# CHAPTER 5

# PREPARATORY FUNCTIONS (G FUNCTIONS)

## 5.1 OVERVIEW

G codes consist of the address G plus a 1 to 3 digit number. The G codes are divided into two types, those that are effective only in the block in which it is specified (designated group 0 in the chart below) and modal commands that are effective until another G code in the same group is commanded.

| G CODE | GROUP | FUNCTION |
|---|---|---|
| 0 | 1 | Rapid traverse |
| 1 | 1 | Linear interpolation (feed) |
| 2 | 1 | Circular interpolation CW |
| 3 | 1 | Circular interpolation CCW |
| 4 | 0 | Dwell |
| 12 | 0 | Helical interpolation CW |
| 13 | 0 | Helical interpolation CCW |
| 17 | 2 | XY plane selection |
| 18 | 2 | ZX plane selection |
| 19 | 2 | YZ plane selection |
| 22 | 0 | Circular interpolation, fillet input CW |
| 23 | 0 | Circular interpolation, fillet input CCW |
| 30 | 3 | Mirror Image off |
| 31 | 3 | Mirror Image X on |
| 32 | 3 | Mirror Image Y on |
| 40 | 4 | Cutter diameter offset off |
| 41 | 4 | Cutter compensation left |
| 42 | 4 | Cutter compensation right |
| 44 | 5 | Cutter compensation, normal feedrate |
| 45 | 5 | Cutter compensation, modify feedrate |
| 70 | 6 | Input in inch |
| 71 | 6 | Input in millimeter |
| 72 | 7 | Transformation off |
| 73 | 7 | Transformation/Rotation, Scaling |
| 74 | 8 | Multi quadrant circle input off |
| 75 | 8 | Multi quadrant circle input on |
| 77 | 1 | Zig Zag Mill cycle |

| | | |
|---|---|---|
| 78 | 1 | Pocket Mill cycle |
| 79 | 1 | Bore Mill cycle |
| 80 | 1 | Drill cycle off |
| 81 | 1 | Z cycle, drill (feed in, rapid out) |
| 82 | 1 | Z cycle, spot face (feed in, rapid out) |
| 83 | 1 | Z cycle, deep hole (peck, rapid out) |
| 84 | 1 | Z cycle, tap (feed in, feed out) |
| 85 | 1 | Z cycle, bore (feed in, feed out) |
| 86 | 1 | Z cycle, bore (feed in, stop-wait, rapid out) |
| 87 | 1 | Z cycle, chip break (peck, rapid out) |
| 89 | 1 | Z cycle, bore (feed in, dwell, feed out) |
| 90 | 9 | Absolute programming |
| 91 | 9 | Incremental programming |
| 92 | 0 | Preset part programming zero point |
| 96 | 10 | Restore base part program coordinate system |
| 97 | 10 | Set work coordinate system |
| 99 | 0 | Deceleration override |
| 170 | 1 | Outside Frame Mill |
| 171 | 1 | Inside Frame Mill |
| 172 | 1 | Pocket Frame Mill |
| 173 | 1 | Outside Face Mill |
| 174 | 1 | Inside face Mill |
| 175 | 1 | Outside Circle Mill |
| 176 | 1 | Inside Circle Mill |
| 179 | 1 | Slot Mill |
| 181-189 | 1 | Z cycle (same as G81-G89) multi-hole |
| 191-199 | 1 | Z cycle (same as G81-G89) frame of holes |

The initial start up or clear settings of the preparatory functions are:

| G CODE | GROUP | FUNCTION |
|---|---|---|
| 0 | 1 | Rapid traverse |
| 17 | 2 | XY plane selection |
| 30 | 3 | Mirror Image off |
| 40 | 4 | Cutter compensation off |
| 45 | 5 | Cutter compensation, normal feedrate |
| 70 | 6 | Input in inch |
| 72 | 7 | Transformation off |
| 75 | 8 | Multi-quadrant circle input on |
| 90 | 9 | Absolute programming |
| 96 | 10 | Base coordinate system |

NOTE

A number of G codes can be specified in the same block. When more than one G code of the same group is specified, the last G code is effective.

## MISCELLANEOUS FUNCTIONS

## 5.2 PLANE SELECTION

This command selects a plane in which circular interpolation takes place.

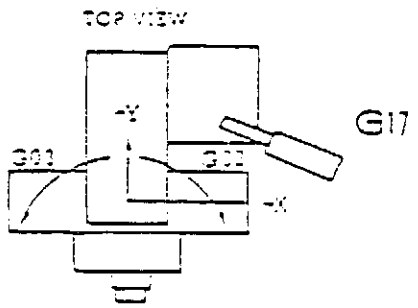
NOTE

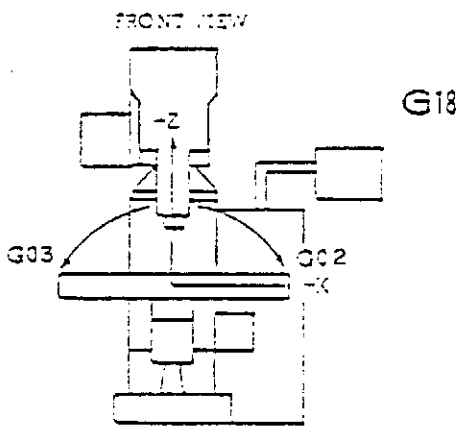
Circular interpolation data is input in multi-quadrant format.

RS-274D states in part:

"G02___ the path of the tool with respect to the workpiece is clockwise when viewing the plane of motion in the negative direction of the perpendicular axis."

G17.... XY plane
G18.... ZX plane
G19.... YZ plane



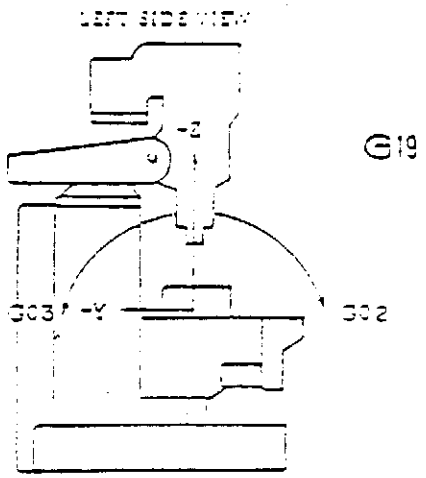
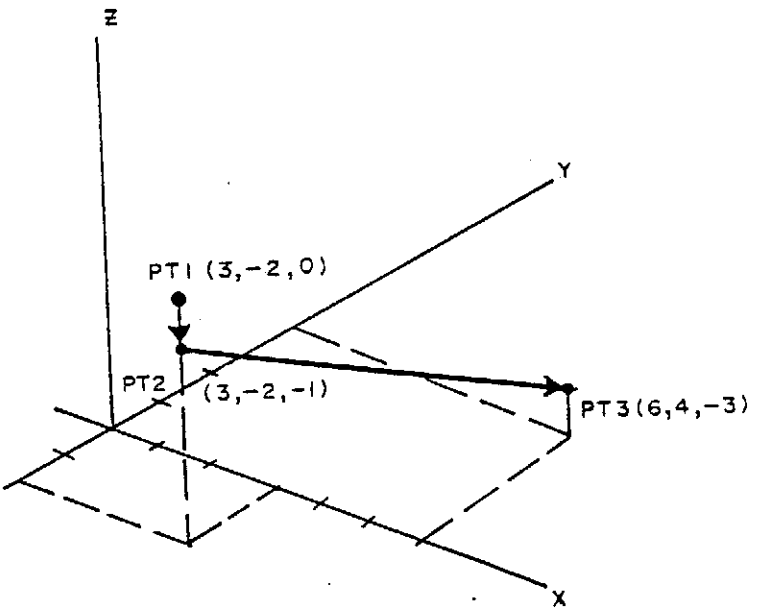




Figure 5-1: CW or CCW in Switchable Plane

EXAMPLE:

```
G90G0X3.Y-2.Z0;     PT1
G1Z-1.F10.;         PT2
X6.Y4.Z-3.F25.;     PT3
```




## 5.5 CIRCULAR INTERPOLATION (G2,G3)

When the control is in the Circular Interpolation mode (G2,G3), a circular arc in the selected plane can be generated by the coordinated motion of 2 axes. The format is:

X-Y plane (G17)

```
G2X___Y___I___J___F___; CW
G3X___Y___I___J___F___; CCW
```

Z-X Plane (G18)

```
G2X___Z___I___K___F___; CW
G3X___Z___I___K___F___; CCW
```

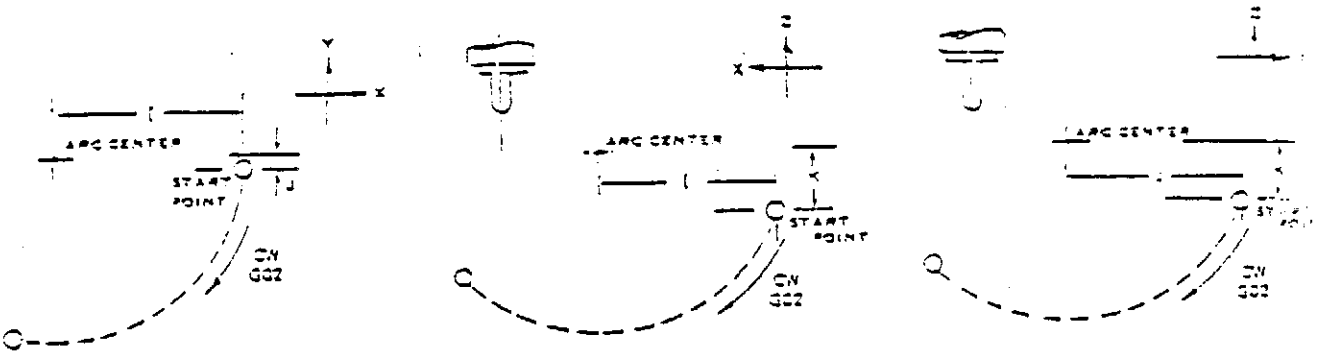
Y-Z plane (G19)

```
G2Y___Z___J___K___F___; CW
G3Y___Z___J___K___F___; CCW
```

NOTE

G18 or G19, if used, must be programmed in the block before the direction of circular interpolation.

The clockwise and counterclockwise directions for the XY, ZX, YZ planes are shown below:




G17 XY PLANE
PLANE VIEW

G18 XZ PLANE
VIEW FROM BACK
OF MACHINE

G19 YZ PLANE
VIEW FROM RHS

The block that calls for circular tool motion must specify both the arc endpoint (X___Y___Z) and the arc center (I___J___) or (I___K___) or (J___K___). If the control is in the Absolute (G90) Programming mode, the values of the axis position words must be the coordinates of the arc endpoint with respect to part program zero. If the control is in the Incremental (G91) Programming mode, the values of the words that define the arc endpoint must be the vector length from the start of the arc.

### 5.5.1 Circular Programming

If G74 is active, a block that defines a circular arc may define an arc in one quadrant of motion only. If G75 is active, a circular interpolation block can define a full circle of axis motion.

NOTE

The reset program condition for the BOSS 8 is G75, multi-quadrant input. The BOSS 4-7 Compatibility mode option sets G74, single quadrant input as the default condition.

The center of an arc must be defined by programmed arc center position words (I___,J___,K). These words specify the distance between the starting point of the arc and the center of the arc. I___ specifies this distance parallel to the X axis, J___ specifies this distance parallel to the Y axis, and K___ specifies this distance parallel to the Z axis.

If G74 is active, I___,J___,K___ specify the unsigned magnitude of the distance from the arc startpoint to the arc center in G90 (Absolute) or G91 (Incremental) mode.

If G75 is active and G90 (absolute data input) is also active, I___,J___,K___ will specify the arc center coordinates with respect to part program zero. If G91 (incremental data input) is active, I___,J___,K___ will specify the signed distance from the arc startpoint to the center of the arc.

NOTES

1. In circular interpolation I0 and J0 may be omitted except if G73 (transformation) is active.

2. To program an arc of 360 degrees (complete circle), either the X, Y or Z endpoint must be programmed together with the arc center.

### 5.5.2 Circular Interpolation By Radius Programming

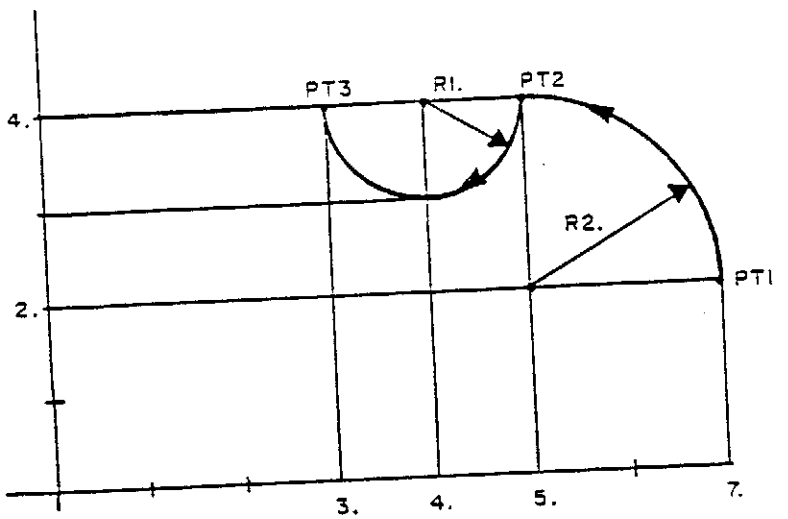
In the XY plane, circular interpolation can be programmed by specifying the radius (R___) instead of the arc center if the arc is 179.998 degrees or less. The format is:

```
G2X___Y___R___F___;
G3X___Y___R___F___;
```

Where X___Y___ are the arc endpoints as described above and R___ is the arc radius.

EXAMPLE:

```
G2V0;       Circle from previous endpoint, center at 0,0
G2X1.I.5;   Circle from X1., center at .5,0
```




G75G90 (Multi-quadrant absolute)

```
(1) G0X7.Y2.Z0;                    PT1
    G3X5.Y4.I5.J2.F15.;            PT2
    G2X3.I4.J4.;                   PT3

(2) G0X7.Y2.Z0;                    PT1
    G3X5.Y4.R2.F15.;               PT2 USING RADIUS
    G2X3.I4.J4.;                   PT3 180 DEGR ARC
```

G75G91 (Multi-quadrant incremental)

```
(1) G0X7.Y2.Z0;
    G91G3X-2.Y2.I-2.J0F15.;        PT2
    G2X-2.I-1.J0;                  PT3

(2) G0X7.Y2.Z0;
    G91G3X-2.Y2.R2.F15.;           PT2 USING RADIUS
    G2X-2.I-1.J0;                  PT3
```

## 5.6 HELICAL INTERPOLATION (G12,G13)

Circular interpolation in the XY plane can be synchronized with linear interpolation in the Z axis. The format is:

```
G0(G1)R___I___J___A___;    MOVE TO HELIX START PT
G12(G13)A___Z___F___;      DO HELIX
```

Where A___ is the total number of degrees of helical travel and Z___ is the absolute depth of travel.

NOTES

1. The resolution of A___ for helical is one degree. If A = 180.5 degrees it must be either rounded up or truncated to a whole number of degrees, for example A181.

2. Cutter compensation cannot be used with helical interpolation.

3. The range of A is from 1. to 65535. degrees.

   For example, a thread with 2 1/2 turns would be programmed as (360.*2.5=800. degrees) A800.

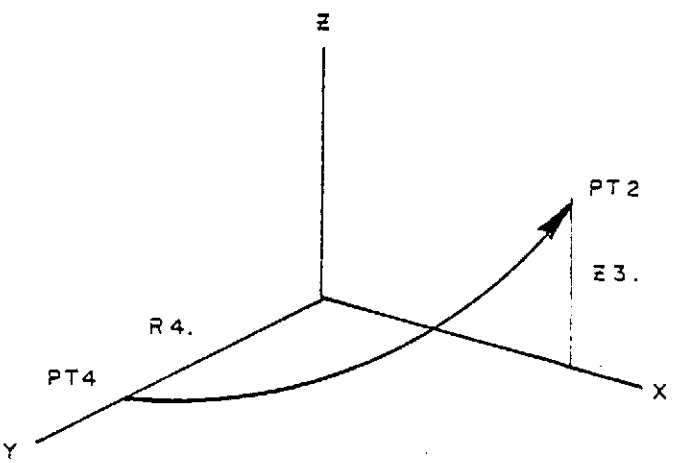
4. The range of R is from .1 to 6.5535 inches with the following feedrate restrictions:

   | | |
   |---|---|
   | .1 to .2 limit | 20. ipm max |
   | .2 to max | 40. ipm |

   G12 will generate a clockwise helix, G13 will generate a counterclockwise helix.

5. Helical interpolation cannot be transformed.

EXAMPLE:




```
G0R4.I0J0A270.Z0;  MOVE TO PT4
G12A90.Z3.F40.;    MOVE HELICAL TO PT2
```

## 5.7 FILLET PROGRAMMING (G22,G23)

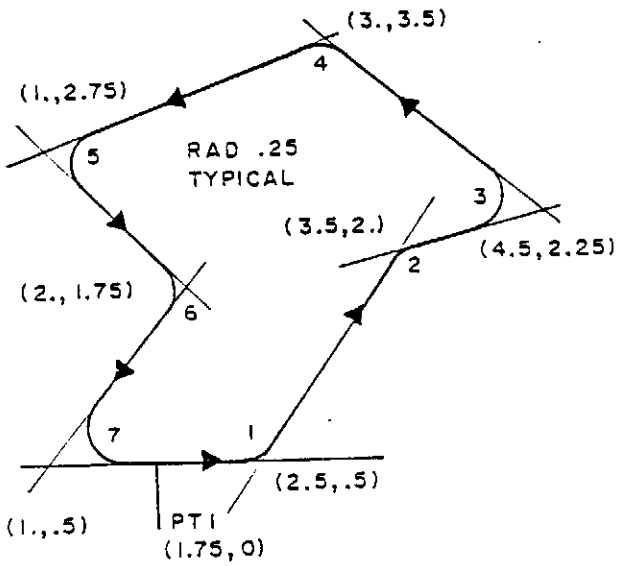
A fillet radius in the XY plane can be specified by the fillet radius together with the intersection of the extended line segments. The fillet arc will be blended tangent to the two lines. The format is:

G22(G23)X___Y___R___F___;

NOTE

Successive lines and fillets may be used. However, the block following the last G22 (G23) must be a linear (G1) move with X and Y coordinates.

EXAMPLE:




```
N5G0X1.75Y0Z0;                  MOVE TO PT1
N10G1X1.75Y.5F20.;              FEED TO WORK
N15G23X2.5Y.5R.25;              DO FILLET 1
N20G22X3.5Y2.0R.25;             DO FILLET 2
N25G23X4.5Y2.25R.25;            DO FILLET 3
N30G23X3.0Y3.5R.25;             DO FILLET 4
N35G23X1.0Y2.75R.25;            DO FILLET 5
N40G22X2.Y1.75R.25;             DO FILLET 6
N45G23X1.Y.5R.25;               DO FILLET 7
N50G1X1.75Y.5;                  LINEAR TO EXIT PT
```

## 5.8 DWELL (G4)

Code G4 specifies and initiates a dwell. The format for specifying the dwell time is:

G4/___

Where /___ is the dwell time.

NOTE

The range of dwell time is .01 to 327.68 seconds. Trailing zeroes must be programmed in the dwell value. For example, a dwell of 10 seconds must be programmed as G4/10.00; trailing zeroes and decimal point required.

The dwell time is modal once it is specified. The dwell is initiated by:

- o At the end of the feed Z move in a G82 or G89 cycle.
- o When a G4 code is programmed in a block that does not call for axis motion.

A G4/___ code defines a dwell and does not cause a dwell to occur.

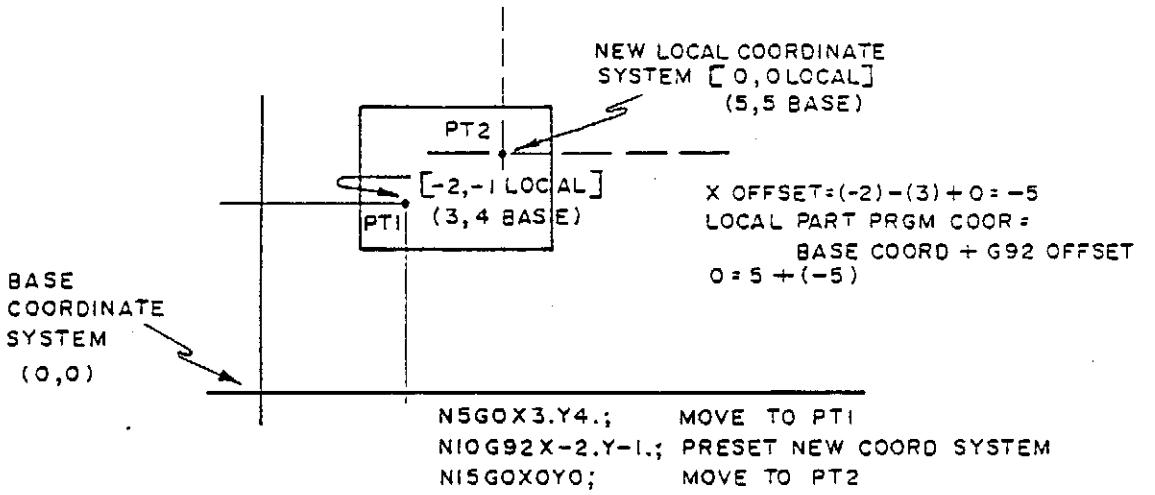
## 5.9 PROGRAMMING OF ABSOLUTE ZERO POINT (G92)

In SETUP, the SET command establishes the base part program coordinate system. The G92 code enables redefining the local part program zero by creating an offset value which is then summed with subsequent part program data. The format of the offset (G92) command is:

G92X___Y___Z___;

Where X___Y___Z___ are the new coordinate values for the current part program coordinate point. The G92 offset value is equal to:

New G92 value - current coordinate value + old G92 value.




## NOTES

1. No motion occurs as a result of a G92 block.

2. The LCD readout shows the value of the coordinates with respect to the local coordinate system.

3. Do not program G92 when cutter compensation is turned on.

4. It is recommended that the fixture offset command (G97) be used for translating the part program when multiple fixtures are used. G92 and G97 may be used together in a program. The G97 command establishes a fixture coordinate system and the G92 command establishes a local coordinate system within the fixture coordinate system.

5. The G92 offset remains in effect until one of the following operations is performed:

   - A new G92 offset is input.
   - RESET PROGRAM or REWIND (M2) will re-establish the base coordinate system.

## 5.10 INCH/METRIC CONVERSION (G70,G71)

Codes G70 and G71 specify whether data is to be input in either inch or metric.

| G CODE | UNIT SYSTEM | LEAST INPUT INCREMENT | FEEDRATE |
|---|---|---|---|
| 70 | Inch | .0001 | in/min |
| 71 | mm | .001 | mm/min |

### NOTES

1. The system startup condition will be G70 (inch input).

2. A block that contains either G70 or G71 cannot contain any other part program word.

3. The following data is stored in the controls as inch values. Input values in mm are connected to inch.

   Tool length offsets
   Tool diameter offsets
   Part program base coordinate system offset
   Feedrate (in/min)

4. The block following a change in the dimension system should redefine the X, Y, Z and (R, I, J, if used) values in the units of the new data input system (G70 or G71).