



## **MASSO Program Header, Tool Change, and End of Program**

No need for (%) or program numbers at the beginning of the program. You can add notes or the program name or number in parentheses if you desire. Anything that is inside the parentheses, the computer will not read. They are just program notes for you.

G90 G20 (Absolute Positioning - Program units: inch)

T2 M6 (Tool Change)

M3 S1200 (---Start Spindle - CW – 1200 RPM)

G90G00X0Y0

G0 Z0.1000 (Go to Z clearance height)

(CHANGE TOOLS)

NOTE: In order to change tools, all you do is move your tool to a clearance position, then enter the tool change command “T\_ M6.” There is no need to incorporate “Tool Length Compensation,” or to cancel it. No G43 or G49 is needed. Upon the tool change command, the tool will move to a change position or machine home. The screen will ask you to change the tool, then hit the cycle start button. As soon as you hit cycle start, the new tool information is live for that tool at that position. The computer keeps track of every move and incorporates all offset changes.

G0 Z0.100 (Go to Z clearance height)

T3 M6 (Tool Change – The tool will move to either the tool change position or Machine Z Zero)

M3 S2000.0 (---Start Spindle - CW – 2000 RPM)

(ENDING)

G0 Z0.1000 (Go to Z clearance height)

M5 (Spindle Stop)

G0 G53 Z0 (Rapid move to Machine Zero in the Z-axis)

G0 G53 X0 Y0 (Rapid move to Machine Zero in the X and Y-axis)

M30 (End of program)

Thank you,  
Sherline Products Inc.