WEAR YOUR SAFETY GLASSES

FORESIGHT IS BETTER THAN NO SIGHT

READ INSTRUCTIONS BEFORE OPERATING

ⓘ **INFORMATION:** This feature is only available for mill and plasma software versions.

SHERLINE PRODUCTS

INCORPORATED 1974

How to Restart or Jump to a Line in a Program on the MASSO G3 Touch Control

Resuming a Program or Jump-to-Line for the G3 Touch Control

When machining complex jobs such as die molds or complex engraving jobs, which can take a long time to machine, power failure or broken tools mean that you need to start the entire g-code file from start. Restarting the g-code file from start is very time and resource consuming. To be able to effectively restart your machining process from where you left, MASSO has a **Resuming Program or Jump-to-Line** feature (from software v3.30).

With this feature, the current g-code line number is saved into a high-speed internal memory and even if the machine is powered off while machining, MASSO will automatically store that last line number in memory.

For G3 Touch Control

1. Power off/on

2. Home machine.

3. Go to the F6 screen and load the program.

4. Go to the F2 screen. Click on "**Jump to line**".

5. When you click on "jump to line," the **Resume Program** page will come up (see Figure 1).

6. Now you have two options depending on your reason for restarting the program.

  A. In the case of a power-off situation while the machine was running.

   1. If this is a power off situation. This page will show the line number when the machine stopped in the "**Line No.**" box. You click on "**Start from Line**" and all of the position, feed rate, and spindle RPM information will show.

   2. Check all of this information to be sure that it is accurate.

   3. If the information is good, click on "RUN". The machine will move in "**Single Block Mode**" to the X and Z positions. You will need to click on "**Cycle Start**" for each axis move. Then it will move down in the Z-axis and continue on with the program.
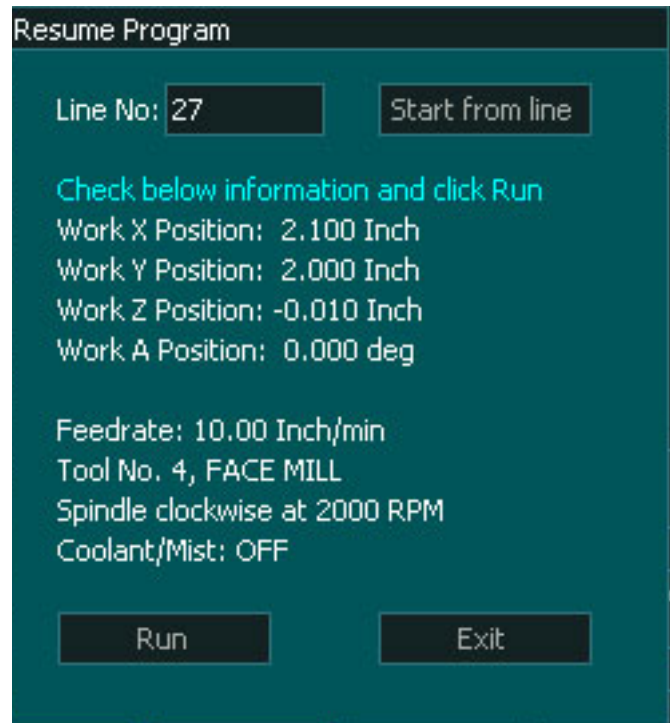

FIGURE 1

  B. If you are restarting a program and you want to skip ahead to a specific tool or line:

   1. First go to the F6 screen and load the program.

   2. Click on "**Edit**" to bring up the program edit page.

   3. Page down to the tool that you want to start on and write down the program line number that is on the side of the page (see Figure 2). This will generally be the line that has "**T— M06**." This way you will be starting with the tool change and all of the axis position and spindle speed information will be in the following lines in the program.

SHERLINE PRODUCTS INC. • 3235 Executive Ridge • Vista • California 92081-8527 • FAX: (760) 727-7857
Toll Free Order Line: (800) 541-0735 • International/Local/Tech. Assistance: (760) 727-5857 • Internet: www.sherline.com

5/12/22

FIGURE 2—The red bracket shows the program line numbers.

4. Exit out of the program edit page.

5. Load the program again.

6. Go to the F2 screen.

7. Click on "**Jump to line**."

8. Now enter the line number that you want to start on in the "**Line No**." box.

9. Then click on "**RUN**."

10. Click on "**Cycle Start**" to "single block" step your machine into position, and the program will begin running the program from that position.

Thank you,
Sherline Products Inc.