****DRAFT! SHADED SECTIONS OF THIS DOCUMENT NOT REVIEWED YET for FABMO 2022**

ShopBot®

888-680-4466 • ShopBotTools.com

Command Reference for FabMo Software and OpenSBP Language

Introduction

Your new CNC Tools is running on ShopBot's **FabMo** Platform. FabMo stands for Digital Fabrication and Motion. It is a highly capable "open source" system that offers considerable flexibility for operating CNC and robotics tools. You can do many machining tasks directly in FabMo's interface. FabMo will also run, cut, or machine files of two types: g-code files and OpenSBP® files. G-code is a standard communication language for CNC, but it is a dated and inflexible system, developed for more primitive electronics, that is difficult for humans to read interact with. OpenSBP® is an "open syntax" created by ShopBot for CNC and digital fabrication. It offers a friendlier and more flexible machining language than the older g-code. Most CAD/CAM systems can output in the OpenSBP® format. There are over 10,000 ShopBot tools around the world already taking advantage of OpenSBP®.

Your FabMo system will start in OpenSBP®-mode in the "Sb4" application by default. But you can readily move to the home-page "Dashboard" to have access to all of FabMo's direct features, other helpful apps, or to use g-code to run your files. FabMo even has an editor that works with either OpenSBP® or g-code. This reference manual describes the operation of the OpenSBP® Language for CNC and covers the details of its commands. OpenSBP is used for FabMo helpful, built-in Macros. For details of FabMo's g-code implementation built into the open-source, G2-core component of FabMo, see: https://github.com/synthetos/g2/wiki/Gcodes.

In OpenSBP, all operations are accomplished with simple 2-letter commands (e.g. “MX” for “move in the X axis”) from part files or from the command line interface. These commands are the primary "objects" of the language. A command may be followed by one or more parameters that further specify of define the operation (e.g. for MX, how far to move in the X axis; "MX, 5.3"). This document is a full listing and explanation of OpenSBP® Commands and their related parameters. Each command description shows the 2-letter command and full command name at the top, followed by a list of parameters for the command as short abbreviations. This list shows the order that the parameters, separated by commas, must be entered in the Parameter Line or in a part file. After this short listing, a full description of the command and its parameters follows.

When you are entering commands at the Command line, for simple commands you will just add a parameter or two to complete the command. Commands with complex parameters such as [CC; Cut Circle] will automatically bring up a Fill-In sheet for you to complete before entering the Command.

For Fast Access to Information about a Command, Click Here

Table of Contents

Coordinate Systems and Locations in FabMo

The number in the displayed DRO (Digital Read Out) location for an axis as well as the number that will be "moved-to" in an instruction represents the combination of the distance from the machine-base or machine-zero location in that axis PLUS the current offset for that axis.

That is: axis_value = (distance_from_axis_base + axis_current_offset)

The current offset of an axis is the offset of the axis in the currently active, offset-coordinate-system (1-6; in g-code g54-g59). By default, the currently active coordinate system is #2, or g55 in g-code. In FabMo and OpenSBP, we think of g55 as the working offset coordinates, or simply, the working offsets from the Table Base Coordinates.

Most of the standard operations for changing or reassigning the value of a tool's location in an axis work by modifying the working offset for the axis to achieve the desired new working location. These operations include setting the working location to Zero using the "zeroing button" in the manual-move Keypad, or by using the Keypad Set-Location function for setting a working location. The OpenSBP Axis Zeroing Commands (e.g. [**ZX**]) similarly work by changing the working offsets.

(Similar manipulations of axis location can be achieved using g-code commands. And g-code commands can be used to set an alternate coordinate system as temporarily active. ***However, not that g-code zeroing manipulations are not currently synchronized with FabMo so should only be done temporarily within a file.***)

Base Coordinates for your table are set using the "Homing Macro" Macro#3 (or the [**C3**] shortcut in the OpenSBP language). The homing process makes use of the proximity switches and targets on your tool. These will have been set at the factory or during the local assembly of your tool. A typical daily CNC session begins with running the Homing Macro to establish the base location of your tool. These are intended to provide a persistent absolute reference for a working session.

Base Coordinates may also be set manually by invoking the [**ZT**; Zero Table Base Coordinates] command from the Sb4 command line. You would move the tool to the location you want to be considered the Base location in all axes, then use [**ZT**]. Note that ZT will be applied to all active Axes.

Base Coordinates may also be set for special cases using the ['VA'] command whose usage is fully described below.

Note that **software-based limit-checking** ([VL], [SF]) is done with reference to the Table Base Coordinate System and not the current working coordinates.

Acceleration and Deceleration Ramping in FabMo

Commands

CA - Cut Arch

arch length, arch height{ , [O]ut-[I]in-[T]rue, direction, alignment-angle, plunge, repetitions, proportion-X, proportion-Y, <skip>, 1-no pull up at end, 1-do plunge from Z-axis 0}

It is often useful to be able to quickly cut an arch that will span a particular distance. This is straightforward to do for a 180 degree arch (where the span length equals twice the height), but involves some calculation for any other arc length because you don't know how many degrees of arc need to be spanned. Making any arch is easy using this built-in routine because you only need to enter the basic parameters of length and height. You could accomplish the same cut with one of the other circle functions, but by using [CA] you can quickly set up the arch without doing any calculations. Just specify the length of the span, and the height you want the arch to rise to, and the rest is taken care of … though you can further modify the arch by specifying additional parameters. Some of the instructions sound more complicated than they are. Try this one in preview to get a feel for how it works. For general purpose circles or arcs, use [CC] or [CP].

Arch length

The span of the arch. Cutting starts at the current cutter position and moves to the other end of the arch as defined by arch height and arch length, taking into account optional parameters. Arch length is required.

Arch height

How tall the arch will be. The degrees of arc represented by the arch are derived from arch height. If arch height is less than twice the length the arc will be some angle less than 180. See proportion, below, for modifying shape. Arch height is required.

[O]ut-[I]in-[T]rue (default = T)

How the cutter size will be figured in defining the path of the cut. 'O' means that the specified lengths refer to the outside of the cut. 'I' specifies that the lengths refer to the inside. Note that this means that for a given length, 'O' will make a bigger part than 'I'. Thus, to cut -out- an arch of a specified size, use the 'O' feature. To cut an arch shaped hole of a specified size, use the 'I' feature. 'T' moves the cutter on the path of the lengths specified and is the default.

Direction (default = 1)

Direction of the path followed. Use 1 to specify clockwise and -1 to specify counter-clockwise.

Alignment-angle (default = 0)

The angle at which the arch will be created on the table. Default (0) is horizontal, which means the arch will be cut to the right from the start point and oriented horizontally. Orientation can be changed by using an alignment angle that is oriented clockwise from horizontal. An alignment angle of 90 would produce an arc oriented vertically and cut from top to bottom on the Y axis.

Plunge (default = 0)

The depth of plunge for each repetition of the cut. Plunges down are specified with negative numbers. You do not need to plunge if the tool is already at cutting depth from a previous move. You can also plunge before calling the circle function by using the commands [MZ] or [JZ]. Plunge is most useful if you are making the circle or arc in several passes as specified with Repetitions.

Repetitions (default = 1)

Number of passes to be made. This is useful if you're cutting thick material and need to make repeated passes, increasing the depth of the cutter each time, to achieve the full depth of cut. This typically is done with full circles. When you're cutting arcs/arches that are connected to other segments and need to make multiple passes, it's probably better to make multiple repetitions of the whole cut rather (e.g. in the [FP] instruction) than cutting the smaller segments separately.

Proportion X, proportion Y (default = 1,1)

Scaling values for the circle or arc/arches in the X and Y axes. You can alter the X or Y proportion to create ellipses or elliptical arcs/arches. For example, a circle with a diameter of 16, executed with an X proportion of 3 and a Y proportion of 1, will produce an ellipse that is 16 inches wide in the Y dimension and 48 inches wide in

the X dimension. Cutting this shape from a start of 270 degrees to an end of 90 degrees would yield a wide arc/arch.

1-No pull up at end (default = 0).

If this parameter is set to 1, the tool will remain at its plunge depth when it moves on to the next instruction.

1-Offset plunge from Z-Axis 0 (default = 0).

If this parameter is set to 1, the cutter will move to the Z-axis location 0 before the plunge instruction is executed and plunge depth will be taken from 0. This allows a plunge or repeated plunges to be done from the surface of the material or from a given location.

CC – Cut Circle

diameter{ , [O]ut-[I]in-[T]rue, direction, begin-angle, end-angle, plunge, repetitions, proportion-X, proportion-Y, [2-POCKET feature ON; 3-SPIRAL feature ON; 4-SPIRAL & Bottom Pass feature ON], 1-no pull up at end, 1-do plunge from Z-axis 0}

This is a very flexible routine for cutting circles, arcs, ellipses, etc.. If you simply specify a diameter, a circle will be transcribed from the current location, with the cutter following a path ('[T]rue') specified by the diameter and with the cut starting from the top of the circle (12 O'clock as you face a clock). But, there's a lot more you can do here ... have a look

Technical Note on ShopBot Circles Functions. (This note applies to all circles and arcs created by ShopBot [C] commands.) ShopBot converts your instructions for circles and arcs to a series of very small vectors/segments. When circle resolution is set to the lowest value (see [VU]), the vectors are so close to the size of steps themselves, that there is little actual difference between executing the circle as a series of small vectors or as the output of a 'circular interpolation' process. Because the vectors are all buffered in a movement block, they are executed efficiently and smoothly. There may be occasions where controlling the resolution of the circle or the size of the movement block is important and in this case you will find the discussion of 'Circle Segment Resolution' under [VU] of interest. When cutting a very small circle that would require a resolution smaller than the current segment resolution, the software automatically downshifts to a lower resolution and then returns to the resolution that existed before the small circle.

Diameter

The diameter of the circle or arc. See the next paragraph for how this is measured. ShopBot circle functions are ultimately generated from very small line segments. The resolution - size of the line segments - can be set with the [VU] Command. Diameter is the only required parameter for a circle.

The [O]ut-[I]in-[T]rue (default = T)

How the cutter size will be figured in defining the path of the cut. 'O' means that the specified diameter refers to the outside of the cut. 'I' specifies that the diameter refers to the inside. Note that this means that for a given diameter, 'O' will make a bigger circle than 'I'. Thus, to cut a wheel of a specified diameter, use the 'O' feature. To make a hole of a specified diameter, use the 'I' feature. 'T' moves the cutter on the path of the diameter specified and is the default setting.

Direction (default = 1)

The direction of the path followed. Use 1 to specify clockwise and -1 to specify counter-clockwise.

Begin-angle, end-angle (default = ,)

Allows you to specify the starting and ending point of a circle or to define an arc of less than 360 degrees. All cuts begin at the current location of the tool. Angles are specified in degrees as measured with 0 degrees at the top of the circle, that is, at the 12 O'clock position. A cut that starts at 0 degrees and ends at 360 degrees will be a full circle that starts at the top of the circle (assuming a clockwise direction). This circle will be cut below the start point. A cut that starts at 180 degrees and ends at 540 degrees will be a circle that starts and ends at the bottom (i.e., 6 O'clock; assuming a clockwise direction). This circle will be cut above the starting location. Starting at 270 degrees and ending at 630 will produce a circle drawn to the right of the starting point. Similarly, begin-angle, end-angle are used to define the starting point and end point for an arc. Starting at 90 degrees and ending at 180 will produce a 90-degree arc in the lower right quadrant when the direction setting is 1 (clockwise). A direction setting of -1 (counter-clockwise) will produce a 270-degree arc filling all but the lower

right quadrant of a circle (an end angle of -180 degrees will also work in this case). For example, if you were cutting a box and wanted to create a rounded corner after having just cut the bottom edge left to right, you would use a start angle of 180 degrees, an end angle of 90 degrees, and a direction of -1 (counter-clockwise) to produce the lower right rounded corner. The default start and end angles are both blanks. A blank in the start angle, causes the angle to be treated as 0 if a clockwise circle and 360 if a counter-clockwise circle. A blank in the end angle is treated as the end of a full circle, whatever the start point and direction.

Plunge (default = 0)

The depth of plunge for each repetition of the cut. Plunges down are specified with negative numbers. You do not need to plunge if the tool is already at cutting depth from a previous move. You can also plunge before calling the circle function by using the commands [MZ] or [JZ]. Plunge is most useful if you are making the circle or arc cut in several passes as specified with Repetitions.

Repetitions (default = 1)

The number of passes to be made. This is useful if you're cutting thick material and need to make repeated passes, increasing the depth of the cutter each time, to achieve the full depth of cut. This typically is done with full circles. When you're cutting arcs that are connected to other segments and need to make multiple passes, it's probably better to make multiple repetitions of the whole cut rather (e.g. in the [FP] instruction) than cutting the smaller segments separately.

Proportion X, proportion Y (default = 1,1)

Scaling values for the circle or arc in the X and Y axes. You can alter the X or Y proportion to create ellipses or elliptical arcs. For example, a circle with a diameter of 16, executed with an X proportion of 3 and a Y proportion of 1, will produce an ellipse that is 16 inches in the Y dimension and 48 inches in the X dimension. Cutting this shape from a start of 270 degrees to an end of 90 degrees would yield a wide arc/arch.

[2-POCKET feature ON; 3-SPIRAL plunge feature ON; 4-SPIRAL & Bottom Pass feature ON]. (Default = 0)

Activating the POCKETing feature will cause the circle to be 'hollowed out' with increasingly smaller circles being cut inside the initial circle. TAB and POCKET are not used at the same time, and each is designed to work only with full circles.

Activating the SPIRAL feature causes the defined plunge to be made gradually as the cutter is circling down. If repetitions is 1 then the plunge distance is distributed over the whole circle. If repetitions is larger than 1 then the plunge distance is moved for each repetition of the circle (e.g. repeated spiraling). SPIRAL with Bottom Pass causes a final pass at the terminal depth. By default, POCKET, and SPIRAL are OFF (0).

1-No pull up at end (default = 0).

If this parameter is set to 1, the tool will remain at its plunge depth when it moves on to the next instruction.

1-Offset plunge from Z-Axis 0 (default = 0).

If this parameter is set to 1, the cutter will move to the Z-axis location 0 before the plunge instruction is executed and plunge depth will be taken from 0. This allows a plunge or repeated plunges to be done from the surface of the material.

CG - Cut G-code Circle

*-obsolete-, X-end-point, Y-end-point, *X-center-offset(I), *Y-center-offset(J) {,[O]ut-[I]in-[T]rue, direction, plunge, repetitions, proportion-X, proportion-Y, [2-POCKET feature ON; 3-SPIRAL feature ON; 4-SPIRAL & Bottom Pass feature ON], 1-no pull up at end, 1-do plunge from Z-axis 0}*

This command is provided primarily for compatibility with G-Code format and G-Code conversions. It has all the capabilities of [CC] but it makes the specification of the circle/arc dependent on its start point (the current tool position) and the specified end-point. There are two methods for the circle or arc to be computed. Either the diameter can be specified (this works only for arcs less than 180 degrees) or the circle/arc can be derived by entering the location of the center-point of the circle. Here, center-point is indicated by its offset in X and Y from the current location. *Note that only the diameter or the center-offsets may be used. If a diameter is entered, the X and Y offsets will be ignored.

Diameter

The diameter of the circle or arc. See the next paragraph for how this is measured. For this function, diameter is not required. If it is provided, a subsequent X and Y center-point offset will be ignored.

X-end-point, y-end-point

The end point of the circle/arc. If no values are provided, it is assumed the end point is the same as the start point. A circle can not be created from only a diameter in this command.

[O]ut-[I]in-[T]rue (default = T)

How the cutter size will be figured in defining the path of the cut. 'O' means that the specified diameter refers to the outside of the cut. 'I' specifies that the diameter refers to the inside. Note that this means that for a given diameter, 'O' will make a bigger circle than 'I'. Thus, to cut a wheel of a specified diameter, use the 'O' feature. To make a hole of a specified diameter, use the 'I' feature. 'T' moves the cutter on the path of the diameter specified and is the default setting.

Direction (default = 1)

The direction of the path followed. Use 1 to specify clockwise and -1 to specify counter-clockwise.

Offset-to-x-center-point(I), offset-to-y-center-point(J)

The location of the center of the circle from which diameter and arc angle are calculated. If no diameter is given, these parameters are required. *Note that only the diameter or the center-offsets may be used. If a diameter is entered, the X and Y offsets will be ignored. Also, for compatibility with 'modal' G-Code operation, the X or Y offset will be maintained for subsequent calls of the function if no new X or Y value is given (i.e. if no diameter and no offset is specified, the arc will be computed from the previous X and Y center-point offsets).

Plunge (default = 0)

Specifies the depth of plunge for each repetition of the cut. Plunges down are specified with negative numbers. You do not need to plunge if the tool is already at cutting depth from a previous move. You can also plunge before calling the circle function by using the commands [MZ] or [JZ]. Plunge is most useful if you are making the circle or arc cut in several passes as specified with

Repetitions.

X indicates the number of passes to be made. This is useful if you're cutting thick material and need to make repeated passes, increasing the depth of the cutter each time, to achieve the full depth of cut. This typically is done with full circles. When you're cutting arcs that are connected to other segments and need to make multiple passes, it's probably better to make multiple repetitions of the whole cut rather (e.g. in the [FP] instruction) than cutting the smaller segments separately.

Proportion X, proportion Y (default = 1,1)

specify scaling values for the circle or arc in the X and Y axes. You can alter the X or Y proportion to create ellipses or elliptical arcs. For example, a circle with a diameter of 16, executed with an X proportion of 3 and a Y proportion of 1, will produce an ellipse that is 16 inches wide in the Y dimension and 48 inches wide in the X dimension. Cutting this shape from a start of 270 degrees to an end of 90 degrees would yield a wide arc/arch. Default proportions are 1 and 1.

[2-POCKET feature ON; 3-SPIRAL plunge feature ON; 4-SPIRAL & Bottom Pass feature ON]. (Default = 0)

Activating the POCKETing feature will cause the circle to be 'hollowed out' with increasingly smaller circles being cut inside the initial circle. TAB and POCKET are -not- used at the same time, and each is designed to work only with full circles.

Activating the SPIRAL feature causes the defined plunge to be made gradually as the cutter is circling down. If repetitions is 1 then the plunge distance is distributed over the whole circle. If repetitions is larger than 1 then the plunge distance is moved for each repetition of the circle (e.g. repeated spiraling). SPIRAL with Bottom Pass causes a final pass at the terminal depth. By default, POCKET, and SPIRAL are OFF (0).

1-No pull up at end (default = 0).

If this parameter is set to 1, the tool will remain at its plunge depth when it moves on to the next instruction.

1-Offset plunge from Z-Axis 0 (default = 0).

If this parameter is set to 1, the cutter will move to the Z-axis location 0 before the plunge instruction is executed and plunge depth will be taken from 0. This allows a plunge or repeated plunges to be done from the surface of the material.

CP - Cut Circle from Center Point

diameter, X-center-point, Y-center-point {,[O]ut- [I]in-[T]rue, direction, start-angle, end-angle, plunge, repetitions, proportion-X, proportion-Y, [2-POCKET feature ON; 3-SPIRAL feature ON; 4-SPIRAL & Bottom Pass feature ON], 1-no pull up at end, 1-do plunge from Z-axis 0}

This command has all the capabilities of [CC] but it adds the specification of the center-point of the circle to the parameters. When this command is executed the tool moves to the start point of the circle from its current location (*it is up to you to make sure the bit is out of the material). The start point is computed from the given center-point of the circle before the move is started. The [CC] command is usually the most convenient to use if you are creating a circle from a known location of the tool, or moving from a line into an arc segment. However, this [CP] command is very useful when you just want to specify circles in a cutting file. With [CP] you won't need to worry about getting the cutter to the right spot on the circumference to start the cut - this is computed for you. This command thus requires diameter and an X,Y center-point location.

X-center-point, y-center-point

Coordinates for the center of the circle. The tool does not actually ever move to this location. The center-point parameters are required to use this [CP] function.

Diameter

The diameter of the circle or arc. See the next paragraph for how this is measured. ShopBot circle functions are ultimately generated from very small line segments. The resolution - size of the line segments - can be set with the [VU] command. Diameter is third required parameter for a [CP] circle.

The [O]ut-[I]in-[T]rue (default = T)

How the cutter size will be figured in defining the path of the cut. 'O' means that the specified diameter refers to the outside of the cut. 'I' specifies that the diameter refers to the inside. Note that this means that, for a given diameter, 'O' will make a bigger circle than 'I'. Thus, to cut a wheel of a specified diameter, use the 'O' feature. To make a hole of a specified diameter, use the 'I' feature. 'T' moves the cutter on the path of the diameter specified and is the default setting.

Direction (default = 1)

The direction of the path followed. Use 1 to specify clockwise and -1 to specify counter-clockwise.

Begin-angle, end-angle (default = 0,0)

Allows you to specify the starting and ending point of a circle or to define an arc of less than 360 degrees. Angles are specified in degrees as measured with 0 degrees at the top of the circle, that is, at the 12 O'clock position. A cut that starts at 0 degrees and ends at 360 degrees will be a full circle that starts at the top of the circle. This circle will be cut below the start point. A cut that starts at 180 degrees and ends at 540 degrees will be a circle that starts and ends at the bottom (i.e., 6 o'clock). This circle will be cut above the starting location. Similarly, begin-angle, end-angle are used to define the starting point and end point for an arc. Starting at 90 degrees and ending at 180 will produce a 90-degree arc in the lower right quadrant when the direction setting is 1 (clockwise). A direction setting of -1 (counter-clockwise) will produce a 270 degree arc filling all but the lower right quadrant of a circle. See discussion of default values for the [CC] command, which are handled similarly here.

Plunge (default = 0)

The depth of plunge for each repetition of the cut. Plunges down are specified with negative numbers. You do not need to plunge if the tool is already at cutting depth from a previous move. You can also plunge before calling the circle function by using the commands [MZ] or [JZ]. Plunge is most useful if you are making the circle or arc cut in several passes as specified with Repetitions.

Repetitions (default = 1)

The number of passes to be made. This is useful if you're cutting thick material and need to make repeated passes, increasing the depth of the cutter each time, to achieve the full depth of cut. This typically is done with full circles. When you're cutting arcs that are connected to other segments and need to make multiple passes, it's probably better to make multiple repetitions of the whole cut rather (e.g. in the [FP] instruction) than cutting the smaller segments separately.

Proportion X, proportion Y (default = 1,1)

Scaling values for the circle or arc in the X and Y axes. You can alter the X or Y proportion to create ellipses or elliptical arcs. For example, a circle with a diameter of 16, executed with an X proportion of 3 and a Y proportion of 1, will produce an ellipse that is 16 inches in the Y dimension and 48 inches in the X dimension. Cutting this shape from a start of 270 degrees to an end of 90 degrees would yield a wide arc/arch.

[2-POCKET feature ON; 3-SPIRAL plunge feature ON; 4-SPIRAL & Bottom Pass feature ON]. (Default = 0)

Activating the Pocketing feature will cause the circle to be 'hollowed out' with increasingly smaller circles being cut inside the initial circle. TAB and POCKET are -not- used at the same time, and each is designed to work only with full circles.

Activating the SPIRAL feature causes the defined plunge to be made gradually as the cutter is circling down. If repetitions is 1 then the plunge distance is distributed over the whole circle. If repetitions is larger than 1 then the plunge distance is moved for each repetition of the circle (e.g. repeated spiraling). SPIRAL with Bottom Pass causes a final pass at the terminal depth. By default, TAB, POCKET, and SPIRAL are OFF (0).

No pull up at end (default = 0).

If this parameter is set to 1, the tool will remain at its plunge depth when it moves on to the next instruction.

1-Offset plunge from Z-Axis 0 (default = 0).

If this parameter is set to 1, the cutter will move to the Z-axis location 0 before the plunge instruction is executed and plunge depth will be taken from 0. This allows a plunge or repeated plunges to be done from the surface of the material.

CR - Cut Rectangle

Length X{ ,length Y, [O]ut-[I]in-[T]rue, direction, start, plunge, repetitions, [2-POCKET Outside-In feature ON; 3-POCKET Inside-Out feature ON], 1-do plunge from Z-axis 0}

This is a flexible routine for cutting rectangles. If you specify only an X length, a square will be cut, starting from the current location, with the cutter following a path ('[T]rue') specified by the X length, and with the cut starting from the top left hand corner of the square. But MUCH more is possible.

Length X , length Y

The size of the rectangle in terms of the X and Y dimensions. If only X is specified, a square is assumed. An X value is required for the [CR] function. See the next paragraph for how lengths are measured.

The [O]ut-[I]in-[T]rue (default = T)

How the cutter size will be figured in defining the path of the cut. 'O' means that the specified lengths refer to the outside of the cut. 'I' specifies that the lengths refer to the inside. Note that this means that for a given length, 'O' will make a bigger rectangle than 'I'. Thus, to cut a block of a specified size, use the 'O' feature. To cut a rectangular hole of a specified size, use the 'I' feature. 'T' moves the cutter on the path of the lengths specified and is the default.

Direction (default = 1)

The direction of the path followed. Use 1 to specify clockwise and -1 to specify counter-clockwise.

Start (default = 4)

The corner of the rectangle from which the cut starts. Use 1 to specify the upper left, 2 for upper right, 3 for bottom right, and 4 for bottom left. Cutting always starts at the current position of the tool. Thus, a rectangle cut with a starting position of 3 will be placed to the upper left of the starting point. Default start is the bottom left (4).

Plunge (default = 0)

The depth of plunge for each repetition of the cut. Plunges down are specified with negative numbers. You do not need to plunge if the tool is already at cutting depth from a previous move. You also can plunge by using the commands [MZ] or [JZ] before calling the rectangle function. Plunge is most useful if you're making the rectangle cut in several passes as specified with Repetitions.

Repetitions (default = 1)

The number of passes to be made. This is useful if you're cutting thick material and need to make repeated passes, increasing the depth of the cutter each time, to achieve the full depth of cut.

[2-POCKET feature ON] (default = 0).

Activating the POCKETing feature will cause the rectangle to be 'hollowed out' with increasingly smaller rectangles being cut inside the initial rectangle. TAB and POCKET are not used at the same time. By default, POCKET is OFF.

1-Offset plunge from Z-Axis 0 (default = 0).

If this parameter is set to 1, the cutter will move to the Z-axis location 0 before the plunge instruction is executed and plunge depth will be taken from 0. This allows a plunge or repeated plunges to be done from the surface of the material.

J2 - Jog 2 Dimensions

{x-location/distance, y-location/distance}

Makes an X and Y axis move to the designated location or a specified distance (if in Relative Mode) at current Jog speed. These 2-D interpolated moves are made on a diagonal from the current location to the end location indicated in the location/distance values, which may be absolute or relative depending on the Move Mode setting. If a value is not entered for one of the parameters, the value defaults to the current location assuming a comma is used as a spacer to designate the correct parameter field (cf. 'modal' in G-code programming).

J3 - Jog 3 Dimensions

{x-location/distance, y-location/distance, z-location/distance}

Makes an X, Y, and Z axis move to the designated location or a specified distance (if in Relative Mode) at current Jog speed. These 3-D interpolated moves are made on a diagonal from the current location to the end location indicated in the location/distance values, which may be absolute or relative depending on the Move Mode setting. If a value is not entered for one of the parameters, the value defaults to the current location assuming a comma is used as a spacer to designate the correct parameter field.

J4 - Jog 4 Dimensions

{x-location/distance, y-location/distance, z-location/distance, a-location/distance}

Makes an X, Y, Z and A axis move to the designated location or a specified distance (if in Relative Mode) at current Jog speed. These 3-D interpolated moves are made on a diagonal from the current location to the end location indicated in the location/distance values, which may be absolute or relative depending on the Move Mode setting. If a value is not entered for one of the parameters, the value defaults to the current location assuming a comma is used as a spacer to designate the correct parameter field.

J5 - Jog 5 Dimensions

{x-location/distance, y-location/distance, z-location/distance, a-location/distance, b-location/distance}

Makes an X, Y, Z, A, and B axis move to the designated location or a specified distance (if in Relative Mode)at current Jog speed. These 3-D interpolated moves are made on a diagonal from the current location to the end location indicated in the location/distance values, which may be absolute or relative depending on the Move

Mode setting. If a value is not entered for one of the parameters, the value defaults to the current location assuming a comma is used as a spacer to designate the correct parameter field.

J6 - Jog 6 Dimensions

{x-location/distance, y-location/distance, z-location/distance, a-location/distance, b-location/distance, c-location/distance }

Makes an X, Y, Z, A, B and C axis move to the designated location or a specified distance (if in Relative Mode)at current Jog speed. These 3-D interpolated moves are made on a diagonal from the current location to the end location indicated in the location/distance values, which may be absolute or relative depending on the Move Mode setting. If a value is not entered for one of the parameters, the value defaults to the current location assuming a comma is used as a spacer to designate the correct parameter field.

JX - Jog X-axis

x-location/distance

Makes an X axis move to the designated location or a specified distance (if in Relative Mode) at Jog speed. These moves are made from the current location to the end location indicated in the location/distance values, which may be absolute or relative depending on the Move Mode setting.

JY - Jog - Y axis

y-location/distance

Makes a Y axis move to the designated location or a specified distance (if in Relative Mode) at Jog speed. These moves are made from the current location to the end location indicated in the location/distance values, which may be absolute or relative depending on the Move Mode setting.

JZ - Jog Z-axis

z-location/distance

Makes a Z axis move to the designated location or a specified distance (if in Relative Mode) at Jog speed. These moves are made from the current location to the end location indicated in the location/distance values, which may be absolute or relative depending on the Move Mode setting.

JA - Jog A axis

a-location/distance

Makes the A axis move to the designated location or a specified distance (if in Relative Mode) at Jog speed. These moves are made from the current location to the end location indicated in the location/distance values, which may be absolute or relative depending on the Move Mode setting. The A axis is sometimes used as a rotational axis. When this is done, the location/distance value may be in degrees or other arbitrary units as defined by its Unit values with [VU].

JB- Jog B axis

b-location/distance

Makes the B axis move to the designated location or a specified distance (if in Relative Mode) at Jog speed. These moves are made from the current location to the end location indicated in the location/distance values, which may be absolute or relative depending on the Move Mode setting. The B axis is sometimes used as a rotational axis. When this is done, the location/distance value may be in degrees or other arbitrary units as defined by its Unit values with [VU].

JC- Jog C axis

c-location/distance

Makes the C axis move to the designated location or a specified distance (if in Relative Mode) at Jog speed. These moves are made from the current location to the end location indicated in the location/distance values, which may be absolute or relative depending on the Move Mode setting. The C axis is sometimes used as a rotational axis. When this is done, the location/distance value may be in degrees or other arbitrary units as defined by its Unit values with [VU].

JS - Jog Speed

{xy-Speed, z-Speed, a-Speed, b-Speed, c-speed}

Sets the speed in inches or mm per second for Jog Speed movement in the XY, Z, A, and B axes. The new values will appear on screen in the upper left panel. This speed represents the actual movement speed of the cutter tip. Because the vector speed along a diagonal represents the geometric addition of the X and Y vectors, for any given speed, the motors will operate more slowly on a diagonal move than on a vertical or horizontal move. The change you notice is the sound of the motors is as it should be. For a circle the speed of the motors will continuously change up and down in order to achieve a constant vectored movement speed.

This command is similar to the [MS] command for setting Move speed. It also has the same effect as setting the 4th and 5th parameter of [VS; Values Speed] command, which lets you set all 6 speeds (including accessory axis) at the same time.

Note: MS and JS, unlike VS, when used in a file as processed as not processed as components of a “move stack” if they are placed between a series of moves. They punctuate stack. Thus MS and JS commands will result in a ramp-down, ramp-up sequence around the speed change. You should use the VS command if you do not want to disrupt ongoing motion. See Programming Handbook for information on “move stacks”.

SEE ALSO: [VR] to set the acceleration ramps and deceleration ramps for each type of movement.

JH - Jog Home

Moves the tool to the home (0,0) XY position at Jog speed. If the tool is positioned at a plunge location below that defined as the Safe-Z height (default .15) on the Z Axis, the Z-axis is first automatically moved to the current ‘Safe-Z’ height (Safe-Z is set or turned off with [VC]). If the tool has already been moved above Safe-Z, it is left in that position. Then an immediate move to X, Y location 0, 0 at Jog speed is executed. Action starts immediately when the [H] is entered with no pause for parameters occurring. Note that Relative mode does not apply to [MH] or [JH]. The tool is always returned to Absolute 0, 0

MX - Move X axis

x-location/distance

Make an X axis move to the designated location or a specified distance (if in Relative Mode) at Move speed. These moves are made from the current location to the end location indicated in the location/distance values, which may be absolute or relative depending on the Move Mode setting.

MY - Move Y axis

y-location/distance

Make a Y axis move to the designated location or a specified distance (if in Relative Mode) at Move speed. These moves are made from the current location to the end location indicated in the location/distance values, which may be absolute or relative depending on the Move Mode setting.

MZ - Move Z axis

z-location/distance

Make a Z axis move to the designated location or a specified distance (if in Relative Mode) at Move speed. These moves are made from the current location to the end location indicated in the location/distance values, which may be absolute or relative depending on the Move Mode setting.

MA - Move A axis

a-location/distance

Makes an A axis move to the designated location or a specified distance (if in Relative Mode) at Move speed. These moves are made from the current location to the end location indicated in the location/distance values, which may be absolute or relative depending on the Move Mode setting. The A axis is sometimes used as a rotational axis. When this is done, the location/distance value may be in degrees or other arbitrary units as defined by its Unit values with [VU].

MB - Move B axis

b-location/distance

Makes a B axis move to the designated location or a specified distance (if in Relative Mode) at Move speed. These moves are made from the current location to the end location indicated in the location/distance values, which may be absolute or relative depending on the Move Mode setting. The B axis is sometimes used as a rotational axis. When this is done, the location/distance value may be in degrees or other arbitrary units as defined by its Unit values with [VU].

M2 - Move 2 Dimensions

{x-location/distance, y-location/distance}

Makes an X and Y axis move to the designated location or a specified distance (if in Relative Mode) at current Move or cutting speed. These 2-D interpolated moves are made on a diagonal from the current location to the end location indicated in the location/distance values, which may be absolute or relative depending on the Move Mode setting. If a value is not entered for one of the parameters, the value defaults to the current location assuming a comma is used as a spacer to designate the correct parameter field (this would be the same as 'modal' in G-code programming).

M3 - Move 3 Dimensions

{x-location/distance, y-location/distance, z-location/distance}

Makes an X, Y, and Z axis move to the designated location or a specified distance (if in Relative Mode) at current Move or cutting speed. These 3-D interpolated moves are made on a diagonal from the current location to the end location indicated in the location/distance values, which may be absolute or relative depending on the Move Mode setting. If a value is not entered for one of the parameters, the value defaults to the current location assuming a comma is used as a spacer to designate the correct parameter field.

M4 - Move 4 Dimensions

{x-location/distance, y-location/distance, z-location/distance, a-location/distance}

Makes an X, Y, Z, and A axis move to the designated location or a specified distance (if in Relative Mode) at current Move or cutting speed. These 3-D interpolated moves are made on a diagonal from the current location to the end location indicated in the location/distance values, which may be absolute or relative depending on the

Move Mode setting. If a value is not entered for one of the parameters, the value defaults to the current location assuming a comma is used as a spacer to designate the correct parameter field.

M5 – Move 5 Dimensions

{x-location/distance, y-location/distance, z-location/distance, a-location/distance, b-location/distance}

Makes an X, Y, Z, A and B axis move to the designated location or a specified distance (if in Relative Mode) at current Move or cutting speed. These 3-D interpolated moves are made on a diagonal from the current location to the end location indicated in the location/distance values, which may be absolute or relative depending on the Move Mode setting. If a value is not entered for one of the parameters, the value defaults to the current location assuming a comma is used as a spacer to designate the correct parameter field.

M6 – Move 6 Dimensions

{x-location/distance, y-location/distance, z-location/distance, a-location/distance, b-location/distance, c-location/distance }

Makes an X, Y, Z, A, B and C axis move to the designated location or a specified distance (if in Relative Mode) at current Move or cutting speed. These 3-D interpolated moves are made on a diagonal from the current location to the end location indicated in the location/distance values, which may be absolute or relative depending on the Move Mode setting. If a value is not entered for one of the parameters, the value defaults to the current location assuming a comma is used as a spacer to designate the correct parameter field.

MH - Move Home

Moves the tool to the home (0,0) XY position at Move speed. If the tool is positioned at a plunge location below that defined as the Safe-Z height (default .15) on the Z Axis, the Z-axis is first automatically moved to the current 'Safe-Z' height (Safe-Z is set or turned off with [VC]). If the tool has already been moved above Safe-Z, it is left in that position. Then an immediate move to X, Y location 0, 0 at move speed is executed. Action starts immediately when the [H] is entered with no pause for parameters occurring. Note that Relative mode does not apply to [MH] or [JH]. The tool is always returned to Absolute 0, 0.

MS - Move Speed Set

{xy-Speed, z-Speed, a-Speed, b-Speed, c-Speed}

Sets the speed in inches or mm per second for Move Speed movement in the XY, Z, A, B and C axes. The new values will appear on the left side of the control console. This speed represents the actual movement speed (feed rate) of the cutter tip. Because the vector speed along a diagonal is the geometric addition of the X and Y vectors, for any given speed, the motors will operate more slowly on a diagonal move than on a straight X or Y move, even though the vector speed of the cutter is accurate. The change you notice in the sound of the motors is as it should be. For a circle or arc the speed of the motors will continuously change up and down in order to achieve a constant vectored movement speed around the circle or arc.

This command is similar to the [JS] command for changing Jog Speed and both of these commands are subsets of the [VS; Values Speed] command, which lets you set all speeds at the same time.

Note: MS and JS, unlike VS, when used in a file as processed as not processed as components of a "move stack" if they are placed between a series of moves. They punctuate stack. Thus MS and JS commands will result in a ramp-down, ramp-up sequence around the speed change. You should use the VS command if you do not want to disrupt ongoing motion. See Programming Handbook for information on "move stacks".

SEE ALSO: [VR] to set the acceleration ramps and deceleration ramps for each type of movement
SEE ALSO: [VU] for setting extremely slow speeds

PX – Probe in X Axis

switch number, speed, target x-location

The PX command sets the tool in motion at the specified speed, until the specified switch is activated. This can be used to probe for material surface, a proximity switch or in other applications. The location specified is the location where the tool will stop if the input switch is never triggered.

PY – Probe in Y Axis

switch number, speed, target y-location

The PY command sets the tool in motion at the specified speed, until the specified switch is activated. This can be used to probe for material surface, a proximity switch or in other applications. The location specified is the location where the tool will stop if the input switch is never triggered.

PZ – Probe in Z Axis

switch number, speed, target z-location

The PZ command sets the tool in motion at the specified speed, until the specified switch is activated. This can be used to probe for material surface, a proximity switch or in other applications. The location specified is the location where the tool will stop if the input switch is never triggered.

PA – Probe in A Axis

switch number, speed, target a-location

The PA command sets the tool in motion at the specified speed, until the specified switch is activated. This can be used to probe for material surface, a proximity switch or in other applications. The location specified is the location where the tool will stop if the input switch is never triggered.

PB – Probe in B Axis

switch number, speed, target b-location

The PB command sets the tool in motion at the specified speed, until the specified switch is activated. This can be used to probe for material surface, a proximity switch or in other applications. The location specified is the location where the tool will stop if the input switch is never triggered.

PC – Probe in C Axis

switch number, speed, target c-location

The PC command sets the tool in motion at the specified speed, until the specified switch is activated. This can be used to probe for material surface, a proximity switch or in other applications. The location specified is the location where the tool will stop if the input switch is never triggered.

P2 – Probe in 2 Dimensions

switch number, speed, target x-location, target y-location

The P2 command sets the tool in motion at the specified speed, until the specified switch is activated. This can be used to probe for material surface, a proximity switch or in other applications. The distance specified is the maximum distance that the tool will travel before stopping, which prevents the tool from going on and on forever in the case that the input switch trigger is never triggered.

P3 – Probe in 3 Dimensions

switch number, speed, target x-location, target y-location, target z-location

The P3 command sets the tool in motion at the specified speed, until the specified switch is activated. This can be used to probe for material surface, a proximity switch or in other applications. The distance specified is the maximum distance that the tool will travel before stopping, which prevents the tool from going on and on forever in the case that the input switch trigger is never triggered.

P4 – Probe in 4 Dimensions

switch number, speed, target x-location, target y-location, target z-location, target a-location

The P4 command sets the tool in motion at the specified speed, until the specified switch is activated. This can be used to probe for material surface, a proximity switch or in other applications. The distance specified is the maximum distance that the tool will travel before stopping, which prevents the tool from going on and on forever in the case that the input switch trigger is never triggered.

P5 – Probe in 5 Dimensions

switch number, speed, target x-location, target y-location, target z-location, target a-location, target b-location

The P5 command sets the tool in motion at the specified speed, until the specified switch is activated. This can be used to probe for material surface, a proximity switch or in other applications. The distance specified is the maximum distance that the tool will travel before stopping, which prevents the tool from going on and on forever in the case that the input switch trigger is never triggered.

P6 – Probe in 6 Dimensions

switch number, speed, target x-location, target y-location target z-location, target a-location, target b-location, target c-location

The P6 command sets the tool in motion at the specified speed, until the specified switch is activated. This can be used to probe for material surface, a proximity switch or in other applications. The distance specified is the maximum distance that the tool will travel before stopping, which prevents the tool from going on and on forever in the case that the input switch trigger is never triggered.

SA - Set to Absolute Distances

This command sets your tool for Absolute Distance. All distances entered for moves will cause tool to move to the specified location measured from the 0, 0 point. In this mode you direct the tool as if it is on an XYZ graph. If you send the tool to location 3, 7 it will go to 3 inches (or mm) from 0 on the X axis and 7 inches from 0 on the Y axis from where ever it is. Once at this location, if you tell it to go to location 3, 7 it will not move, because it is already there. Your tool is always set to either Absolute or Relative distances.

We believe that for most purposes it is best to use Absolute Distances for Part Files. However, there are special situations where Relative Distances can be very useful, see [SR]. The current Distance mode is displayed as a toggle switch in the Status Panel in the upper right portion of the display.

See also: [SR]

SC - Set Current Coordinate System

coordinate system number

This command sets the current working coordinate system to the specified value. Currently, there are two coordinate systems available – the “Table Base” coordinate system (0), which is usually set when the machine

is homed and isn't changed, and the "User" coordinate system (1) which is what is changed when the zeroing commands are used.

See also: [VA]

SK – Set to Keypad Mode

{Keypad Message}

When this command is encountered, the tool will switch to "Manual" or "Keypad" mode – this brings up the keypad display on any connected client, allowing the user to drive the tool around manually, zero the position, run macros, etc. Program execution continues when the keypad is dismissed by the user. The optional message will be displayed in the keypad dialog, and can be used to prompt the user to perform a specific action with the keypad (E.g. "Zero at the table surface before continuing.")

See also: [VA]

SO - Set Output Switch

switch number, [0-OFF or 1-ON]

This command turns OFF or ON a specified output switch. There are eight switches numbered 1-12. [MO] turns off all output switches as well as motors (as explained above). See the Users Guide for information on hooking up and using the output switch circuits. The [SO] command can be used from the keyboard or from within files. The condition of switches is displayed in the setting panel in the upper right portion of the settings display. Switches are all turned OFF when you leave the ShopBot Control program and the state of switches is not saved or available when the software is re-started.

Switch number

is the number from 1 to 12 of the switch you want to turn on or off. Special Cases: Switch number 4 (default = 4,-1) is automatically activated when the tool is in motion. It is set to ON at the time the Warning Signal (see [SW]) begins to flash and beep, and in this way could be used to activate a relay that controls other processes, such as external warning signals or accessories such as dust collectors. This function for Switch 4 can be adjusted or disabled. If switch 4 is set to -1, the switch will come on with the Warning Signal just before movement begins and go off when the warning goes off at the end of the move; if set to -2 it will also go off if the PANIC Bar is hit; if set to -3 it will go off if the PANIC bar is hit or if the tool 'Stalls' (cable drive only); if set to -4 this function will be disabled so that the switch can be used for other purposes.

0-OFF; 1-ON

Sets the condition of the switch. Entering no value turns the switch off.

See also: [MO], [SW], Part File Programming Instruction ON INPUT

SR - Set to Relative Distances

This command sets your tool to Relative Distances. All distances entered for moves will move the tool the specified distance from its current location. If you send the tool to 3, 7 in Relative Distance, it will go 3 inches (or mm) in the positive X direction *from where it is*, and 7 inches in the positive Y direction *from where it is* (i.e. it is as if your current position is temporarily taken to be 0, 0). Once at this new location, if you tell it to go to 3, 7 while still in Relative Distance it will again move 3 inches in X and 7 inches in Y because it makes a move Relative to its current position. (The tool always is set either to absolute or relative distances.)

Relative Distance is handy when doing work from the keyboard. For example, in Relative Distance you might produce a repeated drilling pattern by first drilling the hole, then advancing say 1.5 inches in the X direction, then simply using RR to repeat the commands as many times as you wanted holes. Brackets are used here to indicate these are commands entered from the keyboard, not in a file. Explanatory comments follow the apostrophe.

```
[SA]          'in absolute distance
[M3] 1,1,.2   'move to the start point
[SR]          'set to Relative Distance
[MZ] -.45     'plunge to .25 inches below surface 0, we are going relative from .2
[JZ] .45      'pull back up to where we started
[JX] 1.5      'move to next drilling position 1.5 inches up X axis
```

Relative Distance can be handy when you're operating your tool from the keyboard, but we don't recommend using Relative Distance in Part Files for two reasons: 1) rounding errors can incrementally add up to unexpected errors in large files; 2) it is hard to look at the numbers in a Relative Distance file and know where you are in the file because none of the numbers represent real locations on the cutting table. The current Distance mode is displayed as a toggle switch in the Status Panel in the upper right portion of the display.

See also: [SA]

ST - Set Location to Table Base Coordinates

FabMo maintains a number of internal coordinate systems against which the current position of the tool are resolved. At any time, the current working coordinates can be reset to what they are in terms of the base values using this [ST] command. The location of the Base Coordinate zeros are initially defined using the [ZT] command. Having base coordinates provides a system for maintaining an absolute location vs working location. The Base Coordinate system also provides the information for software limit checking because it maintains an absolute relationship to the physical table.

Note, that as with the working coordinate locations, if the tool is moved manually rather than with the software, this absolute location will be lost.

See also: [ZT], [VA] and various [Z] functions

SV - Set Values to Permanent

Several Commands which change parameter, when used in a Part File, only change the parameters for the duration of that Part File. These Commands include: [VS], [VI], [VU], and Limit System Changes. When these Commands are used in a file, when the file ends or is terminated by the user, the values are are Reset to their condition at the start of the Part File.

The [SV] Command is a way to make changes in Part Files permanent. At the point the [SV] Command is issued in a file, all temporary values become permanent, as if they were in effect at the start of the file. Changes made after the [SV] Command, will then Reset to the new permanent values if the file is terminated.

VA - Values for Axis Locations

{x-location, y-location, z-location, a-location, b-location, c-location, tableBase-x-location, tableBase-y-location, tableBase-z-location, tableBase-a-location, tableBase-b-location, tableBase-c-location }

For background to locations and coordinates see: Coordinate Systems and Locations in FabMo
This command sets two different aspects of the location coordinate system. The first 6 parameters allow changing the working location. The command can be used from the Sb4 command line or from an OpenSBP file to set new working coordinates for any axis. The second 6 parameters allow changing the distance from the base location of an axis. That is, these latter parameters have the effect of moving the tool's 0 location.

If both sets of parameters are utilized, then the effect of the second parameters, the change to the distance from the axis base location, will be applied first.

(x,y,z,a,b or c)-change working location

The first 6 parameters, set a new working coordinate for one or more axis. That is, if you use [VA] to set the x-axis location to, say, 5.3, then no matter where your tool is positioned, the location will now be considered to be 5.3 and this is what the location display will be changed to. You can also use this command to define a new working region of your tool. Note that when you change working coordinates with [VA] you do not change the Table Base Coordinates. All axes working offsets can be removed and Table Base Coordinates restored with the [ST] command.

(x,y,z,a,b or c)-reset distance from axis base location

The second 6 parameters allow specifically setting distance from the Base coordinates independently of moving the tool. Normally, the Base coordinates are set during Homing or are set manually to a location for all axes simultaneously using the [ZT] command at the point the tool is at its new Base (home, or 0,0,0,0,0,0) location. These parameters allow setting each axes' distance from base individually.

Using the command: "VA,0,0,0,0,0,0" would be the same as using the [**ST**] command.
Using the command: "VA, ,,,,,, 0,0,0,0,0,0" would be the same as using the [**ZT**] command.

See also: [ST] and [Z] commands.

VC - Value for Cutter-related Parameters

{diameter, -obsolete-, -obsolete-, safe-Z-pull-up, -obsolete-, % pocket overlap, safe-A-pull-up}

This command sets the values for several cutter-related functions including the Safe-Z-pull-up feature.

Diameter (default = .25)

The diameter of the current cutter in your tool. This diameter is used with the built-in 'Cut' functions where measurement is based on inside, outside, or true diameter or width (e.g. in [CC] or [CR]). Cutter diameter can also be accessed from within a Part file using the system variable %(20). The default is .25 inch.

-obsolete parameters-

Leave empty as space-holder for parameters no longer used.

Safe-Z-pull-up (default = 1)

The height that the Z axis will move up to in several operations intended to pull the bit out of the material and to a safe transit height. These operations include the commands [MH] and [JH] as well as movement block reloads where the bit is withdrawn temporarily from the material. The safe-Z height is also the height the Z axis must be lifted to in order to disable the tabbing function in a file that has several entities to be tabbed.

%Pocket overlap (default = 10)

overlap tells the pocketing feature in [CC], [CR] and related commands, how much overlap should be computed for cutter path when the tool is carrying out 'pocketing functions'.

Safe-A-pull-up (default = 1)

The height that the A axis will move up to in several operations intended to pull the bit out of the material and to a safe transit height. These operations include the commands [MH] and [JH].

See also: [C..] commands and [MH], [JH]

VD - Values for Display Settings

{-obsolete-, linear units, units type A, units type B, units type B}

Note: The parameters for this command are different in FabMo than in SB3.

Linear Units (default = inches)

Sets the X, Y and Z axes to run in inches (0) or millimeters (1). One inch = 25.4mm. The default is in inches, so the values given in the Unit Values for Different Gear Ratios assume that you are working in inches.

When the unit value is changed, all system values are recalculated according to the new unit type and saved. That means that locations, calibrations, tool sizes, speeds, ramps and so on will automatically be converted to the correct value in the alternate unit system. If you are in inches, all added measurement values need to be entered in inches. If you are in mm, all values need to be entered in mm.

Units Type A Axis (default = inches)

Sets the Accessory Axis (A) (the fourth axis) to run in inches (0), mm (1) or degrees (2). An indexer would run in degrees of rotation rather than inches or mm.

Units Type B Axis (default = degrees)

Sets the Accessory Axis (B) (the fifth axis) to run in inches (0), mm (1) or degrees (2). An indexer would run in degrees of rotation rather than inches or mm.

Units Type C Axis (default = degrees)

Sets the Accessory Axis (C) (the fifth axis) to run in inches (0), mm (1) or degrees (2). An indexer would run in degrees of rotation rather than inches or mm.

VI - Values for Axes assignment to Drivers

{Driver Channel 1, Driver Channel 2, Driver Channel 3, Driver Channel 4, Speed, Driver Channel 5, Driver Channel 6}

This command sets various driver channel parameters.

Note: If this Command is used within a Part File, values will automatically revert to the starting value when the Part File ends or is terminated by the user. To over-ride this automatic function, see [SV].

Driver/Channel Definition

Every motor on the ShopBot has a driver in the control box that determines the direction and step of that motor. Each driver is associated with a driver channel.

In a standard configuration, the axes associated with the driver channels are as follows:
Driver Channel 1 – X axis.
Driver Channel 2 – Y axis.
Driver Channel 3 – Z axis.
Driver Channel 4 – A axis.
Driver Channel 5 – B axis
Driver Channel 6 – C axis.

VL - Values for Limits for Table

{low-X, high-X, low-Y, high-Y, low-Z, high-Z, low-Acc, high-Acc, low-B, high-B, low-C, high-C }

This command sets Table Limits for your tool. When a command that would take your tool out of the defined movement range is encountered, an error message is generated and you are offered the opportunity to 'Continue' or to end the process that is currently executing. Table Limits are referenced to the Table Base Coordinates not to the current working location or to the location in an 'offset' file. Limits that don't match the actual physical layout of the tool could be set up for special purposes.

Low-X,Y,Z,A,B or C

Low values are the smallest value for a table axis -- the smallest location point to which travel can go on that axis. In the Z-axis, the low value would be the deepest plunge value accepted

High-X,Y,Z,A,B or C

High values are for the largest value to which a tool can move in an axis. In the Z axis the high value would be the highest pull up value accepted.

VR - Values for Ramps

{XY Jerk Limit, Z Jerk Limit, A Jerk Limit, B Jerk Limit, C Jerk Limit}

Defines the acceleration/deceleration ramps at the beginning and end of each type of move. Ramping makes for smoother cuts and enhanced control of the tool movement. The values represent a “jerk limit” which constrains how fast the machine can accelerate or decelerate. The default values initially set are appropriate for the default speeds. With ramping you should be going for values that give smooth movement of your tool and reduce vibration caused by fast starts or stops – while maintaining efficient cutting. 50 (the default value) is a conservative value that will result in relatively slow acceleration and deceleration. 100 is sort of a middle-of-the-road figure that strikes a good balance between speed and power/vibration. 150 is fairly aggressive. 250 is the default value on most tools used for keyboard/keypad moves that provides very quick starts and stops (but values this high are generally not recommended for modest move speeds)

XY, Z, A, B, C Move Jerk Limit (def =50)

The Move Ramp Speed sets the start and end speed for acceleration and deceleration ramps for each axis for cutting speeds. The

VS - Values for Speeds

{XY Move Speed, Z Move Speed, A Move Speed, B Move Speed, C Move Speed, XY Jog Speed, Z Jog Speed, A Jog Speed, B Jog Speed, C Jog Speed }

This command sets speed values for every axis with a single command. It is similar to [MS] and [JS] but provides access to all 10 speeds. Speeds are defined in inches/sec or mm/sec depending on the measurement unit.

Note: If this Command is used within a Part File, Speeds will automatically revert to the starting value when the Part File ends or is terminated by the user. To over-ride this automatic function, see [SV].

Move Speed

Sets the speed that will typically be used for cutting.

Jog Speed

Sets the speed for rapid moves. From your tool’s point of view, there is no difference between move and jog, only that you have set them up to be carried out at different speeds.

Speeds are set in inches or mm per second, depending on the unit setting. These speeds represent the actual X and Y movement speed of the cutter tip. Because the vector speed along a diagonal represents the geometric addition of the X and Y vectors, for any given speed, the motors will operate more slowly on a diagonal move than on a vertical or horizontal move. The change you notice is the sound of the motors is as it should be. For a circle the speed of the motors will continuously change up and down in order to achieve a constant vectored movement speed.

See also: [MS], [JS], see [VU] for setting extremely slow speeds

Z Commands Note

For background to locations and coordinates see: Coordinate Systems and Locations in FabMo
The 'Z' commands which follow are used to set the working 0 location values for your tool on any axis. Note that they are very powerful and should be used with care, since they reset your position. There are several related commands that give additional control over the designation of location and working offsets from your tool's true zero locations in each axis. We refer to the machine zeroes of each axis as the "Table Base Coordinates".

[VA] is a command that is similar to the 'Z' commands below in that it resets the working location of your tool. With [VA] you can set the current location to any value you like, including zero.

Because FabMo keeps track of Table Base Coordinates, your absolute location in reference to the true axis zeroes can always be re-established with [ST] the command. The Table Base Coordinates are set up with the homing routines or can be established manually with [ZT] (see below). Note that software-based limit-checking ([VL], [SF]) is done with reference to the base coordinate system and not the current working coordinates.

ZX - Zero X axis working location

This command zeros out (sets to 0.000) the current location of the X axis of your tool. That is, after [ZX] is used, the current physical location of your tool will now be considered 0 for the X axis. It is your new working location in this axis with an offset from the machine 0. See Z Commands Notes for more information.

ZY - Zero Y axis working location

This command zeros out (sets to 0.000) the current location of the Y axis of your tool. That is, after [ZY] is used, the current physical location of your tool will now be considered 0 for the Y axis. It is your new working location in this axis with an offset from the machine 0. See Z Commands Notes for more information.

ZZ - Zero Z axis working location

This command zeros out (sets to 0.000) the current location of the Z axis of your tool. That is, after [ZZ] is used, the current physical location of your tool will now be considered 0 for the Z axis. It is your new working location in this axis with an offset from the machine 0. See Z Commands Notes for more information.

ZA - Zero A axis working location

This command zeros out (sets to 0.000) the current location of the A axis of your tool. That is, after [ZA] is used, the current physical location of your tool will now be considered 0 for the A axis. It is your new working location in this axis with an offset from the machine 0. See Z Commands Notes for more information.

ZB - Zero B axis working location

This command zeros out (sets to 0.000) the current location of the B axis of your tool. That is, after [ZB] is used, the current physical location of your tool will now be considered 0 for the B axis. It is your new working location in this axis with an offset from the machine 0. See Z Commands Notes for more information.

ZC - Zero C axis working location

This command zeros out (sets to 0.000) the current location of the C axis of your tool. That is, after [ZC] is used, the current physical location of your tool will now be considered 0 for the C axis. It is your new working location in this axis with an offset from the machine 0. See Z Commands Notes for more information.

Z2 - Zero in 2D (X & Y) axes working location

This command zeros out (sets to 0.000) the current location of the X and Y axes of your tool. That is, after [Z2] is used, the current physical location of your tool will now be considered 0 for the X and Y axes. It is your new working location in these axes with an offset from the machine 0's. See Z Commands Notes for more information.

Z3 - Zero in 3D (X, Y & Z) axes working location

This command zeros out (sets to 0.000) the current location of the X, Y, and Z axes of your tool. That is, after [ZA] is used, the current physical location of your tool will now be considered 0 for the X, Y, and Z axes. It is your new working location in these axes with an offset from the machine 0's. See Z Commands Notes for more information.

Z4 - Zero in 4D (X, Y, Z, & A) axes working location

This command zeros out (sets to 0.000) the current location of ALL axes of your tool. That is, after [Z4] is used, the current physical location of your tool will now be considered 0 for the X,Y,Z, and A axes. It is your new working location in these axes with an offset from the machine 0's. See Z Commands Notes for more information.

Z5 - Zero in 5D (X, Y, Z, A, and B) axes working location

This command zeros out (sets to 0.000) the current location of ALL axes of your tool. That is, after [Z5] is used, the current physical location of your tool will now be considered 0 for all 5 axes. It is your new working location in these axes with an offset from the machine 0's. See Z Commands Notes for more information.

Z6 - Zero in 6D (X, Y, Z, A, B and C) axes working location

This command zeros out (sets to 0.000) the current location of ALL axes of your tool. That is, after [Z6] is used, the current physical location of your tool will now be considered 0 for all 6 axes. It is your new working location in these axes with an offset from the machine 0's. See Z Commands Notes for more information.

ZT - Zero Table Base Coordinates

FabMo maintains a Table Base Coordinate System (reference or absolute) Coordinate System that can be used to re-establish location in terms of your tools real zero points after you have been using working coordinates. This system is set up during 'homing' and the zeroing of the Z-axis, procedures normally carried out at the beginning of a machining session. Table Base Coordinates can also be established with the [ZT] command which has the effect of calling the location that the tool is currently at its true zero in each axis -- thus establishing the Table Base Coordinate System. [ZT] is a powerful command! If you want to manually set the base zero location for your tool, you would use the manual keypad to drive the tool to the positions at which you want the zero established. Then use the [ZT] command to make it so.

If you subsequently use the working zero commands such as [ZX] to set working coordinates, the Base Coordinate system can be re-established with [ST]. Note that software-based limit-checking ([VL], [SF]) is done with reference to the Base Coordinate System and not the current working coordinates.

NOTE: The Base Coordinate System is maintained in your computer. If an axis is pushed manually or is physically restrained during a move, the correct Base Coordinates will be lost. They will also be lost when you turn off power or in the event of a power failure.

In an unusual situation, the second 4 parameters of [VA] can be used to set Base Coordinate System values to specific values.

See also: [VA], [ST], [VL], [SF] other [Z..] commands