Basic Non-CNC User Class Outline

Prerequisite: Basic machining experience

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 10:00 Watch instructor write a simple program, select tools, set up machine, and run program. Introduction of the nomenclature and theories to be covered in the rest of the class.

Break

10:20 to 12:00 Overview of machining technology and terms, including:
- Materials: Steel vs Aluminum vs Brass vs SS and more
- Processes: Drilling vs turning vs threading vs reaming and more
- Measurements: IPR vs IPM, CSF vs RPM: where do I get these numbers?
- Insert geometry
  - Turning vs boring
  - Front turning vs back turning
  - Threading inserts

Lunch

1:00 to 2:00 Work holding – if you don't hold it you can't machine it!!!!
- Collets vs Chucks
- Types of collets:
  - 5C standard
  - 5C dead length
  - 5C expanding, simple and added collar type
  - Expanded range
  - Soft emergency
- 5C accessories
  - Work stops, spring ejectors
  - Dead length spindle nose collars
- Chuck types
  - 2 jaw, 3 jaw
  - Pneumatic vs draw tube type

Break

2:20 – 3:30 Review type of work and operations performed. Review issues that affect the success of a new operator. Students will get a glimpse into what will be important over the next two days.

3:30 – 4:00 Safety first: Review lathe safety and proper usage

4:00 – 5:00 Begin programming class with first program

Day 2

8:00 to 9:30 Basic OmniTurn programming and program format. Nomenclature and rules.

Break

9:45: to 11:00 Write a program – Basic 1 tool program
G90, G72, G94, G95 M30, M08, M03/M05, T1, G10

11:00 to 12:00 Machine safety is first! Entering the program in the OmniTurn and basic machine functions

Lunch

12:45 to 2:00 How to run a program for the first time (safely), setting a turning tool and adjusting offsets

Break

2:15 to 3:30 Modify the first part to add a second tool, corner chamfer and corner radius
Modify the program to add a threading cycle and peck drill
G33, G83

3:30 to 4:45 Enter the last program, set a c drill, drill, and threading tool. Run program.

4:45 to 5:00 Homework assignment of a simple program for next day

Day 3

8:00 to 8:30 Go over homework, add new programming features if needed.

8:30 to 9:30 Enter the program, touch off tools, and correct offsets, run program

Break

9:50 to 12:00 Additional programming examples.
Boring vs turning
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-user's requirements, then prep to run parts

Lunch

1:00 to 2:00 Setup and run an end user's part, or run given program.

Break

2:15 to 4:00 Additional programming codes:
Tool nose radius (TNR), Secondary offsets, C axis programming. ETC

4:00 to 5:00 Question and answers, homework if 4th day is planned

Day 4 (Optional)

Work with end user's parts, tooling and programs
Additional worked examples supplied by instructor
Service, maintenance, and trouble shooting