



Basic CNC Lathe Class Outline

Prerequisite:	Basic machining experience and programming of other CNC lathes
Format:	Combination of classroom instruction with either a chalkboard or whiteboard with hands-on work on fully functional, powered on CNC lathes
Student Materials:	Safety Glasses At least 1 copy of the programming manual Caliper or micrometer Paper and writing tool
Instructor Materials:	Cutting tools Blank material Collet for worked examples in class

Schedule

Day One

8:00 to 9:30	Basic OmniTurn programming and program format. Nomenclature and rules. Differences between OmniTurn and machines in current use.
<i>Break</i>	
9:45: to 11:00	Review 10 different OmniTurn programs to understand differences
11:00 to 12:00	Machine safety is first! Review lathe safety and proper usage. Entering programs into the OmniTurn and basic machine functions
<i>Lunch</i>	
12:45 to 2:00	How to run a program for the first time (safely), setting a turning tool, drill, threading tool, boring tool, and adjusting offsets, live tooling
<i>Break</i>	
2:15 to 3:30	Roughing cycles G74, 75, 78; threading cycles; metric, multistart, tapered, cleanup pass. Write an additional program, either given by teacher or an example from end use requirements. Prep to run parts
3:30 to 4:45	Enter the last program, set a c drill, drill, and threading tool. Run program
4:45 to 5:00	Review tooling catalog and Q&A.

Day 2 (Optional)

Cover additional topics as requested.

Experience additional hands-on work writing programs, entering them into the control, machine set up and running.

Additional programming codes:

- Tool nose radius (TNR)
- Secondary offsets
- C axis programming
- Subroutines
- Looping
- PLC integration and programming

Review current programs, tooling, work holding, and methodology.

Maintenance, trouble shooting and service of machine. What needs to be done and how to do it.