Section 11: Coordinate Systems

Machine Coordinate System

The zero of the Machine Coordinate System (MCS) is the Cold Start Position. The Z axis cold start position is the tool change position. The coding of a G53 moves the machine to the programmed location in reference to the MCS. A G53 X0 Y0 always returns the machine to the MCS position. If a Tooling Coordinate System is not created, the CNC uses the MCS for the execution of G28, tool length offsets and the HO command. The tool change function (M6) automatically positions the Z axis to the zero of the MCS before executing the tool exchange.

Tooling Coordinate System - Home

The Tooling Coordinate System (TCS) is established as the X, Y, Z, A, B zero position. The SET(axis) or SETH command is used to establish this position (see Section 8). The Z axis is generally positioned at the cold start position for safety. After entering the SETH command, the axes positions are displayed on the video screen as the ZERO. Tool length offsets are measured relative to the TCS. The TCS allows the operator to adjust all offsets simultaneously. The operator may jog the Z axis up or down and perform a SETZ to adjust all offsets. Fixture offsets are relative to the TCS.

The axes are positioned to the zero of this coordinate system when the programmer issues a G28. The default Tooling Coordinate System is the MCS (Cold Start).

Enter the command: SETCS, and HO to switch from the TCS to the MCS.

Fixture Offsets

Fixture offsets are used to establish a part home position different from the TCS. This is most commonly used to locate the home positions for multiple part operations. This may also be used to establish a part change position, with the TCS, and a part home position, with the fixture offset. The fixture offset gives the user the capability to individually adjust each fixture. After the TCS is set during the job setup procedure, the locations (X, Y and Z) of up to 24 fixtures may be set in the Fixture Offset Table, each offset being relative to the TCS (Home). To enter a fixture offset, see Section 8, the FO command. The programmer references each offset, by use of the E word; E1 for offset #1; E2 for offset #2 etc. Fixture offsets 1-6 may also be referenced by programming a G54-G59; G54 for offset #1; G55 for offset #2 etc.

Programming a fixture offset in Format 1, without motion words, causes the axes to move to the current absolute position on the specified fixture. In programming Format 2, the move to the fixture is not made until an X, Y, or Z word is specified. The axis move can be specified in the same block or after the block containing the fixture offset code.

EXAMPLE: Fixture offset 1 = X1. Y1. Z1.
Fixture offset 2 = X2. Y2. Z2.

```
N1 O1
N2 G90 G0 X0 Y0 Z0
N3 E1 X0 Y0
N4 G0 X1. Y-1.
N5 E2 X2.5 Y2.5
N6 Z2.5
N7 E0 X0 Y0
N8 Z0
```

Format 1 results of running the example above:

N2.X0.Y0.Z0.
Block 2 the machine is already at X0 Y0 Z0 so no motion occurs.

N3.X1.Y1.Z1.E1.G0
Block 3 causes movement in all three axes because with format 1 the motion for the fixture offset is applied immediately for all axes.

N4.X1.Y-1.G0
Block 4 will move to the X and Y value relative to the fixture offset 1 that has already been applied for each axis.

N5.X2.5Y4.5Z1.E2.G0
Block 5 causes movement in all three axes because with format 1 the motion for the fixture offset is applied immediately for all axes.

N6.Z2.5G0
Block 6 will move to the Z value relative to the fixture offset 1 that has already been applied for each axis.

N7.X-4.5Y-4.5Z-2.G0
Block 7 will cancel the fixture offset currently in effect. Both format 1 and 2 cancel the same way and cause motion immediately for all axes.

N8.Z-2.5G0
Block 8 no fixture offsets are in effect, the Z moves as normal.

Format 2 results of running the example above:

N2.X0.Y0.Z0.
Block 2 the machine is already at X0 Y0 Z0 so no motion occurs.

N3.X1.Y1.E1.G0
Block 3 causes movement only in X and Y because with format 2 the motion for the fixture offset is applied only in moves that have the axes called out.

N4.X1.Y-1.E1.G0
Block 4 causes movement only in X and Y because with format 2 the motion for the fixture offset is applied only in moves that have the axes called out. Note that the Z axis fixture offset has yet to be applied and moved.

N5.X2.5Y4.5E2.G0
Block 5 causes movement only in X and Y because with format 2 the motion for the fixture offset is applied only in moves that have the axes called out.

N6.Z4.5E2.G0
Block 6 finally the Z causes movement because the block has a Z axis motion called out.

N7.X-4.5Y-4.5Z-2.G0
Block 7 will cancel the fixture offset currently in effect. Both format 1 and 2 cancel the same way and cause motion immediately for all axes.

N8.Z-2.5G0
Block 8 no fixture offsets are in effect, the Z moves as normal.

Program Coordinate System

The Program Coordinate System (PCS) is established by the programmer at some convenient point relative to the work piece and fixture. The default PCS is the Tooling Coordinate System. The PCS is set by the use of the G92 (Absolute Preset) or G52 (Coordinate System Shift) code in the part program. If the programmer knows the location of the Tooling Coordinate System, the operator can use G92 with the X, Y, Z, etc. dimension words to set the PCS. The G92 is used to establish the location of the part home from the current location. When using a G92, after a G28 return to home, the G92 should be canceled. This is accomplished by moving to the TCS zero and coding a G92 X0 Y0. The G52 is used to shift the current home position by a specified absolute amount.

EXAMPLE:

The material is 12" X 12", one inch thick. The Tooling Coordinate System (TCS) is set at the X minus Y plus corner of the material. The desired Program Coordinate System (PCS) is at the center of the material.

N1 G0 G90 S2000 M3 X0 Y0   Locate to the TCS
N2 H1 M7 Z.1
N3 G92 X-6. Y6. Establishes the PCS at the center
N4 X2. Y3.

Block #1 establishes that the current position is at the TCS.

Block #3 establishes the PCS at the center of the material. No machine motion occurs.

Block #4 moves the X axis to 2. from the PCS and 8. from the TCS (this is the same position). The Y axis is moved to 3. from the PCS and -3. from the TCS (this is the same position). Once the PCS is established, the programmer may switch between absolute(G90) and incremental(G91) modes without affecting the PCS. The G28 causes the machine to return to the TCS zero position.

EXAMPLE: The material is 12" X 12", one inch thick. The Tooling Coordinate System (TCS) is set at the X minus Y plus corner of the material. The desired Program Coordinate System (PCS) is at the center of the material.

N1 G0 G90 S2000 M3 X0 Y0 Locate to the TCS
N2 H1 M7 Z.1
N3 G52 X6. Y-6. Establishes the PCS at the center
N4 X2. Y3.

Block #1 establishes that the current position is at the TCS.

Block #3 establishes the PCS at the center of the material. No machine motion occurs.

Block #4 moves the X axis to 2. from the PCS and 8. from the TCS (this is the same position). The Y axis is moved to 3. from the PCS and -3. from the TCS (this is the same position). Once the PCS is established the programmer may switch between absolute (G90) and incremental (G91) modes without affecting the PCS. The G52 is canceled by coding a G52 X0 Y0. This returns the machine to the TCS on the next motion word line.