOM off.

Usage

CUTCOM commands can be generated for the Profile and Manual milling operations with the "In-Plane" and "Circular" entry and exit types. CUTCOM commands can be added to other operations by editing the toolpath.

CUTCOM for In-Plane Entries

The Exit

The curve selected for machining

4) CUTCOM/OFF
The exit is split 50% along it's distance when CUTCOM is used

CUTCOM is turned off at this point for exits other than In-Plane and Circular

The entry is split 50% along it's distance when CUTCOM is used

2) CUTCOM/ON, RIGHT

1) The Start Point
The tool positions to this point at the clearance plane, then descends to the cut level

The Entry

The Entry and Exit length is the users clearance value plus the tool radius for the "Clearance Distance" entry and exit types

CUTCOM for Circular Entries

The Exit

The curve selected for machining

4) CUTCOM/OFF

CUTCOM is turned off at this point
for exits other than In-Plane and
Circular

2) CUTCOM/ON, RIGHT

*The
Entry*

1) The Start Point
The tool positions to this point at
the clearance plane, then descends
to the cut level
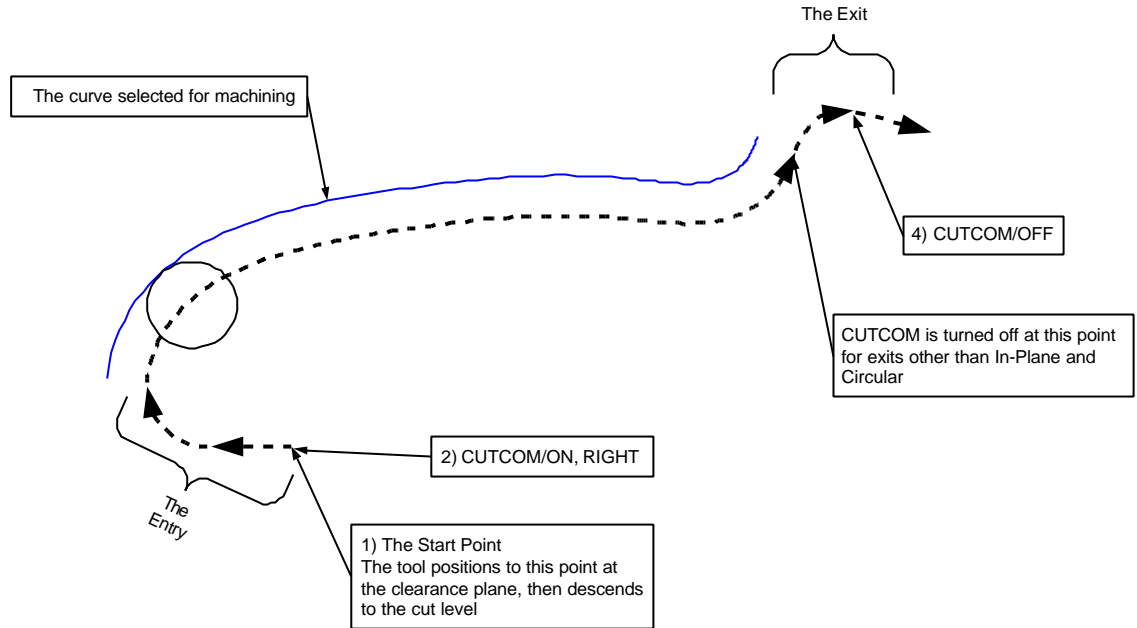
1.3.5. DELAY

This statement is used to insert a temporary delay (dwell) in the program operation.

DELAY/ [s | REV, r]

Variable	Type	Description
s	Real	Dwell time in seconds
r	Real	Spindle Revolutions of dwell

Usage

The minor word REV is only used if the dwell is specified in spindle revolutions instead of
seconds.

1.3.6. END

This statement identifies the end of the NC program and is the second to the last statement
in the CLDATA file It causes the program end code to be output to the tape file.

END

1.3.7. FEDRAT

This statement is used to establish a feedrate for the part program. This feedrate remains in
effect until a subsequent FEDRAT statement is programmed.

FEDRAT/ f, [IPR | IPM | MMPR | MMPM]

Variable	Type	Description
f	Real	Feedrate

1.3.8. FINI

This is the last statement in the CLDATA file and indicates to the postprocessor that the end of the file has been reached.

FINI

1.3.9. HEAD

The HEAD statement is used to specify the current operation head, or turret.

HEAD/ [1 | 2]

Word	Description
1	Specifies the primary turret.
2	Specifies the secondary turret.

Usage

The HEAD statement is output before the tool change (TURRET) to identify which turret is being used for the Lathe and Mill/Turn machine types.

1.3.10. INSERT

This statement is used to insert data directly into the machine program file without postprocessor interpretation.

INSERT/'text'

Variable	Type	Description
text	text	Up to 72 alphanumeric characters, including (. ,) / - + * = and blanks.

Usage

Note that in this format single quotes are required around the text.

1.3.11. LINTOL

The LINTOL statement is used for initiating a sequence which tests for non-linearity errors and corrects them in accordance with a given Linear tolerance.

LINTOL/ t

Variable	Type	Description
t	Real	Linear tolerance

This format of the command specifies the linear tolerance values and starts LINTOL processing.

LINTOL/ [ON | OFF]

Word	Description
ON	Starts LINTOL processing using the previously stated values.
Off	Terminates LINTOL processing.

Usage

LINTOL/ t specifies the linear tolerance and starts LINTOL processing.
LINTOL is specified when performing Mill/Turn operations when the tool axis is parallel to the Z axis of the lathe.
LINTOL/ON is not used by Generative Machining (The form LINTOL/t is always output).

Generative Machining outputs LINTOL for milling operations on MILL/TURN machine tools, the 5-Axis projection operation, and the 5 axis holemaking operation. The value in the LINTOL statement is:

(Operation INTOL + Operation OUTOL) / 2

the lintol value is set in each operation and based on that operations average cut tolerance

1.3.12. LOADTL

This statement specifies the tool to be used in the following portion of the part program and supplies the parameters for the tool. In I-DEAS, LOADTL statement is followed by the CUTTER statement.

LOADTL / t , IN, p, LENGTH, l , OSETNO, o

Variable	Type	Description
t	Integer	The tool number (Generative Machining "Post Tool ID").
p	Integer	The station number
l	Real	The tool gage length
o	Integer	The tool offset register number

1.3.13. MACHIN

This statement identifies the machine tool for which postprocessing is to occur. It also passes parameters to the post processor. All of the data after MACHIN/ is input by the user on the Machine definition form in Generative Machining.

MACHIN/text

Variable	Type	Description
text	text	User-supplied text.

Usage

The user-supplied text comes from the secondary machine form.
The MACHIN statement is output as the third statement in the CLDATA file. It is only output
if the toggle on the machine form is on.

1.3.14. MODE

The MODE statement is used to switch between milling and turning mode on Mill/Turn
machines.

MODE/ [MILL | TURN]

Word	Description
MILL	Selects Milling Mode
TURN	Selects Turning Mode

Usage

The MODE/MILL and MODE/TURN statements are output for all operations for all machine
types.

The MODE/MILL or MODE/TURN statement is the first statement output for an operation.

1.3.15. OFSTNO

The OFSTNO command is used to change the tool offset register number of the current tool
or to specify a fixture offset register number.

OFSTNO/o

Variable	Type	Description
o	Integer	The tool offset register number

OFSTNO/FIXTUR,o

Variable	Type	Description
o	Integer	The fixture offset number

Usage

Generative Machining only uses this statement to change the tool offset in turning finish
groove operations, where different offsets are available for the left and right side of the
groove. It can also be added via a Transiton move or a user CLDATA edit.

The fixture offset version of the command is output whenever the active fixture offset
changes.

1.3.16. OPSKIP

This statement causes the output of the "optional skip" code, indicating that all programming statement between ON and OFF can be ignored by the machine control at the operator's discretion.

OPSKIP / [ON | OFF]

1.3.17. OPSTOP

This statement causes the output of the "optional stop" code, indicating a location where the program will stop at the operator's discretion.

OPSTOP

1.3.18. PARTNO

This statement lists the part program identifier. Contains the name of the Setup from Generative Machining.

PARTNO/'text'

Variable	Type	Description
text	text	Up to 72 alphanumeric characters, including (. ,) / - + * = and blanks.

Usage

Note that this format of the PARTNO statement requires single quotes around the text. The PARTNO command is the first command in the CLDATA file.

1.3.19. PITCH

The PITCH statement is used to specify the parameters of the thread to be generated by subsequent THREAD moves. This statement must always be given prior to a THREAD statement and is modal until changed.

PITCH/ p {,MULTRD, m} {[, INCR, pf | , DECR, pf]}

Variable	Type	Description
p	Real	The number of threads per unit (inch or mm)
MULTRD	Minor Word	Specifies a multiple start thread
m	Integer	The number of thread starts
INCR	Minor Word	Specifies an increasing variable thread lead
DECR	Minor Word	Specifies a decreasing variable thread lead
pf	Real	The lead change of the thread in units per revolution.

Usage

The PITCH statement only needs to be output once for a threading operation (Note: in this context, "threading operation" refers to the entire set of motions needed to generate one complete thread feature).

The PITCH statement must be output once for each threading operation, even if the value is the same as that for a previous thread.

The only formats used by Generative Machining are PITCH/p and PITCH/p,MULTRD,m , where p is the pitch of the thread in the CLDATA file units and m is the number of starts for a multiple start thread.

1.3.20. PPFUN

PPFUN is never actually output by Generative Machining. For some third party postprocessors, the PPFUN statement is used to change internal post options.

There is confusion surrounding the Cldata statement and the edit function labeled "PPFUN" in Generative Machining. The edit function is used to permit the user to add a CLDATA line to the clfile. Anything that is typed into the text box on the Edit/PPFUN dialog is passed directly to the CLDATA file. For example, the if the user entered this statement into the toolpath via the PPFUN function:

FOFF/4

It would appear in the cl edit form as:

PPFUN/FOFF/4

It would be output in the CLDATA as the user entered it:

FOFF/4

If the user wishes to output a PPFUN command, he would actually enter a PPFUN command in the PPFUN edit dialog in Generative Machining. It would appear in the cl edit form like this:

PPFUN/PPFUN/1,2

Following the model above, the command would be output in the CLDATA file as:

PPFUN/1,2

1.3.21. PPRINT

This statement is used to print programmer comments.

PPRINT/'text'

Variable	Type	Description
text	text	Up to 72 alphanumeric characters, including (. ,) / - + * = and blanks.

Usage

Note that this format of the pprint statement requires single quotes around the text.

1.3.22. ROTABL

This statement indicates machine tool table rotation.

ROTABL/ a [,AAXIS | ,BAXIS | ,CAXIS]

Variable	Type	Description
a	Real	Absolute table position angle
AAXIS	Minor Word	Specifies the A rotary axis.
BAXIS	Minor Word	Specifies the B rotary axis.
CAXIS	Minor Word	Specifies the C rotary axis.

Usage

An axis identifier must be specified.
In Generative Machining, only positioning of rotary tables is supported.
ROTABL moves will be preceded by a RAPID command.

1.3.23. SPINDL

1.3.23.1. Mill Format

This statement specifies machine tool spindle parameters.

SPINDL/ s, RPM [, CLW | , CCLW] {, RANGE, r }

Variable	Type	Description
s	Real	Spindle speed in revolutions per minute
r	Integer	Spindle Range

This statement has an alternate form to turn the spindle on and off. The spindle parameters programmed in the previous SPINDL command are maintained:

SPINDL/ [ON, OFF]

Usage

Generative Machining does not output a spindle range for milling machines.
Generative Machining does not output the SPINDLE/ON form of the command, but it can be added by the user via Transition moves or the CLDATA editing function.

1.3.23.2. Turning Format

The following is the SPINDL format for turning operations to specify RPM mode spindle speeds:

SPINDL/ s, RPM [, CLW | , CCLW] , RANGE , n

Variable	Type	Description
s	Real	Spindle speed in revolutions per minute
n	Integer	Spindle range

The following is the SPINDL format for turning operations to specify constant surface speed mode spindle speeds:

SPINDL/ s [, SFM | , SMM] [, CLW | , CCLW] , MAXRPM, m ,RANGE, n

Variable	Type	Description
s	Real	Spindle speed in surface feet or surface meters per minute
m	Real	Maximum spindle RPM
n	Integer	Spindle Range

Usage

For constant surface speed mode Generative Machining outputs the spindle speed after the first motion command because some older machine tools require a X axis position to be supplied at the start of constant surface speed machining.

1.3.24. STOP

This statement causes the machine tool to stop.

STOP

1.3.25. THREAD

This statement causes a threading sequence to occur on the following linear move.

THREAD/ [TURN | FACE | TAPER]

Variable	Type	Description
TURN	Minor Word	Specifies a turning thread. The thread lead is specified along the turning axis. Tapered threads, such as API threads, are considered TURN threads because the lead of the thread is measured along the part axis, not along the taper angle.
FACE	Minor Word	Specifies a face thread. The lead is specified along the face, or perpendicular to the turning axis. This is used for scrolls and gasket seating surfaces on flange faces.
TAPER	Minor Word	Specifies a taper thread. The thread lead is specified along the

		thread angle.

Usage

Generative Machining supports onlyt the THEAD/TURN output.
The THREAD statement only modifies the next GOTO move, so it must be restated for each thread chasing pass.
Thread parameters are determined by the PITCH command, which is modal and only needs to be stated once.

1.3.26. TURRET

This statement indicates that a new tool is to be used in the following portion of the part program and gives the parameters of the new tool.

TURRET/ t, IN, p, OSETNO, o, SETOOL, glx, 0, glz [,SETANG, xy, ATANGL, zx]

Variable	Type	Description
t	Integer	The tool number (Generative Machining "Post Tool ID").
p	Integer	Tool station number
o	Integer	Tool offset register number
glx	Real	X axis gage length
glz	Real	Z axis gage length
xy	Real	Tool axis angle in the XY plane (always 0 for Generative Machining)
zx	Real	Tool axis angle in the ZX plane (determined by the operation coordinate system in Generative Machining)

An additional format to change the tool offset only is:

TURRET/t, OFFSTNO, o

Variable	Type	Description
o	Integer	Tool offset register number

Usage

The SETANG and ATANGL values are only specified for milling operations for Mill/Turn machines.
Generative Machining only supports tool angles in the ZX plane, so the xy angle value will always be 0.
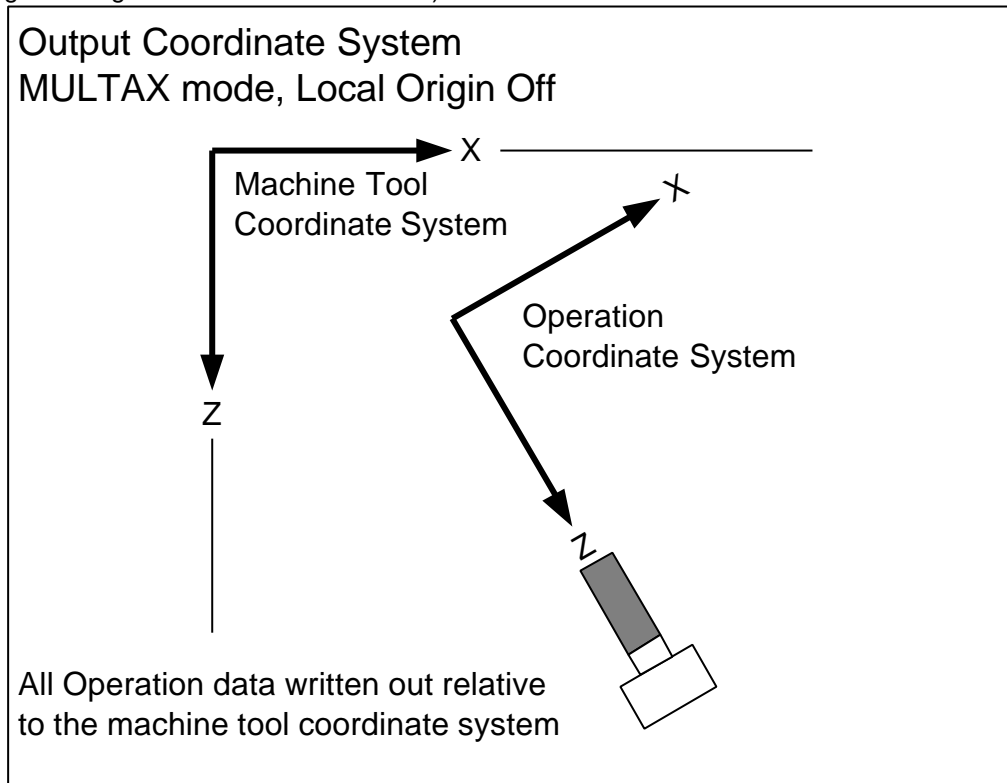
2. Coordinate Systems

2.1. Mill

2.1.1. MULTAX (IJK) OUTPUT

2.1.1.1. Local Origin Off

In this case, all CLDATA is written out in the Machine Tool Coordinate System (which is the global origin if a machine is not used).

Output Coordinate System
MULTAX mode, Local Origin Off

X

Machine Tool
Coordinate System

X

Operation
Coordinate System

Z

Z

All Operation data written out relative
to the machine tool coordinate system

2.1.1.2. Local Origin On

If the local origin is on, all CLDATA is written in the axis orientation of the Machine Tool Coordinate System, but relative to the origin location of the Operation Coordinate system. The Origin values produced must be transformed in the same way as the toolpath data itself if they are to be used.

**Output Coordinate System
MULTAX mode, Local Origin On**

Machine Tool Coordinate System

ORIGIN Z Value

ORIGIN X Value

Operation Coordinate System

CL Output Coordinate System when ORIGIN output is specified

The cutter location data for the operation will be output from the Operation Coordinate System origin, but in the axis orientation of the Machine Tool Coordinate System.

Note that if the ORIGIN values are added to the X, Y, and Z toolpath location values, the result would be the same as with the local ORIGIN output turned off.
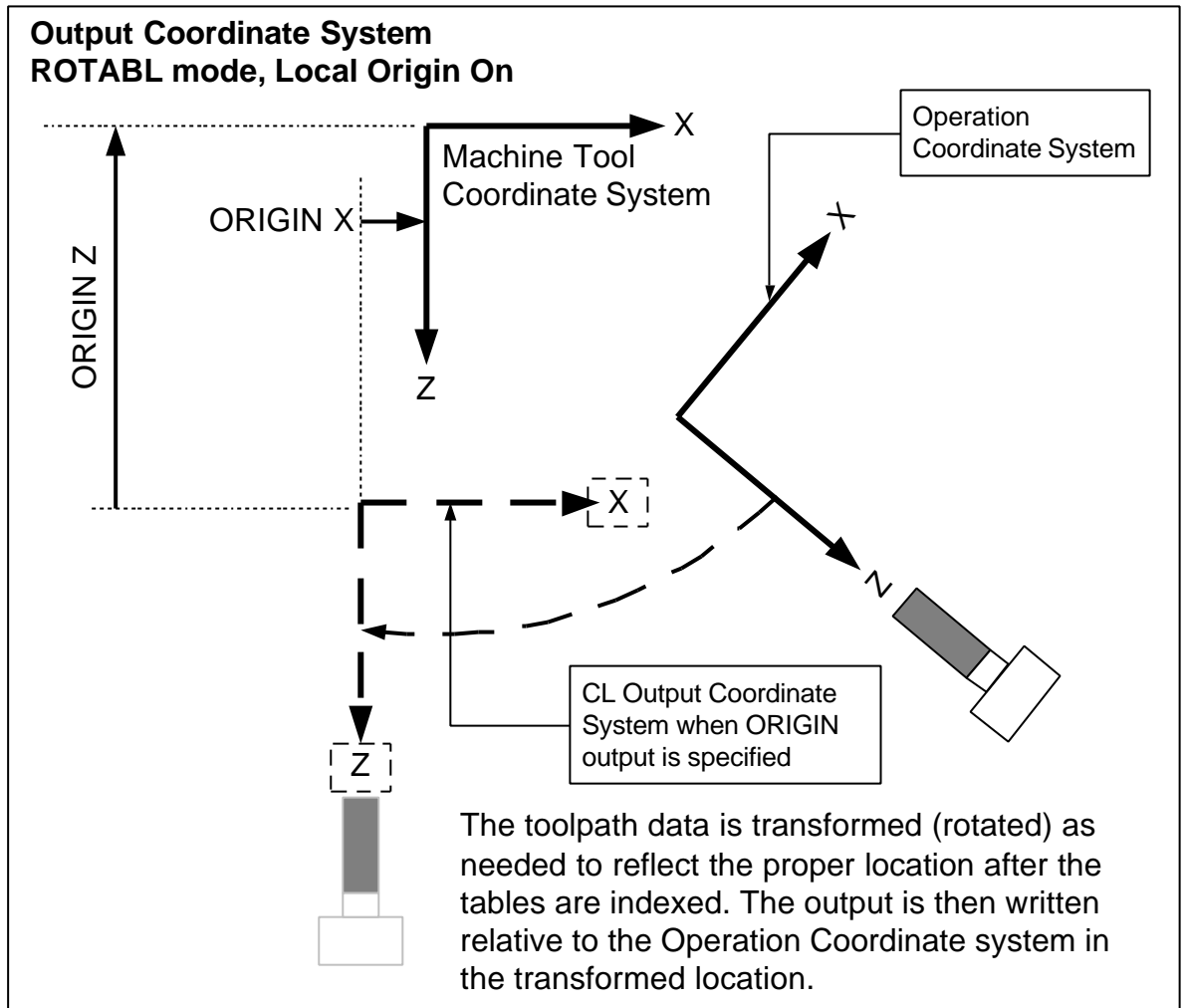
2.1.2. ROTABL Output

The control of rotary axis motion via ROTABL commands is available for 4 and 5 axis milling machines rotary tables and mill/turn machines only. ROTABL output is not supported for machines with rotary heads. For these machines the toolpath data is always output relative to the machine coordinate system.

2.1.2.1. Local Origin On

The Local Origin Switch is always on when using ROTABL output, the toolpath output is transformed as needed so that the Operation Z-axis and the Machine Z-axis are parallel. The toolpath data is output relative to the operation coordinate system in the transformed position.

Output Coordinate System
ROTABL mode, Local Origin On

The toolpath data is transformed (rotated) as needed to reflect the proper location after the tables are indexed. The output is then written relative to the Operation Coordinate system in the transformed location.
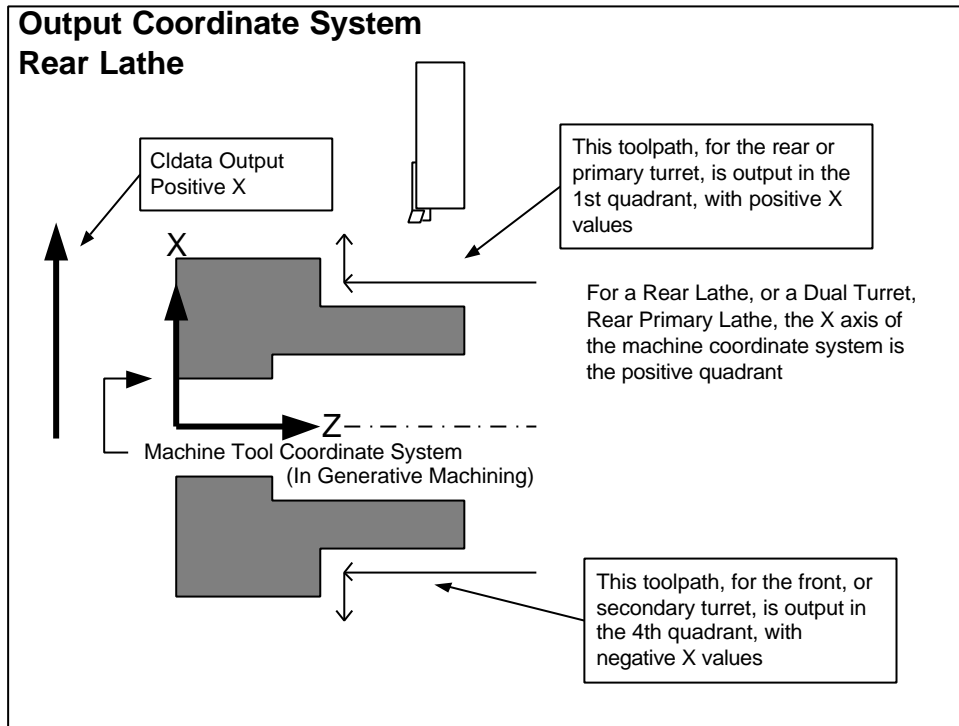
2.2. Lathe

Cldata for Lathes and Mill/Turns is always output relative to the machine coordinate system. There is no support for Fixture Offsets and localized toolpath data.
The coordinate system for Lathes and Mill/Turns is defined as that of the machine tool, with the Z axis being the axis of spindle rotation. Turning toolpaths are output in the ZX plane.
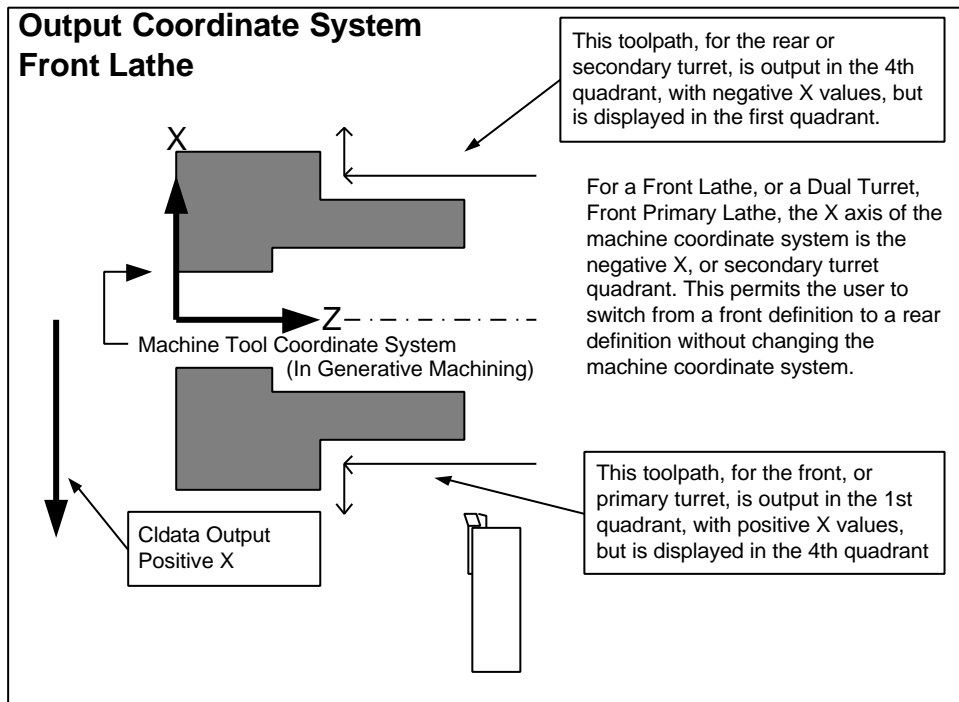
2.2.1. Rear

A "Rear" lathe is a machine for which the turret (or the primary turret for a dual turret machine) is "above" the part when the ZX plane is viewed from above (looking in the negative Y direction). I can also be thought of as a "1st quadrant" machine.

**Output Coordinate System
Rear Lathe**

Cldata Output
Positive X

X

This toolpath, for the rear or primary turret, is output in the 1st quadrant, with positive X values

For a Rear Lathe, or a Dual Turret, Rear Primary Lathe, the X axis of the machine coordinate system is the positive quadrant

Z

Machine Tool Coordinate System
(In Generative Machining)

This toolpath, for the front, or secondary turret, is output in the 4th quadrant, with negative X values

2.2.2.   Front

A "Front" lathe is a machine for which the turret (or the primary turret for a dual turret machine) is "below" the part when the ZX plane is viewed from above (looking in the negative Y direction). It can be thought of as a "4th" quadrant machine, however the X axis is really inverted, so the output in the lower quadrant is positive.

**Output Coordinate System
Front Lathe**

This toolpath, for the rear or secondary turret, is output in the 4th quadrant, with negative X values, but is displayed in the first quadrant.

X

For a Front Lathe, or a Dual Turret, Front Primary Lathe, the X axis of the machine coordinate system is the negative X, or secondary turret quadrant. This permits the user to switch from a front definition to a rear definition without changing the machine coordinate system.

Z

Machine Tool Coordinate System
(In Generative Machining)

This toolpath, for the front, or primary turret, is output in the 1st quadrant, with positive X values, but is displayed in the 4th quadrant

Cldata Output
Positive X

The real distinction between the treatment for a front and a rear lathe is primarily the display in Generative Machining. The output is always positive for the primary turret side of the centerline and negative for the side opposite of the primary side. It's possible to program a tool to move across the centerline, so positive or negative output can be generated for the primary or the secondary turret. It is also possible to program a "front" lathe in "rear" lathe mode. The output will be perfectly valid, but the view in I-DEAS will be inverted.

2.3. Mill/Turn

2.3.1. Turning Operations

The output for turning operations on a mill/turn machine follows all of the rules described for turning above.

2.3.2. Milling Operations
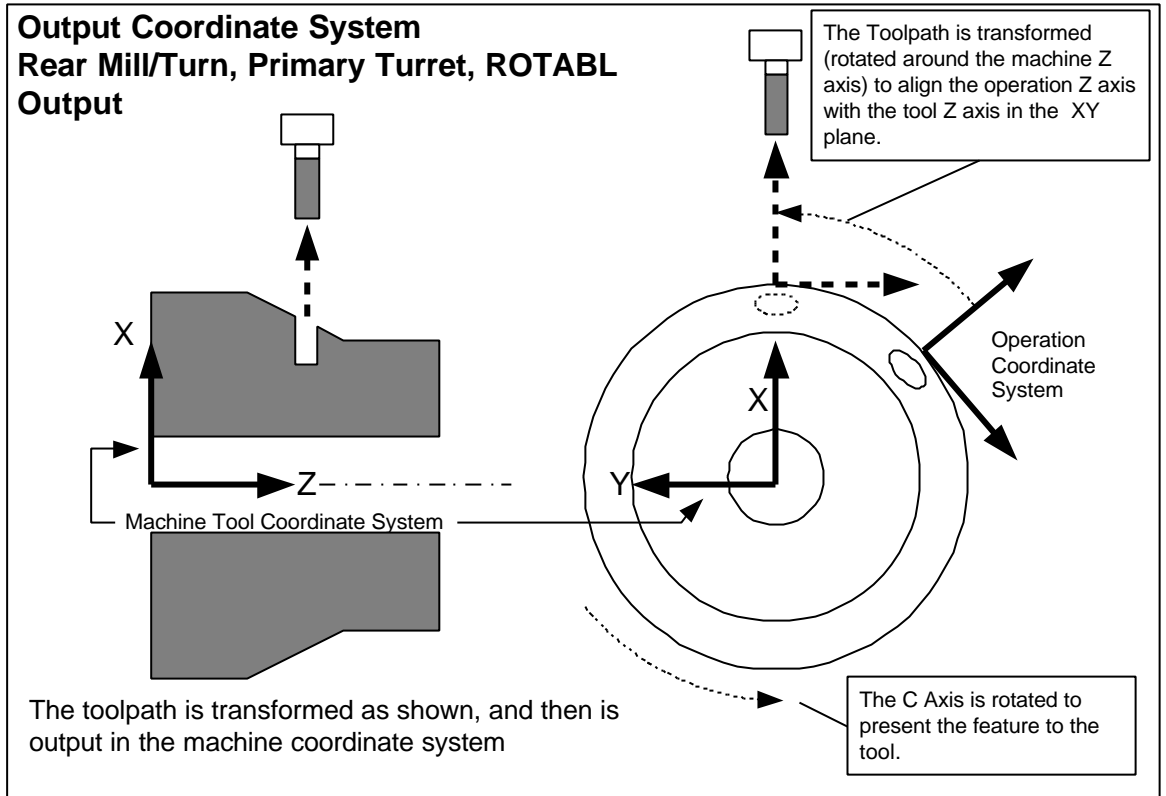
2.3.2.1. Multax (IJK) Output

Milling operations for a mill/turn machine are output in the lathe machine tool coordinate system. This is exactly the same as the Generative Machining Machine Coordinate System for a Rear lathe, and rotated 180 degrees around the Machine Coordinate System Z axis for a Front lathe (an inverted X axis).

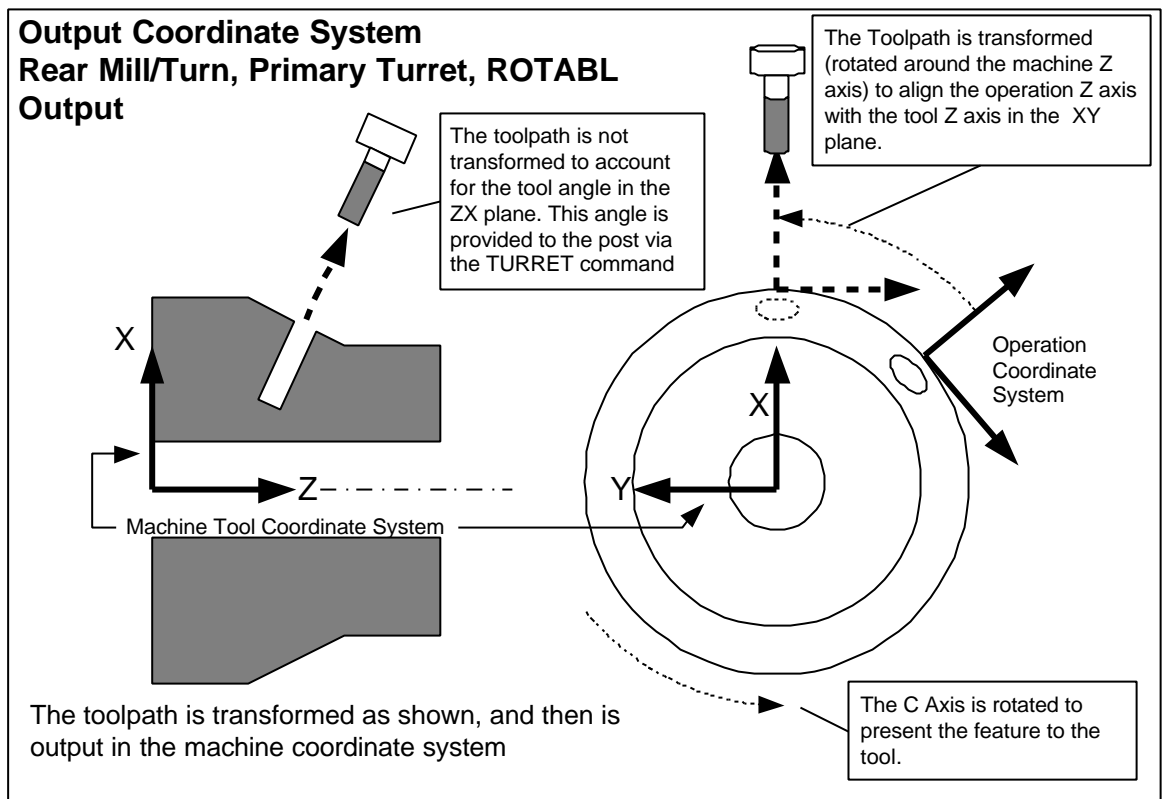#### 2.3.2.2. ROTABL Output, Tool axis of 0,0,1 (Front Face Mill/Drill)

In this case, the C axis is positioned to the 0-degree position (ROTABL/0) at the start of the operation, and the output is in the turning center X, Y, and Z axis. The postprocessor has to modify this output to be X, Z, and C axis rotation if the machine does not have a Y axis.

2.3.2.3. ROTABL Output, Tool axis other than 0,0,1 (Radial or Cross Mill/Drill)

If the tool axis of a milling operation is not 0,0,1, then a ROTABL command is output to index the C axis so that the operation coordinate system Z axis aligns with the turning center +X axis (primary turret) or -X axis (secondary turret). The toolpath is then transformed (rotated around the machine Z axis) to account for the rotary axis movement. This is the same as the milling case except the rotation is only done about Z.

**Output Coordinate System
Rear Mill/Turn, Primary Turret, ROTABL
Output**

The Toolpath is transformed (rotated around the machine Z axis) to align the operation Z axis with the tool Z axis in the XY plane.

X

Operation Coordinate System

X

Z

Y

Machine Tool Coordinate System

The toolpath is transformed as shown, and then is output in the machine coordinate system

The C Axis is rotated to present the feature to the tool.

If the tool is tilted in the ZX plane, it is assumed that the tool holder is fixed at this angle or that the turning center has a programmable rotary head. The CLDATA is not altered or transformed based on the ZX plane angle.

**Output Coordinate System
Rear Mill/Turn, Primary Turret, ROTABL
Output**

The toolpath is not transformed to account for the tool angle in the ZX plane. This angle is provided to the post via the TURRET command

The Toolpath is transformed (rotated around the machine Z axis) to align the operation Z axis with the tool Z axis in the XY plane.

X

Operation Coordinate System

X

Y

Z

Machine Tool Coordinate System

The toolpath is transformed as shown, and then is output in the machine coordinate system

The C Axis is rotated to present the feature to the tool.

3. Output Order and Modal Commands

The basic, standard, output order of CLDATA commands from Generative Machining is:

3.1. Beginning of the cldata file

The following commands are output at the beginning of the CLDATA file:

PARTNO Contains the name of the Setup
UNITS
PPRINT… Setup Note PPRINT statements (if any setup notes exist)

Note: This is output once per clfile. If more than one setup is selected for output, this data is output only once at the beginning.

3.2. For Each Operation

MODE The MODE statement is the first thing output for each operation, and is always output.
PPRINT… Operation Note PPRINT statements, if any operation notes exist.
HEAD The HEAD command is only output for multiple turret lathe and mill/turn machines.
FROM The FROM statement is output here if appropriate (see the FROM statement Usage Notes)
LOADTL Output if the tool changes (if any tool data changes). This could be the same tool with a new offset, for example.

or	
TURRET	
CUTTER	Output if the tool changes
PPRINT…	Tool Description PPRINT statements
MULTAX	Specifies Output Mode.
ROTABL	Output if the rotary table position or the tool changes in the ROTABL output modes
ORIGIN	Output if the Operation Coordinate System changes, the tool changes, or there is rotation.
OFSTNO	Output to specify the fixture offset if the fixture offset or the tool changes, if fixture offsets are being used.
LINTOL	Output for Multi-axis contouring operations and front face milling (tool axis 0,0,1) on mill/turns. Value is average of machining tolerance for the operation.
SPINDL	Output if the spindle values change or the tool changes
COOLNT	Output if a coolant mode other than off is used.
RAPID…	All motion for the operation
FEDRAT	Output during the motion of the operation. Not part of the Transition move
COOLNT/ OFF	Turn off coolant if coolant was turned at the beginning of the operation

3.3. End of the CLDATA file

The following commands are output at the end of the cldata file:

END
FINI

These command are output once at the end of the cldata file. If multiple setups are selected for output, these are only output at the end of the file.

3.4. Output Modality

3.4.1.   MODE

The MODE command is always output for all machine types.

3.4.2.   HEAD

The HEAD command is output whenever there is a tool change, whether the turret (head) changes or not. If there is no tool change, the HEAD command is suppressed. The HEAD command can be thought of as a functional part of the tool change sequence.

3.4.3.   Operation Description and Note PPRINT statements

These are output for every operation.

3.4.4.   FROM

For milling machines and lathes and mill/turns with a single turret, FROM is output only for the first operation of the CLDATA. FROM establishes the coordinates of the machine Home Point.

For dual turret lathes and mill/turns, FROM is output on the first operation to establish the home point for the first turret used, and a second FROM is output on the first operation using the other turret to establish the home point for that turret.

3.4.5. TURRET or LOADTL

If a different tool is specified(see the CLDATA vocabulary sections for LOADTL and TURRET), the toolchange will be output.

If no tool parameters change, the tool change output to CLDATA will be suppressed, with the exceptions noted below.

The tool change resets the modal behavior for all other operation commands. For example, if the tool changes but the spindle speed is the same, the spindle command is still output.

3.4.5.1. Threading Gage Pass

The tool change and spindle commands are restated after the program stop in a threading gage pass. This allows the offset portion of the tool change to be restated if needed, to ensure that any changes to the compensation value are activated. The spindle is turned off by a program stop, allowing the spindle command to be restated.

3.4.5.2. Finish Grooving

The finish grooving operation can change tool offsets in the middle of the operation. If the same tool is used in an operation following the finish groove operation, the tool change is restated to ensure that the "primary", or leading edge offset, is reinstated.

3.4.6. CUTTER

The cutter statement is output whenever a tool change is output. It is paramter APT standard format.

3.4.7. Tool Description PPRINT statements

The tool description PPRINT statements are output from Generative Machining whenever a tool change is output.

3.4.8. MULTAX

This statement is output whenever the tool changes or when the MULTAX mode changes.

3.4.9. ROTABL

The ROTABL statement(s) is output whenever the tool changes or when the table position changes.

3.4.10. ORIGIN

The ORIGIN statement is output whenever the tool changes or when the ORIGIN values change (different Operation Coordinate System).

3.4.11. OFSTNO

The OFSTNO statement is output whenever the tool changes or when the fixture offset changes (different Operation Coordinate System).

3.4.12. LINTOL

The LINTOL statement is output whenever the tool changes or when the LINTOL values change.
LINTOL is only output for multi-axis contouring or mill/turn front face (tool axis 0,0,1) work.

3.4.13. SPINDL

The SPINDL statement is output whenever the tool changes, or when the SPINDL values change (any spindle value).

3.4.14. COOLNT

The COOLNT statement is output for every operation that uses any mode of coolant (ON, FLOOD, MIST, TAP, THRU). If coolant is off for the operation, no COOLNT command is output. If coolant is turned on at the beginning of the operation, it is turned off at the end.