4.5 CIRCULAR INTERPOLATION PROGRAMMING

Circular interpolation moves the cutting tool along an arc from the starting point to an end point specified in the NC block. The curvature of motion is determined by the location of its center point (I,J, or K), which must also be specified in the same NC block.

*NOTE*: Circular interpolation normally works along the X,Y plane. Use G17, G18, or G19 to switch planes for circular interpolation.

The direction of rotation from the starting point determines the actual shape of the arc relative to the spindle axis. A G02 code moves the tool in a clockwise (CW) motion from the starting point; a G03 code moves the tool in a counterclockwise (CCW) motion from the starting point (see Figure 4.3).

![Figure 4.3: G02 (CW) and G03 (CCW) Cutting Paths](image)

A typical block of NC code using circular interpolation is shown below:

```
N10 G90 G03 X0 Y1 I0 J0 F2
```

The NC block shown above can be analyzed as follows:

- **N10** Block sequence number = 10
- **G90** Coordinates are given using absolute dimensioning
- **G03** The tool will proceed in a counterclockwise direction from the starting point to specified (X,Z) coordinates; centerpoint of arc is specified by (I,K) coordinates
- **X0** X-axis coordinate of end point = 0
- **Y1** Y-axis coordinate of end point = 1
I coordinate of centerpoint of arc = 0
J coordinate of centerpoint of arc = 0
Feed rate = 2 inches per minute

Figure 4.4 shows the tool path generated by the sample NC block with circular interpolation.

![Diagram of tool path with coordinates](image)

**Figure 4.4: Typical Tool Movement Using Circular Interpolation**

The equivalent movement can also be achieved with incremental dimensioning (G91) as shown below:

```
N10G91G03X-1Y1I-1J0F2
```

In this NC block, the X and Y values are the distance the tool is to move from its current position. The I and J values are the distance from the starting point of the centerpoint of the curvature of motion.

Advanced Users: To perform circular interpolation on another plane than the XY plane, a G18, or G19 code is used to select the XZ or YZ plane. This feature is rarely used in manual part programming but may be used by CAM systems to generate surfaces of revolution (e.g. a bottle mold). The G17 code is used to return to the XY plane for circular interpolation. An example NC block for circular interpolation on the XZ plane would be:

```
N10G90G18G03X0Z1I0K0F2
```
In this NC block, the X and Z values are the destination position of the tool. The I and K values are the location of the centerpoint of the curvature of motion. To determine the appropriate direction of motion generated by the G03, use the right-hand rule as explained in chapter 2 of this manual.

Advanced Users: Helical interpolation is performed when the axis not used in circular interpolation is commanded to move. For Example:

```
N10 G90 G03 X0 Y1 Z1 I0 J0 F2
```

would cause the Z axis to move at a constant feed to Z=1 while the X and Y axes are moving in a circular path, resulting in a helical motion. Helical interpolation works with circular motion on the XZ and YZ planes as well.

4.6 RAPID TRAVERSE PROGRAMMING

In the spectraLIGHT system, the rapid traverse code (G00) can move the tool at the maximum available feedrate to specified coordinates. Rapid traverse is used to reposition the tool in preparation for the next cut, or before ending a program. The tool should not be engaged in a cutting operation while being traversed to new location.

Rapid traverse can be used for all tool positioning motions resulting in reduced run time for the part program. The G00 code remains in effect until linear (G01) or circular (G02, G03) interpolation is again specified. Linear or circular interpolation will resume at the feed rate last specified prior to the rapid traverse motion(s) unless a new feed rate is specified.

Rapid traverse is not affected by the feedrate scale factor which can be varied under the spectraLIGHT Mill Control Program (Section 6). If all the positioning motions are done in rapid traverse (G00) and all the the cutting motions are done in linear (G01) or circular interpolation (G02, G03), cutting rates can be adjusted up or down using the feedrate scale factor without affecting rapid traverse motions.