

ME 363 [Lab #6] CNC Machining I

This week's laboratory assignment involves using the CAM software (Mastercam) to create part programs and post-processing the programs (CAM-CNC integration). The purpose of this exercise is to show how part programs can be automatically generated by a CAM package and how the information can be processed and transferred to a specific CNC. In general, a CAM package can generate part programs for many different CNC's. The post processors included in Mastercam support Fanuc, Fadal, Haas, and Emco. Others can be added if needed. Mastercam has been installed on the computers in ME28 and ME1030. From the desktop, go to Start → Programs → Mastercam and then click on Mastercam. When the software is ready, proceed as described below. You can work as a team; however, individual participation is very important.

1. Follow mill tutorial #2 in the Mastercam manual and create a part program. This tutorial will guide you through the steps to create geometry files and part programs.
2. Once the geometry and/or part program is created, you must store it on your network drive space or personal storage media. To save the file, click the File menu and highlight "Save as". When the submenu is displayed, type in the file name.
3. Estimate the total machining time by clicking on the "Backplot" button, and then selecting the "Info" tab as shown in Figure 1. If the backplot tabs are not visible, click on the upper left icon (with two blue down arrows) to expand the backplot dialog box. Write down the number or print out the screen.
4. Post process the part program. Post processing will create ISO or EIA codes in an ASCII format for the CNC you specify. Execute the CNC post processing. Next exit Mastercam and print out the generated .NC file.
5. Since you have mastered the milling part of Mastercam by now, you can repeat steps 1–4 above with lathe tutorial #1 to work on turning.
6. Write a short report with the printouts generated in steps 4 and 5.

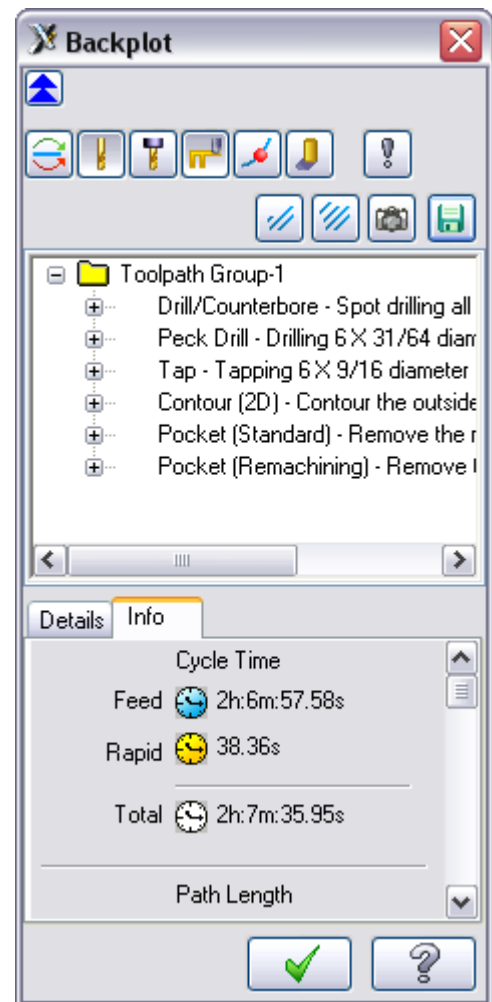


Figure 1: Total cycle time displayed in Backplot Info tab.

General comments:

Post-processing converts a generic part program generated by a CAM package into a specific format required for a given CNC by using a "template". This template information is usually available from the CNC you have. Like many other CAM packages, Mastercam allows you to build your own post-processors.

File Designations:

- .MCX Part program source file
- .NC CNC data file
- .PST Post processor file

