

HAAS BASIC MILL PROGRAMMING COURSE OUTLINE

◆ SYSTEMS of PROGRAMMING

- A. Absolute (G90)
- B. Incremental (G91)
- C. Cartesian Coordinate System

◆ Program Structure

- A. Introduction
 - 1. Program title
 - 2. Part Zero
 - 3. First tool Selection
- B. Body
 - 1. Calling out tool offsets
 - 2. Manufacturing Processes
- C. End

◆ COMMON “G” & “M” CODES

- A. LINEAR
 - 1. G00
 - 2. G01
- B. CIRCULAR
 - 1. G02
 - 2. G03
 - 3. Use of I & J
 - 4. Use of R
- C. CUTTER COMPENSATION
 - 1. G40 CANCEL
 - 2. G41 CUTTER COMPENSATION LEFT
 - 3. G42 CUTTER COMPENSATION RIGHT
- D. MODE
 - 1. G90 vs. G91
 - 2. G20 vs. G21

E.CANNED CYCLES

1. Hole Manufacturing
 - a. Drilling
 - b. Peck Drilling
 - c. Chip Breaking
 - d. Tapping
 - e. Bolt Hole
 1. Circle
 2. Arc
 3. Angular

◆ MISCELLANEOUS FUNCTIONS

- A. M03
- B. M05
- C. M06
- D. M08
- E. M09
- F. M30
- G. M31
- H. M33
- I. M42
- J. M97
- K. M98
- L. M99

◆ PROGRAMMING EXERCISES

- A. LINEAR
- B. CIRCULAR (G17)
 1. Using R
 2. Using I & J

- C. Hole Manufacturing
 1. Drilling
 2. Tapping