

**Title:** Compass Manual Restart**Product(s):** MP3xxxiec, Compass 1.x**Doc. No.** AN.MWIEC.08

### **Compass Manual Part Restart**

The purpose of this application note is to outline procedures for executing a manual program restart in Compass for both additive and subtractive machine types. The method of executing the manual restart will depend on the application and tools available to the operator.

Restarting is used when a machine cycle is aborted before completion. This may occur due to power failure, tool breakage, running out of filament, or some other failure while a file is in progress.

### **Rerun unmodified G Code File**

The simplest case of a manual program restart in Compass is when you can just reset and click Cycle Start again. For example, if only five percent of the file had been executed then it may be acceptable to just restart the file from the beginning and have the machine rerun the previously executed commands.

For a subtractive (cutting) process, this would mean that the machine cuts in air until it reaches the actual desired restart location, at which point the cutting resumes. If the part program begins in relative positioning mode or uses the current position as a work offset, the user must manually position the machine to the same start location, otherwise the machine operation will have a different offset and will not resume at the correct position.

For an additive process where the extruder or other tooling would contact the existing part, the user would need to remove the part and start from the beginning.

#### **Step-by-Step:**

1. If motion is still running, click Feed Hold
2. Click Reset to cancel the G Code file
3. Complete any operation required before restart
4. Click Cycle Start in Auto Mode to start the file from the beginning

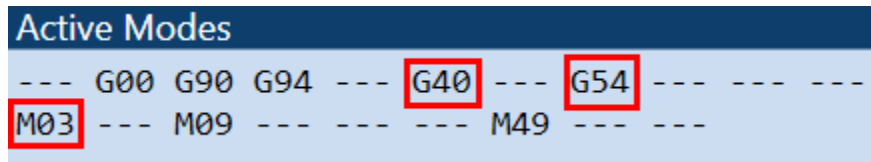
### **Restart with Modified G Code File**

Duplicate the G Code file and remove all segments that have already been completed. It is important to make note of modal states, spindle settings, and any other machine settings such as temperatures and offsets when the machine stops so that these states can be restored in the modified G code file. Add these settings to the beginning of the modified file so that the machine state is properly restored before resuming normal operation. Note that relative motion

commands may produce incorrect motion if the machine position is not properly positioned when the g-code program is resumed. It may be necessary to either jog the machine to a safe approach position or add a safe motion routine to position the tool in such a way that it does not crash into the existing work piece before moving to the position required to resume operation. It is recommended to reduce the feed rate override until the operator is satisfied that the program is operating as intended.

#### Step-by-Step:

1. If the machine did not abort by itself, the user should feed hold the machine
2. Note all machine conditions such as modal states, spindle settings, temperatures, etc.
  - a. Example: M03, G40 and G54 must all be noted and restored at the start of the edited G Code file.



Active Modes

```

--- G00 G90 G94 --- G40 --- G54 ---
M03 --- M09 --- --- M49 ---
  
```

3. Click Reset and cancel the G Code file and clear any errors.
4. Duplicate the G Code file and open in a text editor.
5. Delete all G Code that had already been completed.
6. At the start of the new file add the G Codes required to properly restore the last state of the machine (G54 Coordinate System, M03 spindle on, G40 no cutter compensation, any local offsets).
7. Add a restart motion script to the start of the file for approaching the restart position safely and set the feedrate, spindle speed, and modal states.

```

// Manual Restart Script
G55
G1 Z50 F600 // Move Z to safe location
G1 X50 Y50 // Move XY to local zero offset location
G92 X0 Y0
S18000 // Set Spindle Speed RPM
M3 // Turn on Spindle
F500 XY // Set feed rate for XY
F250 Z // Set feed rate for Z
G40 // Turn off cutter compensation

// Restart Location
G1 X-9.222782 Y-6.112924 Z-0.236220
G1 X-9.225317 Y-6.114855 Z-0.236220
  
```

- a.
8. Users may want to reduce the feed rate override to an acceptable speed before starting the edited G Code file.
9. Load the edited G Code file into Compass and Cycle Start.

### Modification of G Code Post from CAM Software

Using the CAM software, the user can remove completed tool paths and re-post the G Code file. For example, in Fusion 360 the user can simply select the tool path operations that need to be executed, beginning with the failed operation, and re-post the file. In Figure 1 below from Fusion 360, if the tool broke in the "2D Pocket3" operation, choose all the operations from there to the end and have the software post the G Code file for only those operations. This way any parameter settings, such as offsets and spindle settings, can be restored from the freshly posted G Code file. The new file would then be loaded into Compass and the user hits Cycle Start for the restart to initiate. The partially completed toolpath will cut air until the point where the prior cycle had stopped, but completed toolpaths do not need to be re-run, so little time is lost.

Another benefit of this method is that an M01 Optional Stop may be added at the end of each operation. The operator may then stop the machine at the end of an operation. To resume, the operator either continues from the optional stop, or if the machine is reset, the user would re-post the file starting at the next operation, and restart at a known location in the operation while starting at the beginning of a G Code file, rather than somewhere in the middle of the file.

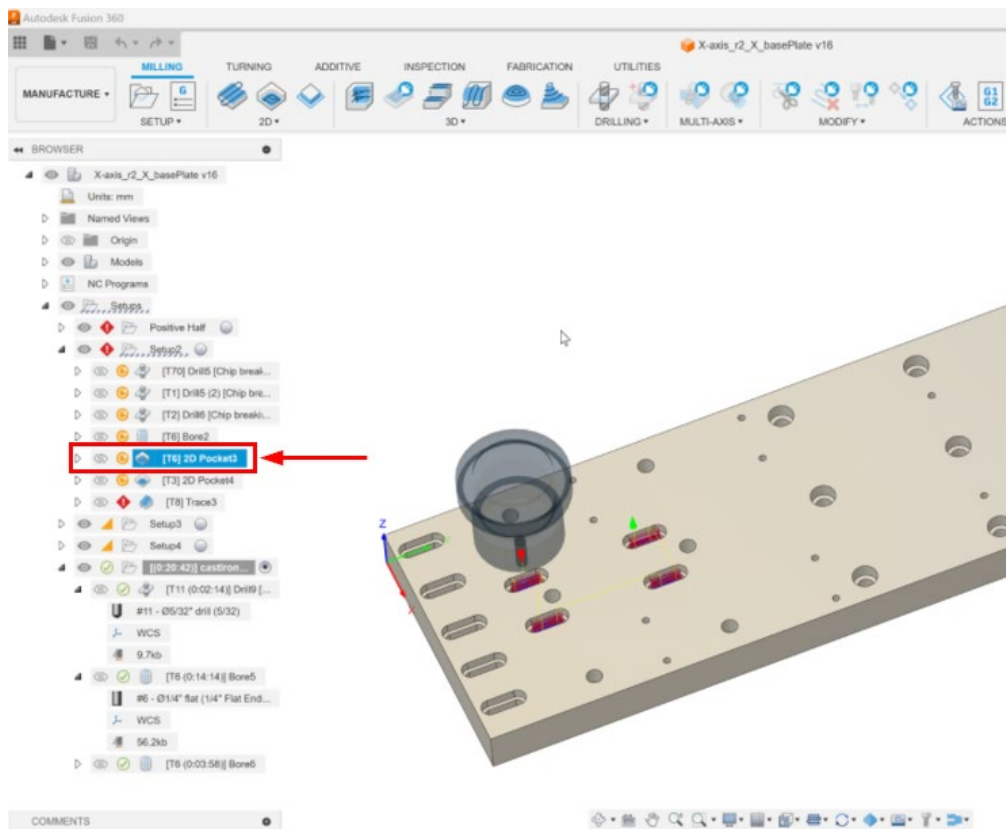


Figure 1: Selection of 2D Pocket3 and operations thereafter to post a G Code file with only the operations left to be completed.

### Posting Multiple G Code Files

Posting each operation in a subtractive process as a stand-alone G code file creates a set of fully-formed G code programs that can be run individually or as a batch using the “Open Multi-File List” function. The advantage of this workflow is that each step of the process is captured in a smaller file that properly sets the machine state. Should the process be interrupted for any reason, the operator may start at the beginning of the last operation started, minimizing lost time and potential errors from modifying the files by hand and uncertainty of the machine state.

#### Step-by-Step:

1. Create G Code files for all operations that still need to be completed
2. Open the Multi-file List from the Program Display in Compass
3. Add all G Code files to the list
4. Select the first G Code file in the list and click the green checkmark
5. Click Cycle Start
6. Once the first file is completed the Multi-file list will automatically open with the next file selected. Click the Green Checkmark and Cycle Start again.
7. Repeat until all files in the Multi-file list are completed.

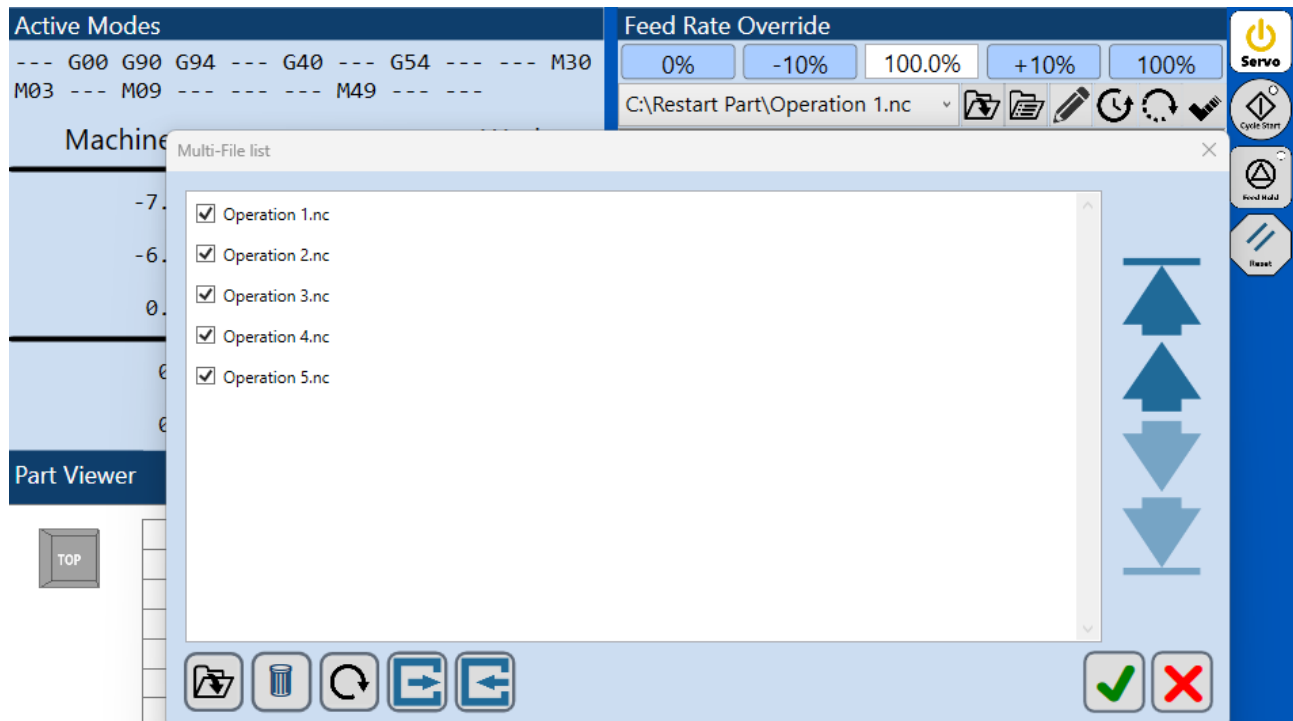


Figure 2: Use of the Multi-File List function from Program Display to restart multiple operations

**Recovery from power loss**

Recovery from power loss may be the most difficult of situations due to the machine operator not being able to observe the current state of the machine in Compass once the machine has stopped. Utilization of an uninterruptable power supply is suggested on both additive and subtractive machines. The presence of a UPS will allow the operator to capture things such as current modal states from Compass, the last line executed from Compass and the position of the machine to assist with the restart procedure of their choosing, such using one of the methods described in this application note.