CNC Programming Assignment: “Dot-to-Dot 9-Line Programs”

Learning Goals: Read and write basic Fanuc-style G-code using only the codes on the sheet. Use absolute programming (G90) with G54, G17/G18, G00 rapid, G01 feed, offsets, spindle, and coolant.

Part A — Quick Checks (10 pts)

1) What does G54 do?

2) Difference between G00 and G01?

3) What plane does G17 select? What about G18?

4) Why do we turn on G43 H1 (or tool offset) before Z-moves?

5) What is the difference between S (RPM) and F (feed)?

Part B — Mill, 9-line “Dot-to-Dot” (45 pts)

Write a 9-line program that starts safely, connects four XY points in order at Z = –0.10 in, feed = 5.0 ipm, then rapids home and ends.

Points (in inches): P1 (0.5, 0.5) → P2 (3.0, 0.5) → P3 (3.0, 2.0) → P4 (0.5, 2.0)

Template:

```
% O1 (MILL DOT-TO-DOT) T1 M06 M03 S7500 G54 G90 G17 G43 H1 Z? M08 G01 F5.0 Z-0.10 G01 X__ Y__ X__ Y__ X__ Y__ X__ Y__ G00 Z__ G00 X0 Y0 M30 %
```

Part C — Lathe, 9-line Straight Turn (35 pts)

Write a 9-line program using Tool 1 (T0101), G54, G18, coolant, spindle S1000 M03, face to Z=0.0, then turn the OD from X1.50 to X1.00 over Z–2.000, feed .006 ipr.

```
% O2 (LATHE STRAIGHT TURN) T0101 G97 S1000 M03 G54 G90 G18 G00 X__ Z__ M08 G01 F.006 Z0.0 G00 X1.60 G00 Z0.1 G01 X1.00 G01 Z-2.000 G00 X__ Z__ M30 %
```

Part D — Stretch Challenges (Extra Credit up to 10 pts)

1) Mill: Add two more XY points to draw a closed shape (≤12 lines). 2) Lathe: Add a short chamfer on the front edge (≤12 lines). 3) Explain how you would convert your mill code to incremental (G91) and why that might be safer for beginners.

Rubric Summary (100 pts)

Part A	10 pts
Part B	45 pts

Part C	35 pts
Extra Credit	10 pts
Total	100 pts