

HAAS COURSE OUTLINE LATHE BASIC PROGRAMMING

I. Power up defaults

- A.) G00 - Rapid
- B.) G40 - Cancel Cutter Compensation
- C.) G80 - Canned Cycle Cancel
- D.) G97 - Constant Surface Feet Cancel
- E.) G99 - Feed per Revolution

II. Program Structure

- 1.) Introduction
- 2.) Body
- 3.) Ending

III. Stock Removal

- 1. Rough Turning / Boring
 - A.) G71 Roughing
 - 1.) D - Depth of Cut
 - 2.) P - Starting Block of Path
 - 3.) Q - Ending Block of Path
 - 4.) U - X axis Finish Stock amount
 - 5.) W - Z axis Finish Stock amount

2.) Finishing

A.) G70

- 1.) P - Starting Block
- 2.) Q - Ending Block

IV. Drilling

- 1.) Drilling - G81
- 2.) Peck Drilling - G83

V. Tool Nose Radius Compensation

- 1.) G41 - Left
- 2.) G42 - Right
- 3.) G40 - Cancel Tool Compensation

VI. Threading

- 1.) G76 - Multiple Pass Threading
 - A.) D - First Depth of Cut
 - B.) K - Thread Height
 - C.) Z - Finish Length of Thread
 - D.) F - Feed Rate (Pitch of Thread)
 - E.) Use of M24