

- To “exit” Data Block 0, depress any control button except ENTER.
- Depress ENTER ADV to step through the following parameters:
 - X The X and Y offsets are the offsets at the part’s zero reference point relative to the permanent table zero reference point. These offsets can be entered or established with the TABLE ZERO pushbutton. Refer to section 3.5.2.
 - Y
 - CUR-TOOL The *current tool* parameter indicates which tool (by number) the machine “thinks” is in the spindle. Tool 0 indicates the absence of a tool. The operator sets this parameter to establish the number of the tool in the spindle.
 - O CONV 1 CLIMB Circle or frame milling operations are performed as either *conventional* (0) or *climb* (1).
 - Z RET The programmed Z *retract* parameter (different from hardware Z retract or calibration height) is the distance the tool retracts above the Z UP height after completing each machining operation and during rapid table positioning. If Z UP is negative, however, the Z Retract position will be referenced above the tool zero height (section 3.5.3).
 - TOOL # This parameter identifies up to 24 tools for programming DIA, RPM and TCAL. When a tool number is displayed on the bottom line of the CRT, any other tool can be selected by entering the desired number.
 - DIA Programming of tool *diameter* is required only for end mills and cutters used in cutter compensated milling. If required, enter actual measured cutter diameter.
 - RPM The spindle speed in RPM must be entered for each tool used. If no RPM is entered, an emergency stop condition results when an attempt is made to use the tool. Ensure that the correct spindle speed is selected for the tool, material, and application. Negative RPM dimensions can be programmed causing the spindle to rotate in the opposite direction (e.g. tapping left-hand threads). The operator must ensure that the gear selection coincides with the programmed speed, i.e. 60-500 rpm (low gear) 500-4000 rpm (high gear). The computer will automatically maintain the correct spindle direction if the rpm/gear rule is followed.
 - TCAL The zero reference point (on the Z axis) for a tool is established by jogging the Z axis to the desired position and depressing the TCAL pushbutton. An uncalibrated tool is indicated by “****”. Refer to section 3.5.3.

3.4.5 PROGRAMMING ELEMENTS

3.4.5.1 BASIC ELEMENTS

The *data block* contains the information used to describe the programmable machine operations. Basic elements X, Y and Z dimensions, feed rate, and tool number are defined first, followed by the data block format for each major machine operation.

- X The X and Y coordinates define a position on the table. Figure 3-2 illustrates the positive and negative directions for each of the three axes as viewed by a person facing the mill. Note the opposite relationship of the programmed motion of the tool relative to the table when actually the table moves relative to the stationary spindle. X and Y can be programmed as either *absolute* or *incremental* dimensions. Absolute dimensions are referenced from the part or fixture zero point determined by the table zero and offsets in Data Block 0. Incremental (INC) dimensions are referenced from the current table position. (Arc end points and repeat blocks are exceptions and are explained later.)
- Y

3.4.5.1 BASIC ELEMENTS - (continued)

The INC pushbutton is depressed to change the mode from absolute to incremental, and vice versa, *only* when the CRT is prompting for either the X or Y dimensions. The mode, which appears on the second line of the data block display, will change for the current block and all new blocks generated. As later described, only incremental mode is used for data blocks in a LOOP.

All dimensions in a data block are in inches or millimeters as indicated on the second line of the data block display. Dimensions are programmable to the nearest 0.001 in. or 0.01 mm. Leading and trailing zeros and decimal points do not have to be entered.

Z DN

Z Down is the programmed tool depth in drilling, boring, tapping, and milling operations. All Z dimensions are referenced from the tool zero point (as calibrated in Data Block 0) for the tool being used. As shown in figure 3-3, the positive Z direction is up from the table and the negative direction is down.

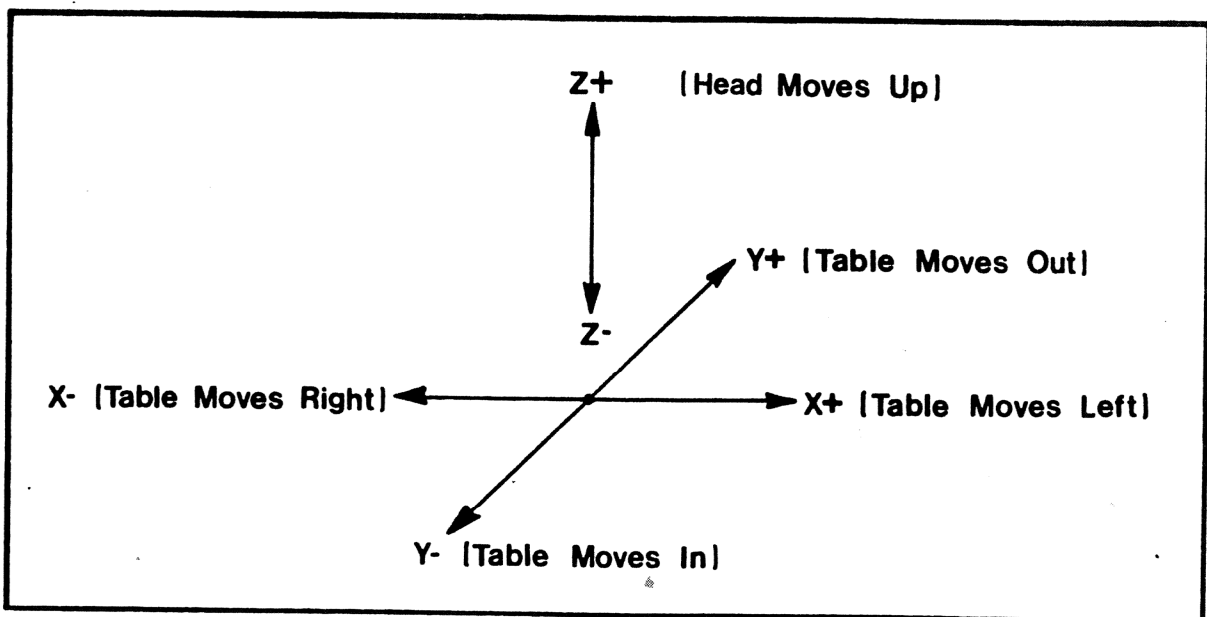


Figure 3-2. Relationship of X, Y, and Z Axes

FEEDR Z The Z feed rate is the spindle feed rate during drilling, boring, and tapping operations. The Z feed rate is programmable in inches per minute (0.1 to 250.0 ipm) or millimeters per minute (2 to 6350 mm per minute). However, in metric, the feed is only in increments of 2 mm per minute.

FEEDR XY The XY feed rate is the feed rate of the table with respect to the spindle, regardless of table direction during milling operations. The XY feed rate is programmable in inches per minute (0.1 to 100.0 ipm) or millimeters (2 to 2540 mm per minute). However, in metric, the feed is only in increments of 2 mm per minute.

TOOL # The tool number indicates which of the 24 possible tools is used for the programmed operation. The tool number must be the same in all data blocks of a milling or LOOP sequence; otherwise, an emergency stop condition occurs during operation. The data block display shows both tool number and diameter as stored in Data Block 0.

3.4.5.2 TYPES OF OPERATION

To program a data block, first select the type of block: standard, LOOP, REPEAT, STOP, or COPY. These special data blocks and data block qualifiers are described after the operation blocks are defined.

Next, select the type of operation, as detailed in the following sections. The data block type and subtype are displayed in the upper right corner of the CRT.

POS During a *position* operation, the spindle remains retracted and rotating, and the table moves at 250 ipm to the programmed X and Y dimensions. The parameters for a POS block are X, Y, INDEX, and TOOL #.

The INDEX parameter is used in conjunction with a rotary index table option. A signal is sent to a rotary fixture causing it to index to the next position. The mill assumes a MOTION HOLD state while the fixture is changing. Multiple position indexing can be accomplished with a POS block in a LOOP sequence. The INDEX parameter has no effect if the option is not installed.

DRILL Drill blocks are used for drilling holes or drilling down at the start of a milling sequence. In a drilling operation, the table positions to the X and Y dimensions, and then the spindle positions to the programmed Z retract height, both at 250 ipm. When the dimensions have been reached (X, Y), the spindle rapid traverses at 250 ipm to the Z UP dimension. The spindle then immediately initiates the programmed feed rate until it reaches the Z DN dimension. If a hole is being drilled, the spindle dwells for 0.8 seconds and then rapid retracts to the programmed Z retract height at 250 ipm. For a milling sequence, the spindle remains at the Z DN depth after drilling.

After choosing a drill operation, the operator can 1) select a bolt circle pattern (BOLT CIR), 2) add cutter compensation (CUT RT or CUT LT) to the milling sequence that follows, or 3) leave the block as a standard drill block. Cutter compensation is explained in the milling section (3.4.5.2). The parameters for a drill operation are X, Y, Z UP, Z DN, FEEDR Z, # PECK, and TOOL #.

Z UP The Z UP dimension is the height at which the spindle begins to feed at the programmed feed rate. This height is usually set slightly above the surface of the part to allow for spindle rapid travel to stop shortly before the tool engages into the work at the programmed feed rate.

PECK The pecking process is used to remove chips, reduce cutter load, and improve coolant distribution. In drilling holes with pecks, the tool goes only part of the distance to Z DN, rapidly retracts to Z UP, rapidly plunges back to the level at which it stopped, and proceeds at the programmed feed rate to a deeper level where it may repeat the process. In milling, the tool goes to a level above Z DN and runs the entire milling sequence at that level, rapidly retracts to programmed Z retract, repositions to the starting point, rapidly plunges to the just completed level, and drills down at the programmed feed rate to a deeper level where it may repeat the process.

3.4.5.2 TYPES OF OPERATION - continued

PECK (cont'd)

The number entered for the PECK parameter is the number of levels, including the final Z DN level, at which the pecking process occurs. The distance between Z UP and Z DN is divided by the number of pecks to determine the depth of these levels.

For example: If two pecks are programmed, the first peck is half the distance between Z UP and Z DN. The second peck goes to Z DN. A maximum of 40 pecks may be programmed.

BOLT CIR

The bolt circle routine automatically drills, bores or taps a series of holes equally spaced on a common circle. This routine requires two drill, bore or tap data blocks. The X and Y dimensions of the first data block (a standard drill, bore or tap block) defines the center of the bolt circle pattern. Other parameters describe the holes to be made. The second data block has the BOLT CIR option selected. This option will not be listed unless the previous data block is programmed with the same type of operation (drill, bore, tap). This block has parameters for the number of holes, (up to 250 can be programmed), the angle to the nearest 0.01 degree measured clockwise from the lower vertical centerline and the radius (RAD) of the circle. Refer to figure 3-3.

It is possible for the operator to skip over holes in the pattern during operation. Refer to SINGLE CYCLE operation in section 3.6.3.

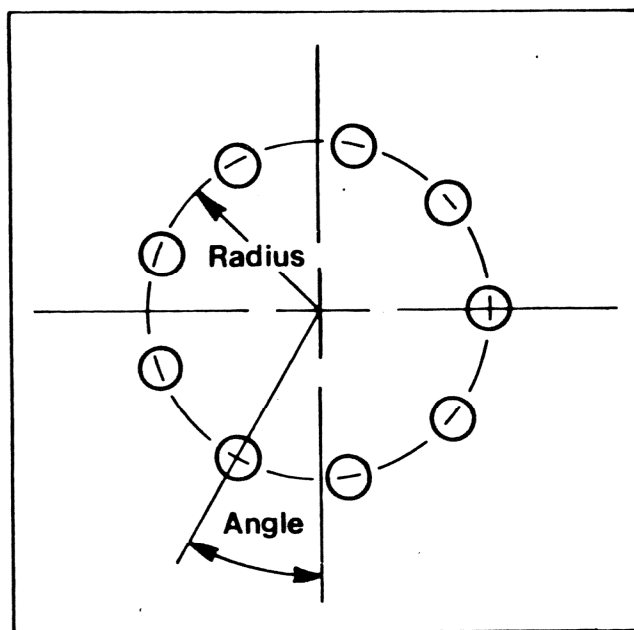


Figure 3-3. Bolt Circle Routine

3.4.5.2 TYPES OF OPERATION - continued

- BORE** A boring operation is similar to drilling a hole except the tool retracts out of the hole at the programmed feed rate until it reaches Z UP. No pecks are allowed. The bolt circle pattern is the only option for a boring operation.
- TAP** A tapping operation causes the spindle to follow the drill type of cycle to Z DN, then reverse direction and feed back up at the programmed feed rate until it reaches 0.3" above Z UP. The spindle starts rotating again, in the *original* direction. Match the feed rate with the spindle speed to achieve the desired threading. No pecks are allowed. A bolt circle is the only option for a tapping operation.
- MILL** Milling is defined as the cutting of metal by moving the table with respect to the tool. Every mill operation or sequence must be initiated with a drill data block to establish the depth of the tool (with the optional use of peck levels). Two groups of mill data blocks are available: one describes point to point line or arc segments; the other, special rectangular frame and circle routines. After choosing a mill data block, select the type of mill operation.

A sequence of line and arc mill blocks can be used to define a contour path which the tool will follow without retracting until the end of the sequence.

A *standard* mill operation (STD) is a straight line segment from the current position to the point specified in the data block.

Quarter circle clockwise or counterclockwise (QCIR C and QCIR CC) cuts a quarter circle from the current position to the point specified in the data block. The center point and radius are automatically calculated from the two points and directional sense. This feature can only be used when the two centerlines, the lines from the center to the start and end points, are parallel to the X and Y axes. (Figure 3-4)

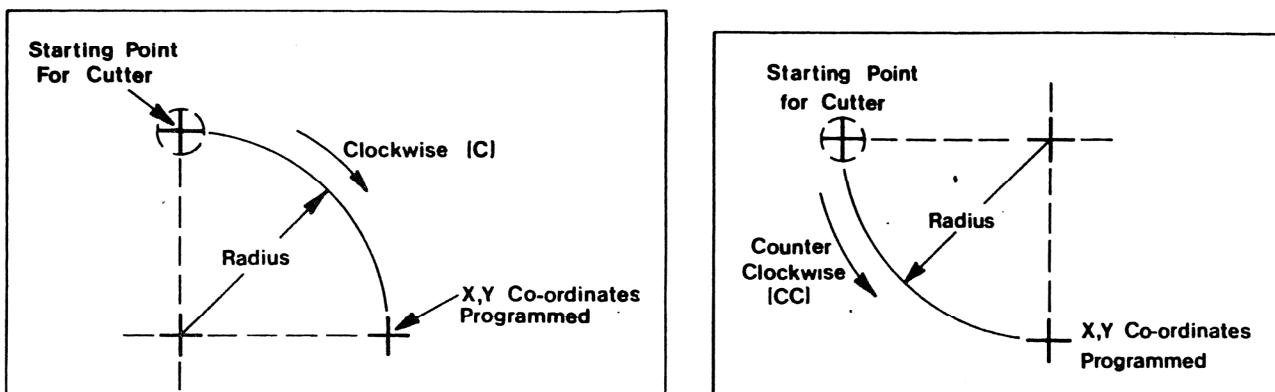


Figure 3-4. Quarter Circle Mill Routine (Clockwise and Counterclockwise)

3.4.5.2 TYPES OF OPERATION - continued

Arcs clockwise or counterclockwise (ARC C and ARC CC) cut circle segments as defined by two data blocks. The current position is the start point. The first arc data block specifies the center point of the circle and the direction of travel along the circle. This data block is labeled XY-CENT. The second data block containing the end point (XY-END) is automatically generated in ENTER mode after the first arc data block is programmed. In incremental mode, the end point is defined relative to the start point. (Figure 3-5)

NOTE: The start and center points define the radius of the arc. The distance between the center and end points must be equal to this radius. Otherwise, the cutter path will produce a divot.

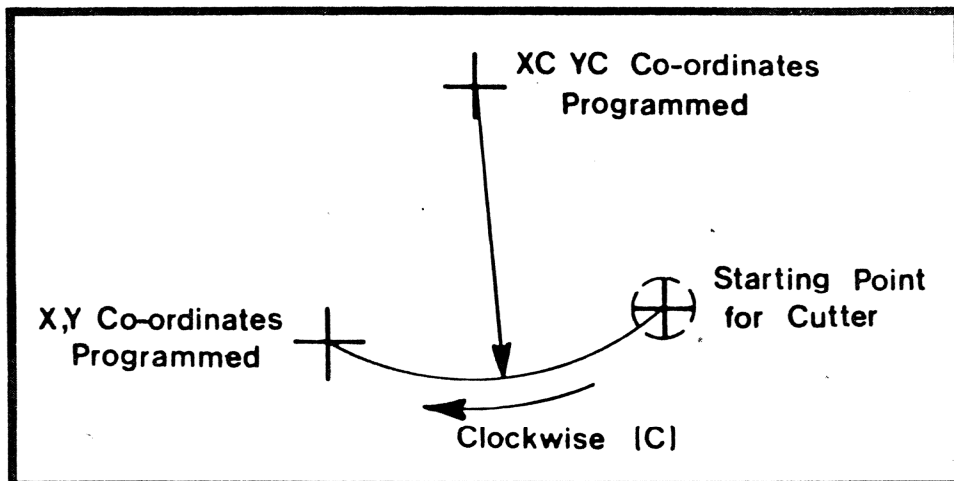


Figure 3-5. Arc Mill Routine

3.4.5.2 TYPES OF OPERATION - continued

Line and arc data block parameters are X, Y, Z DN, FEEDR XY, and TOOL #. If Z DN is different in one mill block than in the previous mill or drill block, the head feeds to the new Z DN before executing the mill block.

Cutter compensation can be selected in the initial drill block for line and arc milling. Without cutter compensation, the programmed contour is followed by the center point of the tool. With cutter compensation, the programmed contour is the contour of the finished surface, and the control automatically offsets the path of the tool by the tool's radius as entered in Data Block 0. Cutter right compensation places the tool on the right side of the contour and cutter left on the left side (visualizing a person standing at initial drill position and looking in the direction the tool moves).

LIMITATIONS IN CUTTER COMPENSATION

Some restrictions exist in the use of cutter compensation for line and arc milling. In the case of a line segment meeting another line segment and forming an angle of less than 20° , a tool cannot be programmed on the inside of the small angle. An emergency stop condition will occur during operation. (Figure 3-6)

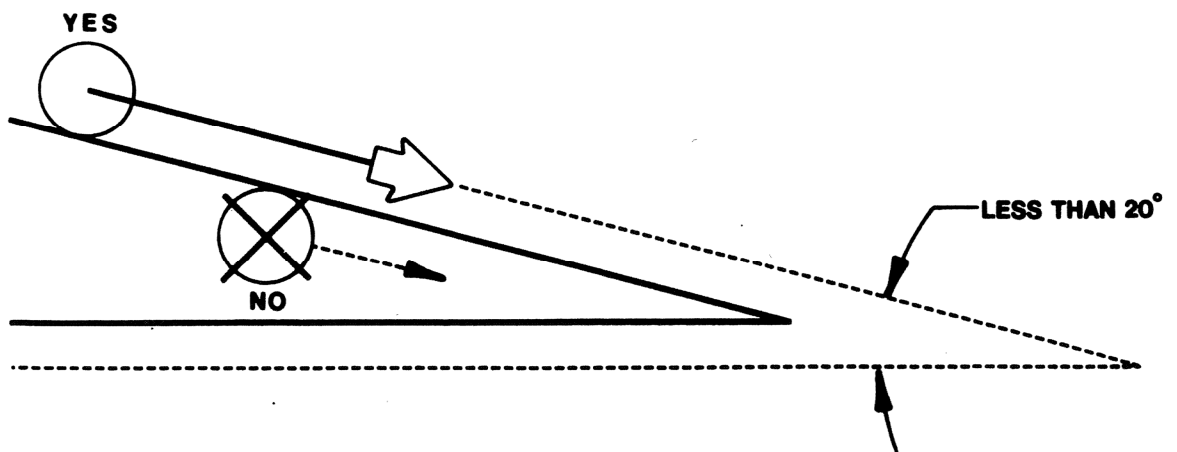


Figure 3-6. Cutter Compensation Limitation - Line to Line Segments

3.4.5.2 TYPES OF OPERATION - continued

Line to arc, arc to line, and arc to arc comprise another case. For this discussion, quarter circles will be considered arcs. If a segment blends into (i.e., meets tangentially) the next segment, there are no limitations. (Figure 3-7) If two segments do not blend, an angle of less than 180° is formed at the meeting point. If the tool is on the *outside* of this angle, there are no limitations. The cutter compensation control, however, is not designed to handle the tool on the *inside* of this angle. (Figure 3-8, 3-9) The control will execute a program containing one of these instances of inside compensation, but it will produce a divot in the part.

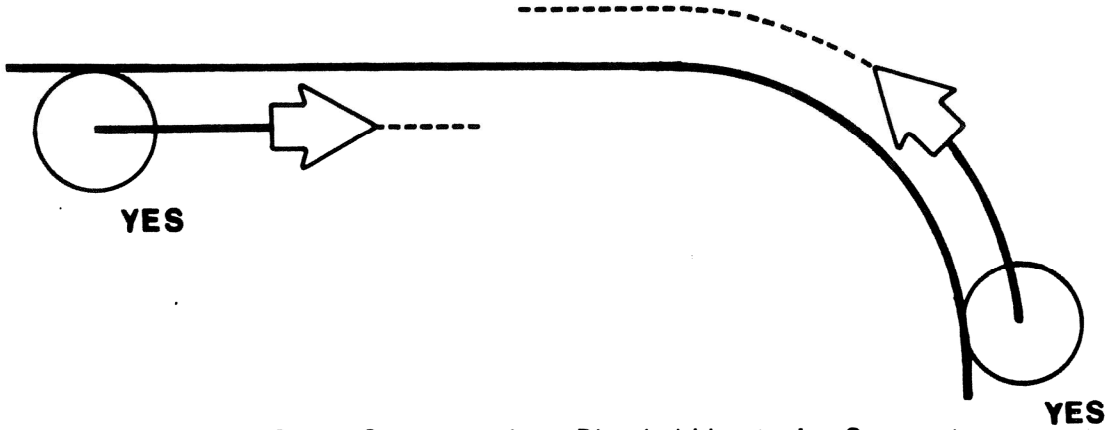


Figure 3-7. Cutter Compensation - Blended Line to Arc Segments

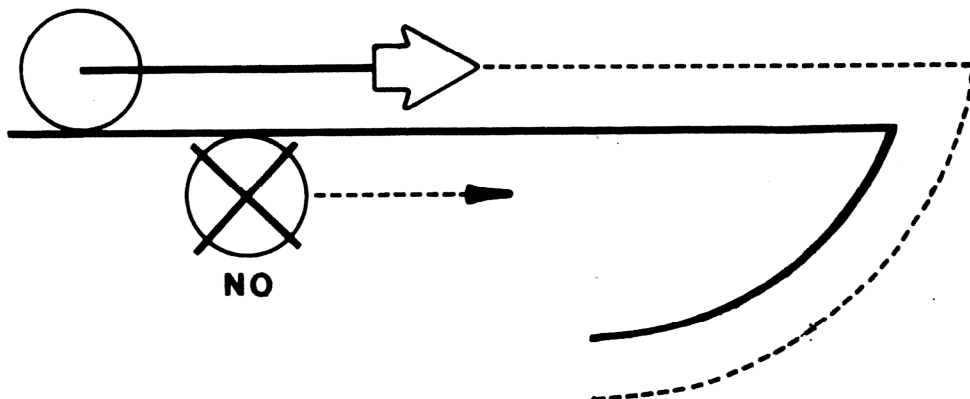


Figure 3-8. Cutter Compensation Limitation - Non-Blended Line to Arc Segments

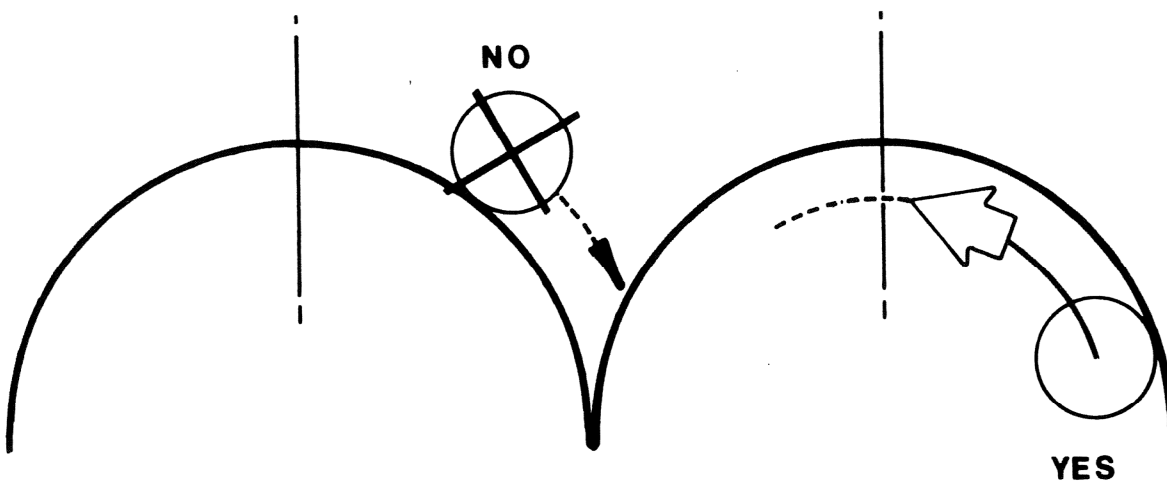


Figure 3-9. Cutter Compensation Limitation - Non-Blended Arc to Arc Segments

3.4.5.2 TYPES OF OPERATION - continued

Finally, the control does not look beyond the next segment. Therefore, the operator must ensure that the starting point and tool path prevent the tool from cutting into another contour of the part. Figure 3-10 shows a correct and an incorrect starting point on a part.

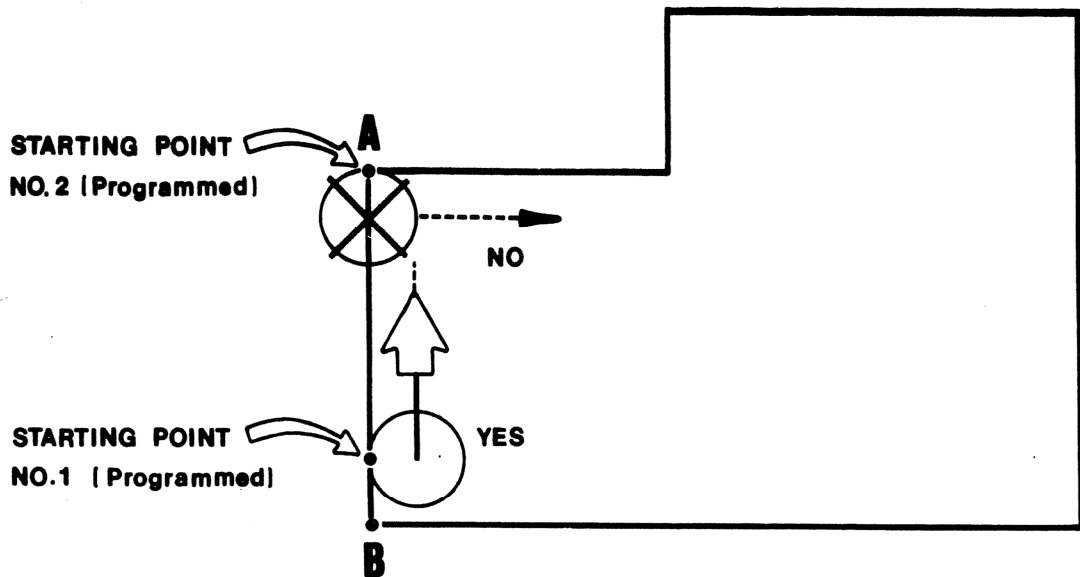


Figure 3-10. Cutter Compensation - Proper Starting Point

The second group of mill routines includes circles and rectangular frames with or without a two pass option (CIRCLE, FRAME, CIR 2P, FRM 2P). Cutter Compensation is automatically incorporated into these routines with the operator programming the finished surface contour.

First, select cutter placement. INSIDE positions the cutter on the inside of the programmed contour, while OUTSIDE places it on the outside. Both INSIDE and OUTSIDE initially plunge the tool 0.125" away from the finished surface, then move into the contour surface, and when finished, back the tool the same distance away before retracting. Refer to figures 3-11 and 3-12.

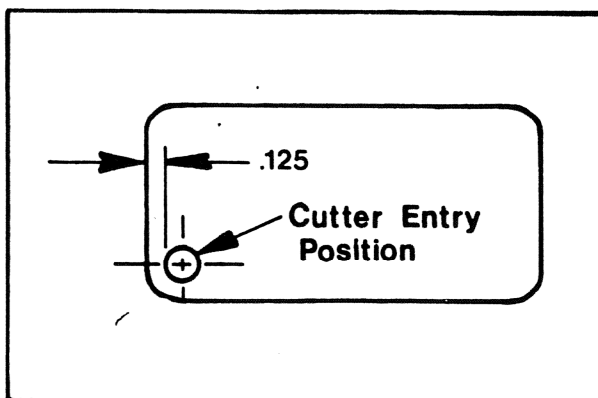


Figure 3-11. Inside Cutter Positioning Rectangular Frame

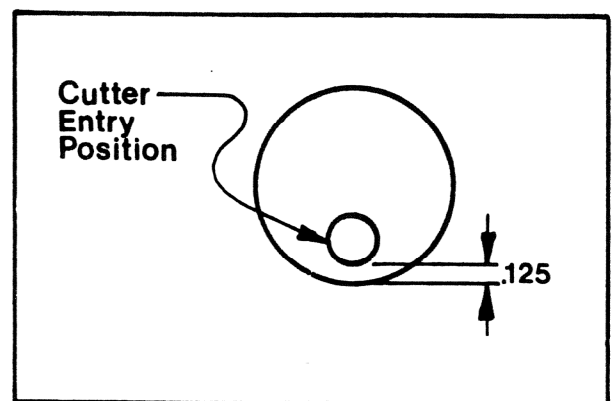


Figure 3-12. Inside Cutter Positioning Circle

3.4.5.2 TYPES OF OPERATION - continued

IN/TAN and OUT/TAN (inside/tangent and outside/tangent) perform in the same manner as IN-SIDE and OUTSIDE except the tool plunges and retracts tangent to the contour surface as opposed to 0.125" away. POCKET cleans all the metal out of the programmed contour down to the Z DN depth. POCKET starts like the IN/TAN option, but after finishing the pass along the contour, the tool moves toward the center and repeats a smaller version of the contour, overlapping the previous path by 0.050". This process is repeated, moving toward the center until the pocket is cleaned out. If pecks are selected, the entire clean-out process will be repeated at each peck level with the tool retracting before moving back to the start point.

The two-pass option (CIR 2P or FRM 2P) performs a roughing and finishing pass. The roughing pass is made 0.010" from the XY contour and 0.005" above the Z DN depth. After the roughing pass, the tool moves to the final dimensions and completes a finishing pass at the same feed rate. In a POCKET, the complete clean-out process is repeated during the finishing pass. If pecks are chosen, all are performed down to the roughing depth, prior to the finishing pass.

For rectangular frames, the initial drill block defines the lower left corner. The mill block contains XL and YW: the length along X and the width along Y (positive values only) respectively. A corner radius can be specified from 0 to a value which allows the corners to meet. Other parameters are FEED XY and TOOL #. Refer to figure 3-13.

In both circles and frames, the direction the tool travels around the contour is determined by whether conventional or climb milling is in effect.

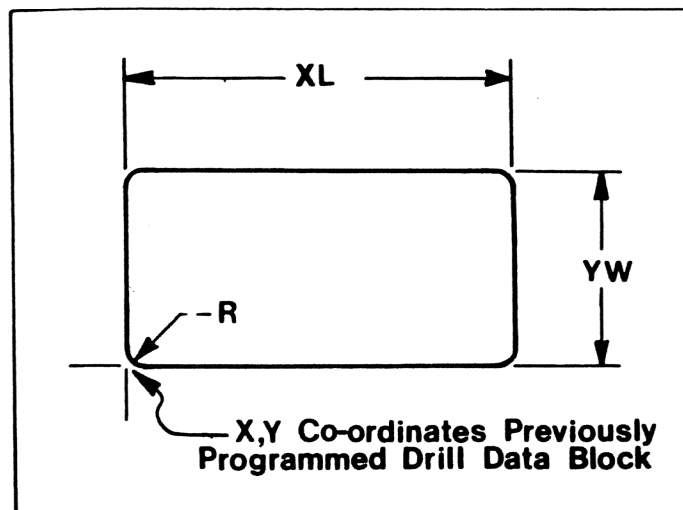


Figure 3-13. Rectangular Frame Routine Dimensions

3.4.5.3 LOOP, REPEAT AND STOP

The following special data blocks and data block qualifiers are selected as the first choice of options in all data blocks.

LOOP A LOOP data block with a matching repeat (REP) data block indicates both the start and end of a sequence of blocks which define a pattern to be repeated. The number of repeated patterns and the locations of the repeats are specified in the REP data block. A LOOP block is actually a POS, DRILL, BORE, or TAP block with a LOOP qualifier as displayed on the top line of the CRT. All data blocks in a LOOP sequence are automatically programmed in the incremental position mode; therefore, a LOOP sequence should be preceded by a data block that establishes an absolute starting position. A

3.4.5.3 LOOP, REPEAT AND STOP - (continued)

LOOP and REP sequence can be "nested" (i.e. entirely contained) within another LOOP and REP sequence. These sequences can be "nested" up to four deep. The lower right corner of the CRT shows how many LOOP sequences are "open" ($L = n$). Only one tool can be used during the entire LOOP sequence.

REP *Repeat*, a special data block containing the X and Y increment, is used to determine the starting location of each repeated pattern relative to the previous starting location. For each Y increment, the pattern is repeated at every X increment determined by the repeat counts. NX and NY are the number of times a pattern is repeated in both the X and Y directions after being performed once. Therefore, the total number of patterns completed is equal to $(NX + 1) \times (NY + 1)$. Figure 3-14 is programmed with an absolute POS block specifying the location of the first hole, a LOOP DRILL block, and a REP block. The repeated pattern is a single drilled hole.

The REP option is offered only among the list of special data block options when preceded by an unmatched LOOP block.

STOP Added usually to a program's last data block to cause the program to stop. After the data block is executed, the head retracts and the spindle shuts off.

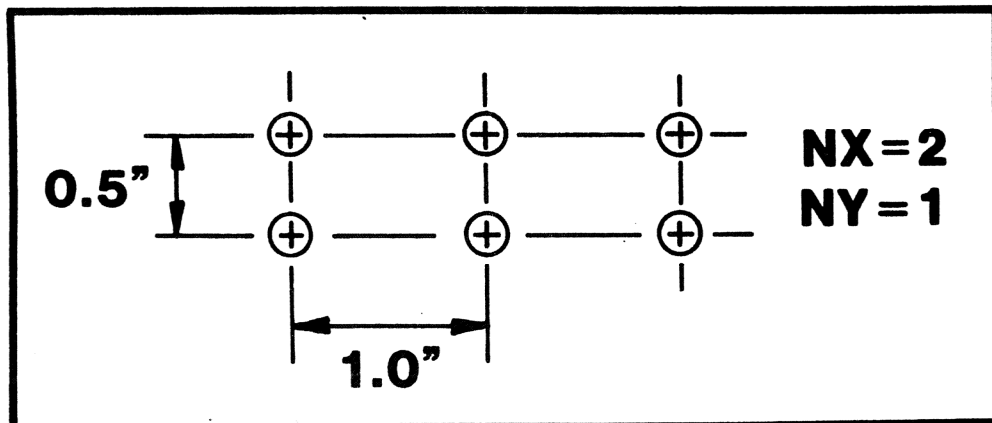


Figure 3-14. LOOP and REP Example

3.4.6 COPY

A special data block existing only in the ENTER mode generates new data blocks which are copies of an existing sequence of blocks. More importantly, the dimensions, parameters, and in some cases, the types of operation can be changed in creating new data blocks. This copying function makes it easier to program operations which are similar, but use a different tool or other parameter. After being programmed, a COPY will modify and copy forward from the specified range of data blocks to the new sequence, starting with the data block number of the COPY block. The COPY block is deleted after performing its function. A COPY block cannot be an *inserted* data block, i.e. a new data block between existing data blocks.

The control first "asks" for the number of the first block in the sequence to be copied (from data block) and then the number of the last block in the sequence (to data block).

3.4.6 COPY - continued

A parameter will be modified only if a new value is specified in the COPY block. The operator can choose to change all DRILL, BORE, and TAP blocks in the COPY range to either a DRILL, BORE or TAP operation. The number of PECKS in all DRILL blocks can then be changed.

Offsets (XOF and YOF) can be added to the X and Y dimensions in all absolute data blocks in the COPY range. The *first* block in the COPY range must be absolute or all blocks in the range must be incremental. If all the blocks are incremental, the Copy function will create an incremental POS block using the values of XOF and YOF for X and Y and place this block first in the new sequence of data blocks.

The Z UP, Z DN, FEEDR Z, FEEDR XY, and TOOL # can be changed for data blocks having these parameters. If a new parameter is given, then all blocks with this parameter will be changed.

The final choice in the COPY block is 1) to step through the COPY block again, (default choice), or 2) execute the COPY function (EXEC), or 3) CANCEL, (delete) the COPY function.

Dimensions and feed rates cannot be changed in a series of blocks when the INCH and METRIC measurements are mixed. The COPY block must use the same units as the last block.

The following errors may occur when executing a COPY block:

ERROR: COPY 1	FROM or TO block does not exist.
ERROR: COPY 2	INCH and METRIC blocks are mixed in the COPY range and COPY block.
ERROR: COPY 3	All blocks in the range are not in incremental mode; the first block is incremental.
ERROR: COPY 4	COPY is being selected in an inserted data block.

3.5 SET-UP

3.5.1 CALIBRATION OF AXES

The X, Y and Z axes must be calibrated after loading the MASTER, after a power failure, and before operation of a parts program. With the servos energized (Power ON and in OPERATE mode), press the TABLE ZERO pushbutton (button illuminates). The table travels to the right rear corner seeking a machine reference limit switch and a marker signal on each axis. When calibration is complete, the light goes out, the table moves out one inch in each direction, and the CRT will read 0.0" for X and Y. Press TOOL CAL button to calibrate the Z axis (button illuminates). The spindle moves up until it reaches the Z retract limit switch. The spindle will then move until the marker on the digital encoder has been found. The TOOL CAL pushbutton light will go out when this reference is reached.

3.5.2 ESTABLISHING PART ZERO

The control internally measures all X and Y positions from its calibrated zero point. However, all absolute program dimensions are relative to a part zero with offsets relative to calibration zero stored in Data Block 0. Refer to figure 3-15. These offsets can be entered via the keyboard in Data Block 0. If, however, the TABLE ZERO button is pressed when in Data Block 0, the current table position becomes the new part zero and is displayed accordingly in X and Y offsets. The following three methods will accurately locate the zero of the part.

