Introduction to G-Code Programming

Computer Integrated Manufacturing

CNC Machining

# In this lesson:

Review Coordinate Geometry Basics

Identify common Terminology

Examine G and M - Code language

Provide opportunities for Review and Practice

Rectangular Coordinate System
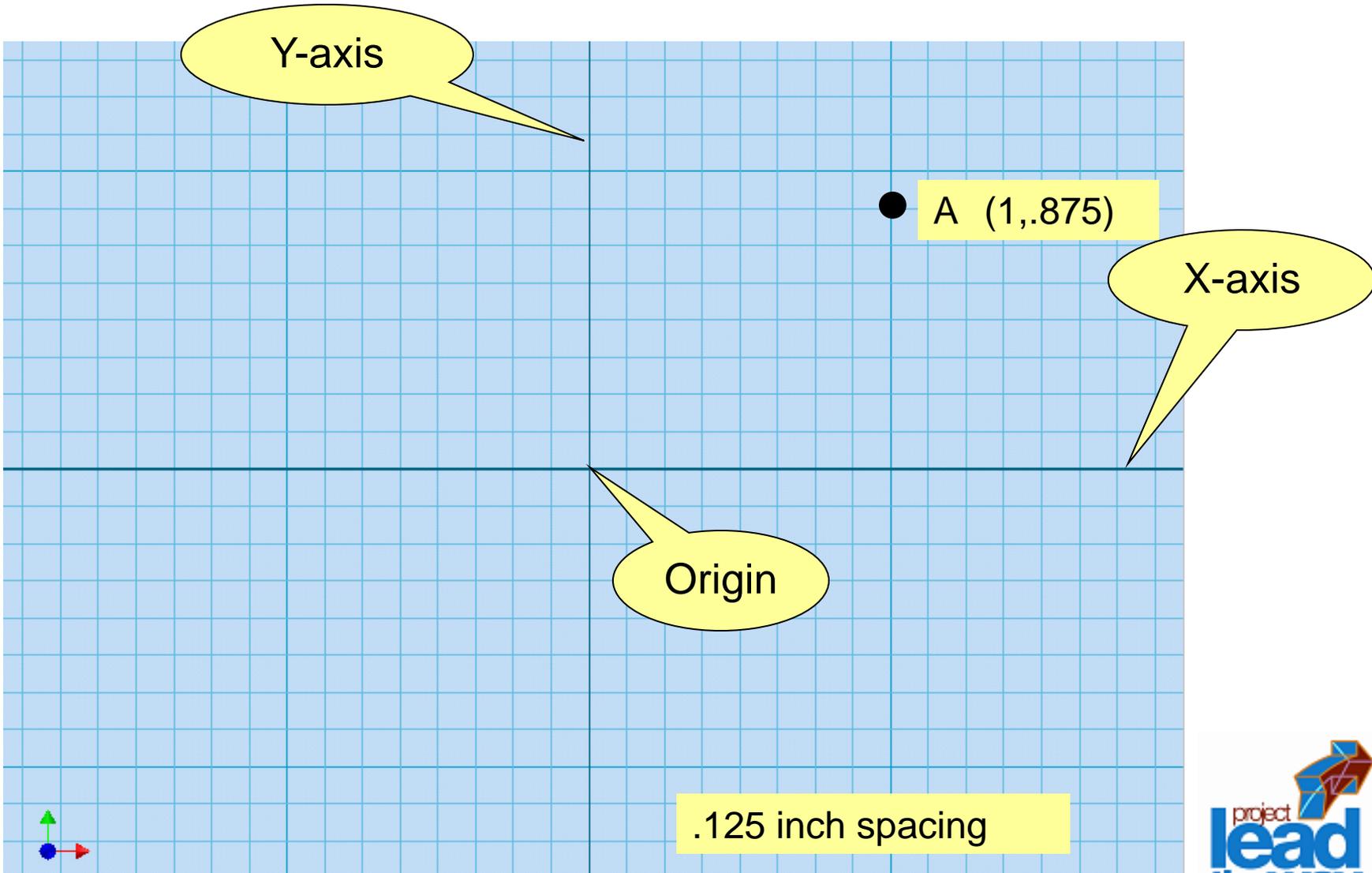
Ordered Pairs
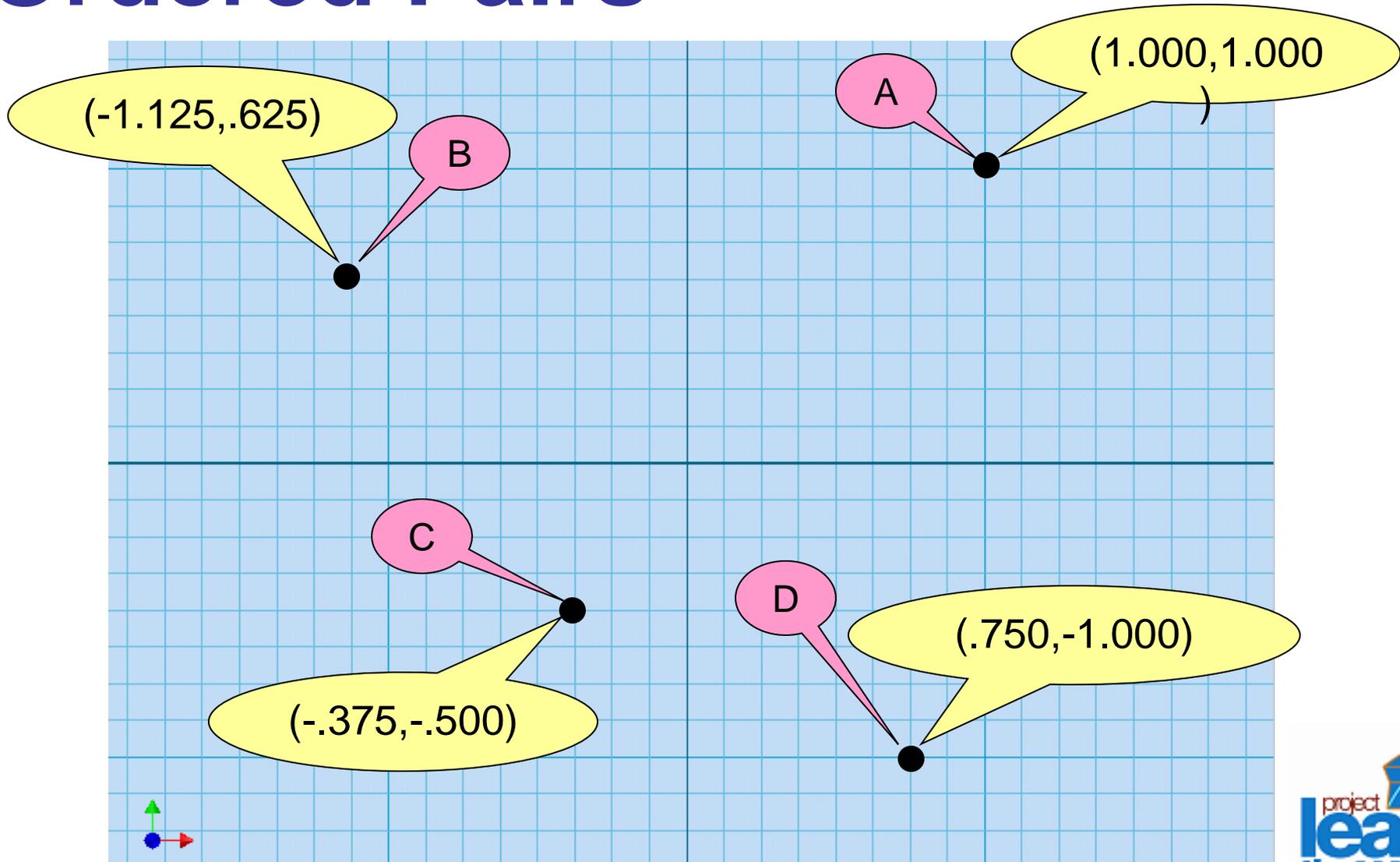
3D Coordinate System (X,Y,Z)
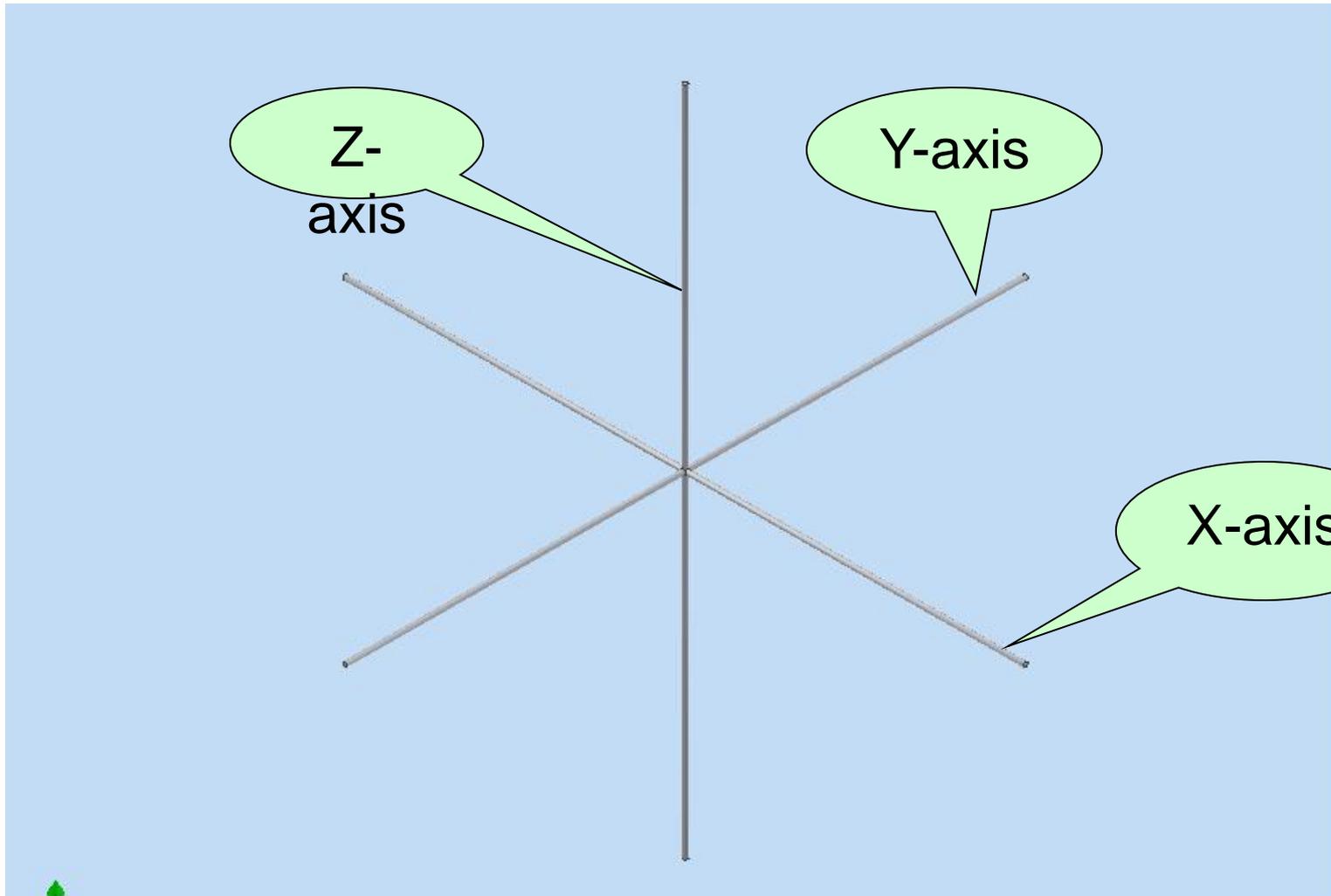
Basic Machine Axes: 3 axis

Milling Machines: 3 axis

- X – axis (table left and right)
- Y – axis (table in and out)
- Z – axis (usually the spindle axis)
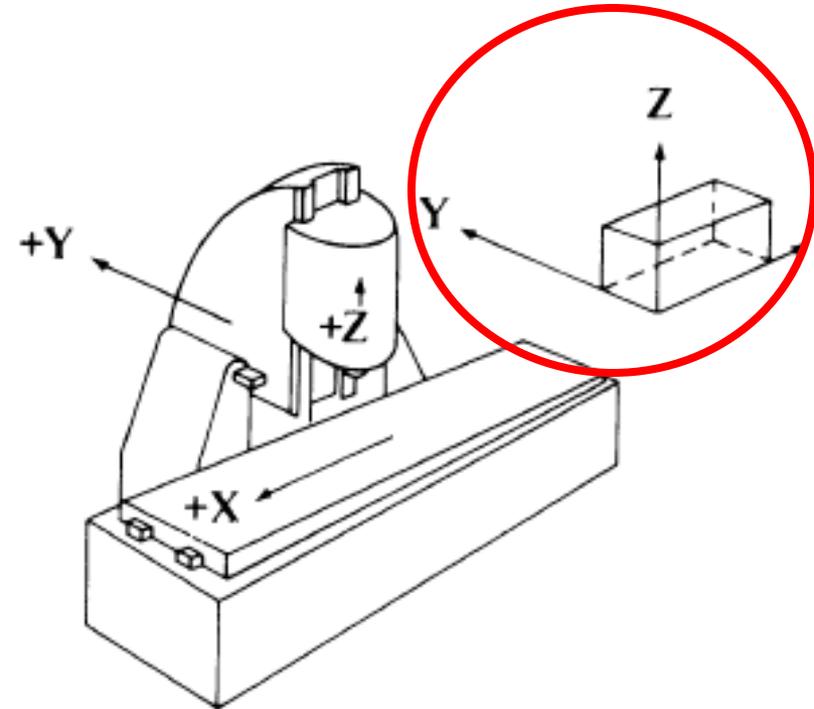
Additional Axes

A – axis (angular axis about X - axis)

B – axis (angular axis about Y – axis)

C – axis (angular axis about Z – axis)

U – axis (secondary axis parallel to X)

V – axis (secondary axis parallel to Y)

W – axis (secondary axis parallel to Z)

Milling Machines: 4 axis
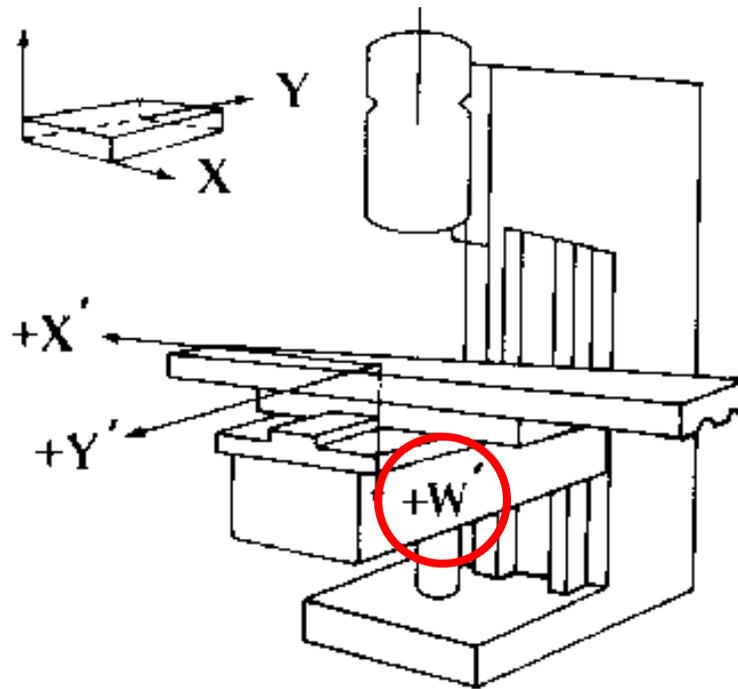
Terminology

NC – Numerical Control

CNC – Computer Numerical Control

DNC – Direct Numerical Control

APT – Automatic Programmed Tool

CAD – Computer Aided Design

CAM – Computer Aided Manufacturing

CIM – Computer Integrated Manufacturing

Download Code Sheet
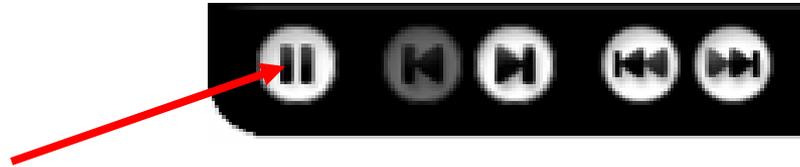
[Click here to open Code Sheet](#)

G - Code Programming

G – Code Programming

Originally called the "Word Address" programming format.

Processed one line at a time sequentially.

Common Format of a Block

Sequence #   Preparatory Function   Dimension Words   Feed Rate   Spindle Function   Tool Function   Misc. Function

N50   G90 G01   X1.40Y2.25   F10   S1500   T01   M03

Individual Words
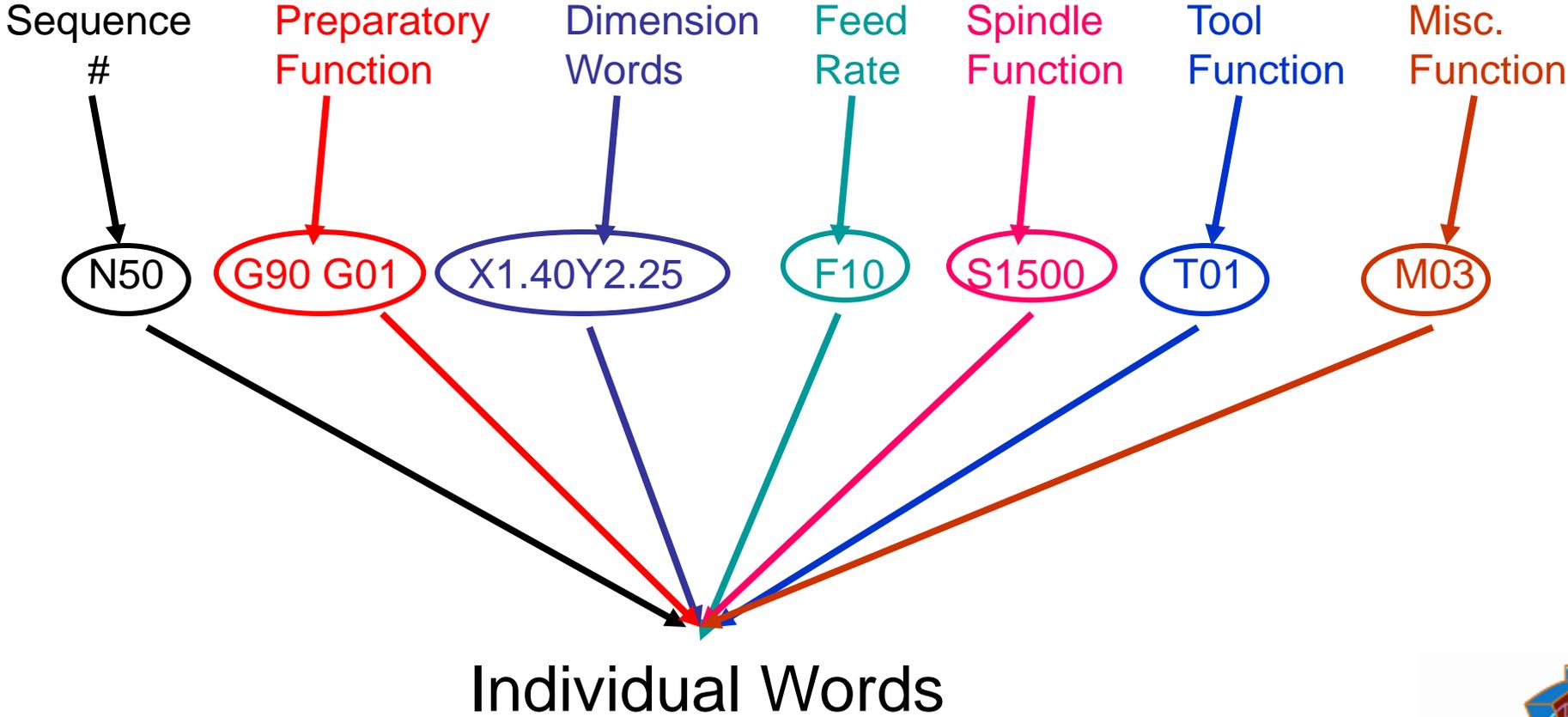
Word Address 1

N – Sequence or line number

A tag that identifies the beginning of a block of code. It is used by operators to locate specific lines of a program when entering data or verifying the program operation.

G – Preparatory function

G words specify the mode in which the milling machine is to move along its programmed axes.

Word Address 2

Dimension Words

    X – Distance or position in X direction

    Y – Distance or position in Y direction

    Z – Distance or position in Z direction

M – Miscellaneous functions

    M words specify CNC machine functions not related to dimensions or axial movements.

Word Address 3

F – Feed rate (inches per minute or millimeters per minute)

Rate at which cutting tool moves along an axis.

S – Spindle speed (rpm – revolutions per minute)

Controls spindle rotation speed.

T – Tool number

Specifies tool to be selected.

Word Address 4

I – Circular cutting reference for x axis

J – Circular cutting reference for y axis

K – Circular cutting reference for z axis

G Word

G words or codes tell the machine to perform certain functions. Most G words are modal which means they remain in effect until replaced by another modal G code.

Common G Codes

G00 – Rapid positioning mode

    Tool is moved along the shortest route to programmed X,Y,Z position. Usually NOT used for cutting.

G01 – Linear Interpolation mode

    Tool is moved along a straight-line path at programmed rate of speed.

G02 – Circular motion clockwise (cw)

G03 – Circular motion counter clockwise (ccw)

Common G Codes, con.,

G17 – XY plane

G18 – XZ plane

G19 – YZ plane

G20 – Inch Mode

G21 – Metric Mode

G28 – Return to axis machine Zero (Home)

G Codes: G90, G91

G90 – Absolute Coordinate Reference

References the next position from an absolute zero point which is set once for the entire program.

G91 – Incremental Coordinate Reference

References the next position from the previous position.

G Codes: Canned Cycles

G80 – Cancel canned cycle

G81 – Drilling cycle

G83 – Peck drilling cycle

G84 – Tapping cycle

G85 – Boring cycle

G86 – Boring cycle

NOTE: A canned cycle stays in effect until cancelled by a G80.

Canned Cycles: G81

G81 – Drilling Cycle

Feed to depth, rapid return

Example of program code:

N35 G81 X.500Y.500Z-1.000 R.100 F1.50

N36 X1.000Y1.500

N37 X1.500Y2.000

N38 G80

Canned Cycles: G83, G84

G83 – Peck Drilling Cycle

Feed to an intermediate depth, rapid out, rapid back to just above previous depth, feed to next depth, rapid out, repeat until reaching full depth.

G84 – Tapping Cycle

This cycle creates internal threads in an existing hole.

NOTE: One *cannot* over-ride the feed rate.

Canned Cycles: G85, G86

G85 - Boring Cycle

Feed to depth, feed back out.

G86 – Boring Cycle

Feed to depth, rapid out.

G Codes: Cutter Compensation

G40 – Cancel cutter diameter compensation.

G41 – Cutter compensation left.

G42 – Cutter compensation right.

M Word

M words tell the machine to perform certain machine related functions, such as: turn spindle on/off, coolant on/off, or stop/end program.

Professional Development ID Code: 6006

Common M words

M00 – Programmed pause

Automatically stops machine until operator pushes a button to resume program.

M01 – Optional stop

A stop acted upon by the machine when operator has signaled this command by pushing a button.

M02 – End of program

Stops program when all lines of code are completed. Must be last command in program.

Common M words

M03 – Turn spindle on

   In clockwise direction

M04 – Turn spindle on

   In counter clockwise direction

M05 – Stop spindle

   Usually used prior to tool change or at end of program.

M06 – Tool change

   Stops program and calls for a tool change, either automatically or manually.

Common M words

M08 – Turns Accessory 1 on.

M09 – Turns Accessory 1 off.

M10 – Turns Accessory 2 on.

M11 – Turns Accessory 2 off.

M30 – End of program

Similar to M02 but M30 will also "rewind" the program. Must be last statement in program. If used, DO NOT use M02.

Zero Points

Part Zero
- – Used for absolute programming mode.
- – Usually a position on the part that all absolute coordinates are referenced to.
- – Changes with different parts and programs.

Machine Zero or Machine Home Position
- – Fixed for each machine from the manufacturer.
- – Not changeable.

Cutter Path Generation

Cutter path is generated by moving the tool from point to point. The points are previously defined from the part drawing dimensions.

Each line of code will show the destination point of where the tool will go to.

Interpolation

Method of determining intermediate points along a cutting path.

Two methods:

- Linear interpolation – cut a path along a specified angle at a specified feed rate.
- Circular interpolation – cut a path along an arc or circle at a specified feed rate.

Axis movements: Caution!

Multiple axis movements are possible.

"Best Practice" is NOT to make a 3-axis movement using one line of code.

Move to position using two axes, X,Y; then move the Z with an additional line of code.

# Example 1: NC Block
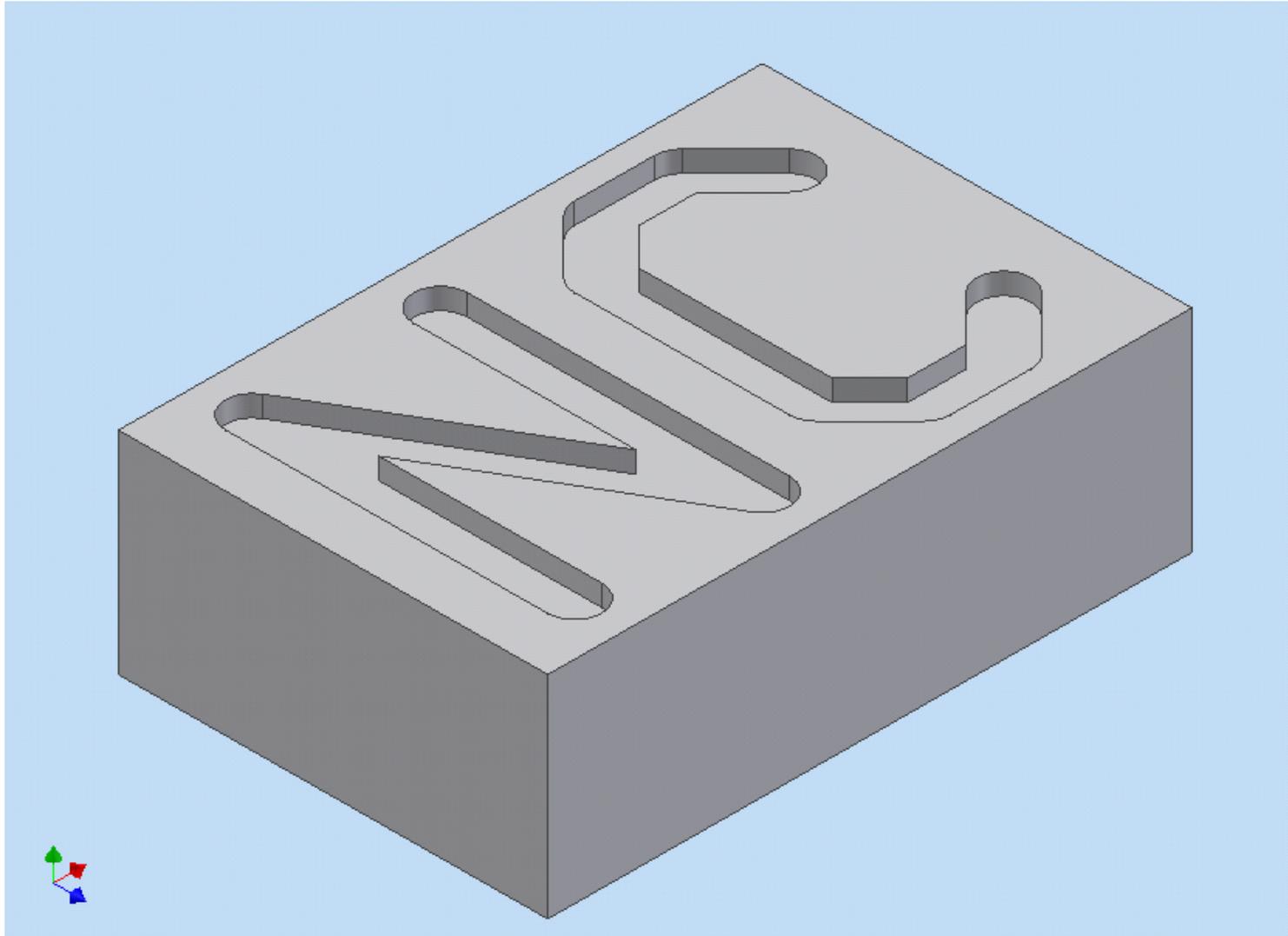
Top View NC Block

.125 GRID SPACES
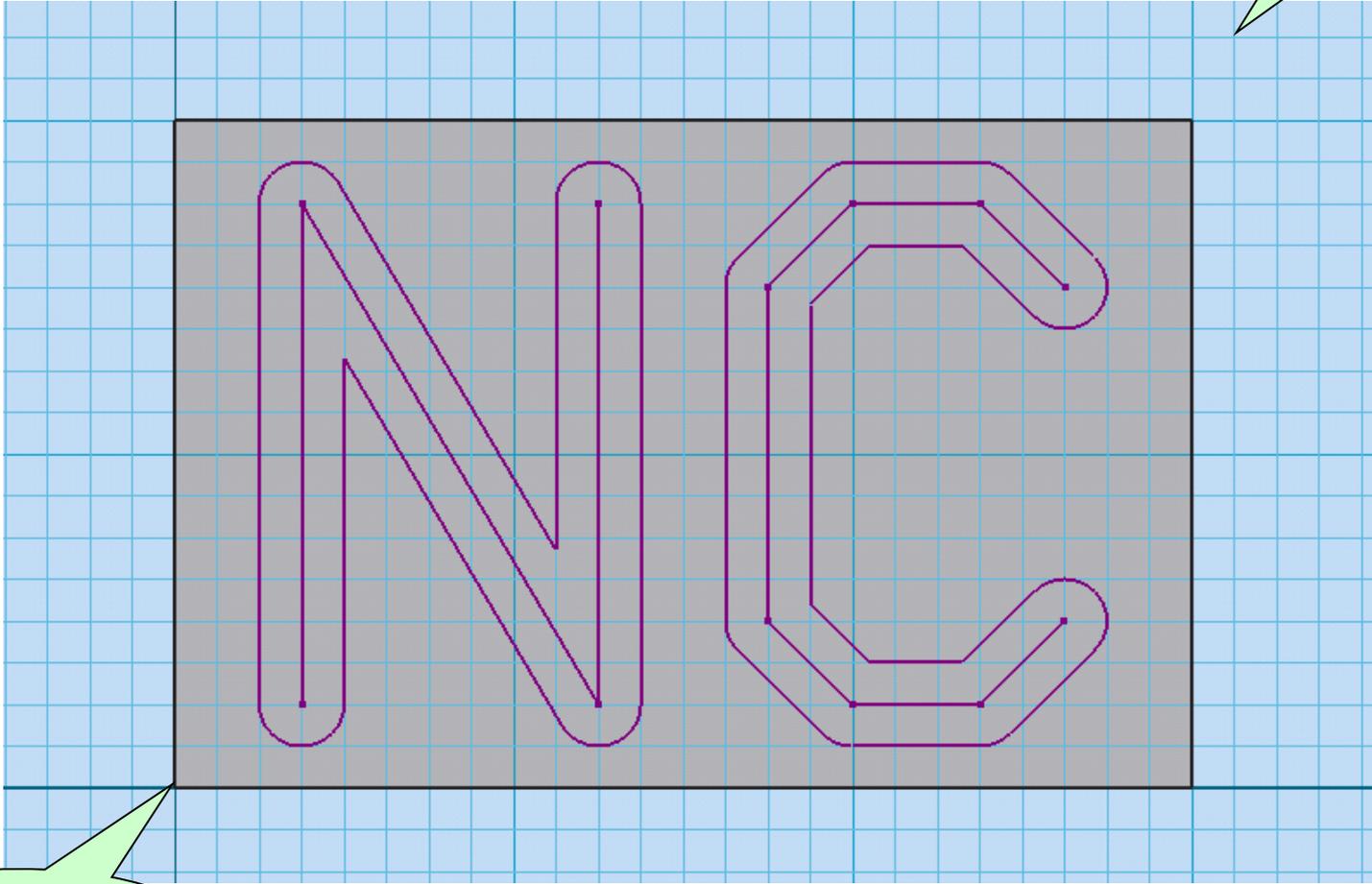
Origin
(0,0)

Download Worksheet
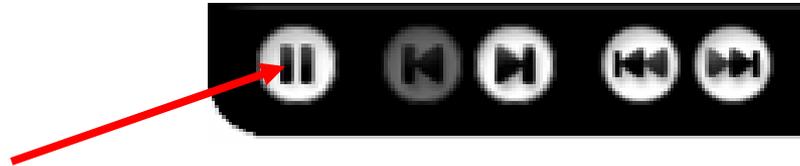
[Click here to open Practice Exercises](#)

Worksheet Problem 1

.125 GRID SPACES

B(,)

D(,)

I(,)

J(,)

K(,)

L(,)

A(,)

E(,)

F(,)

C(,)

H(,)

G(,)

Origin (0,0)
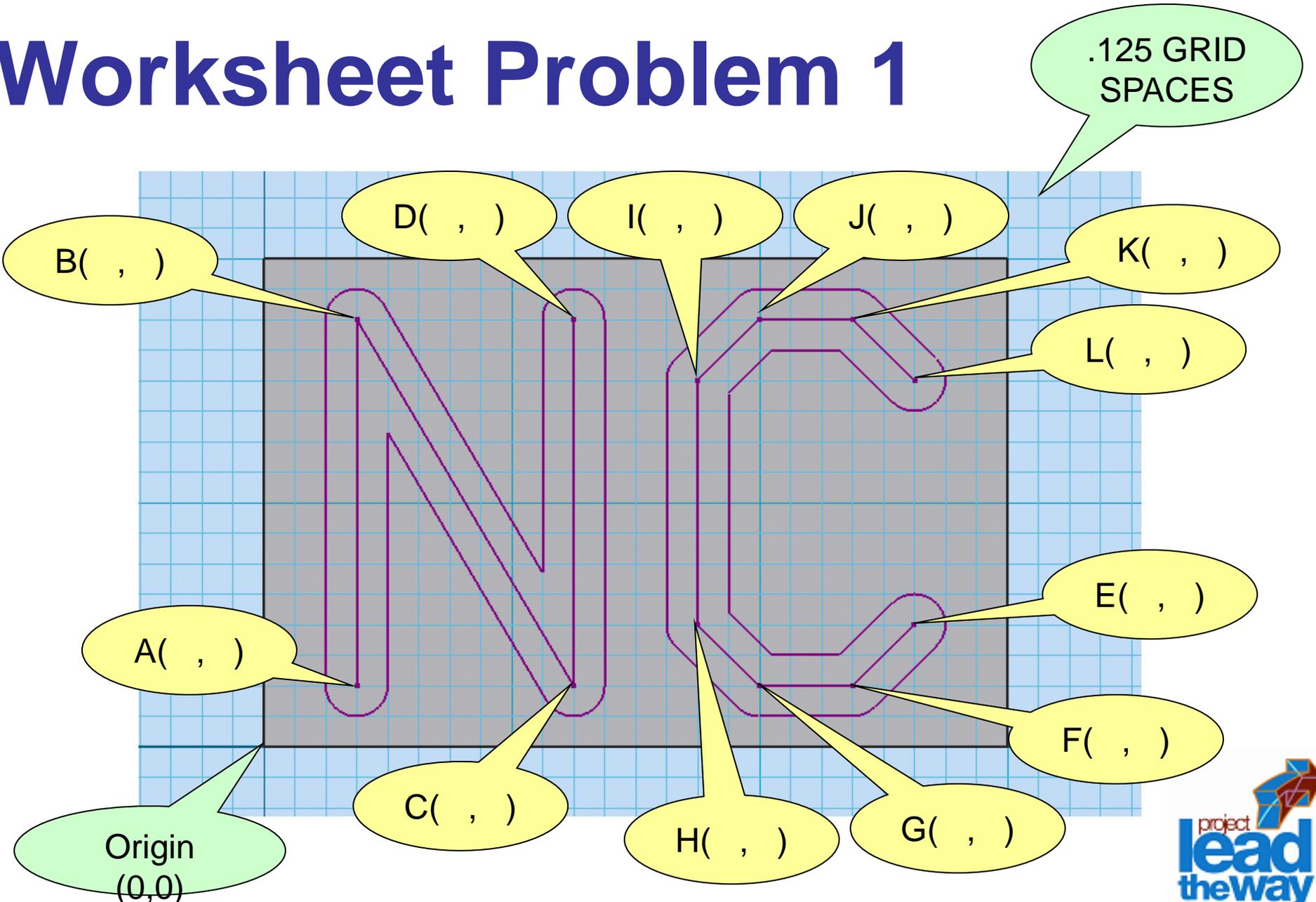
Pause Lesson
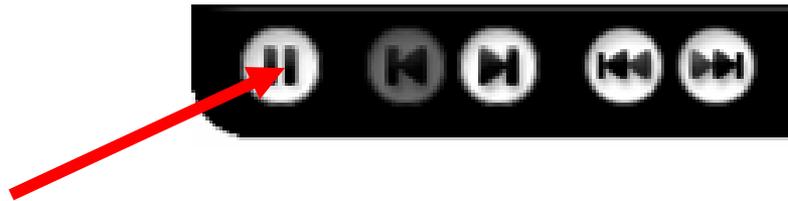
Example 1: Program NC

N01 G90 G80 T00

N02 G00 X0 Y3.000 Z1.000

N03 M03 S1000

Example 1: Program NC

N01 G90 G80 T00

N02 G00 X0 Y3.000 Z1.000

N03 M03 S1000

N04 X.375 Y.250 Z1.000

N05 Z.100

N06 G01 Z-.100 F5.00

N07 Y1.750

Example 1: Program cont'd

N08 X1.250 Y.250

N09 Y1.750

N10 G00 Z.100

N11 X2.625 Y.500

N12 GO1 Z-.100

N13 X2.375 Y2.50

N14 X2.000

Example 1: Program cont'd

N15 X1.750 Y.500

N16 Y1.500

N17 X2.000 Y1.750

N18 X2.375

N19 X2.625 Y1.500

Example 1, con.,

N20 G00 Z1.000

N21 X0 Y3.000

N22 M05

N23 M30

Example 2: PLTW Block
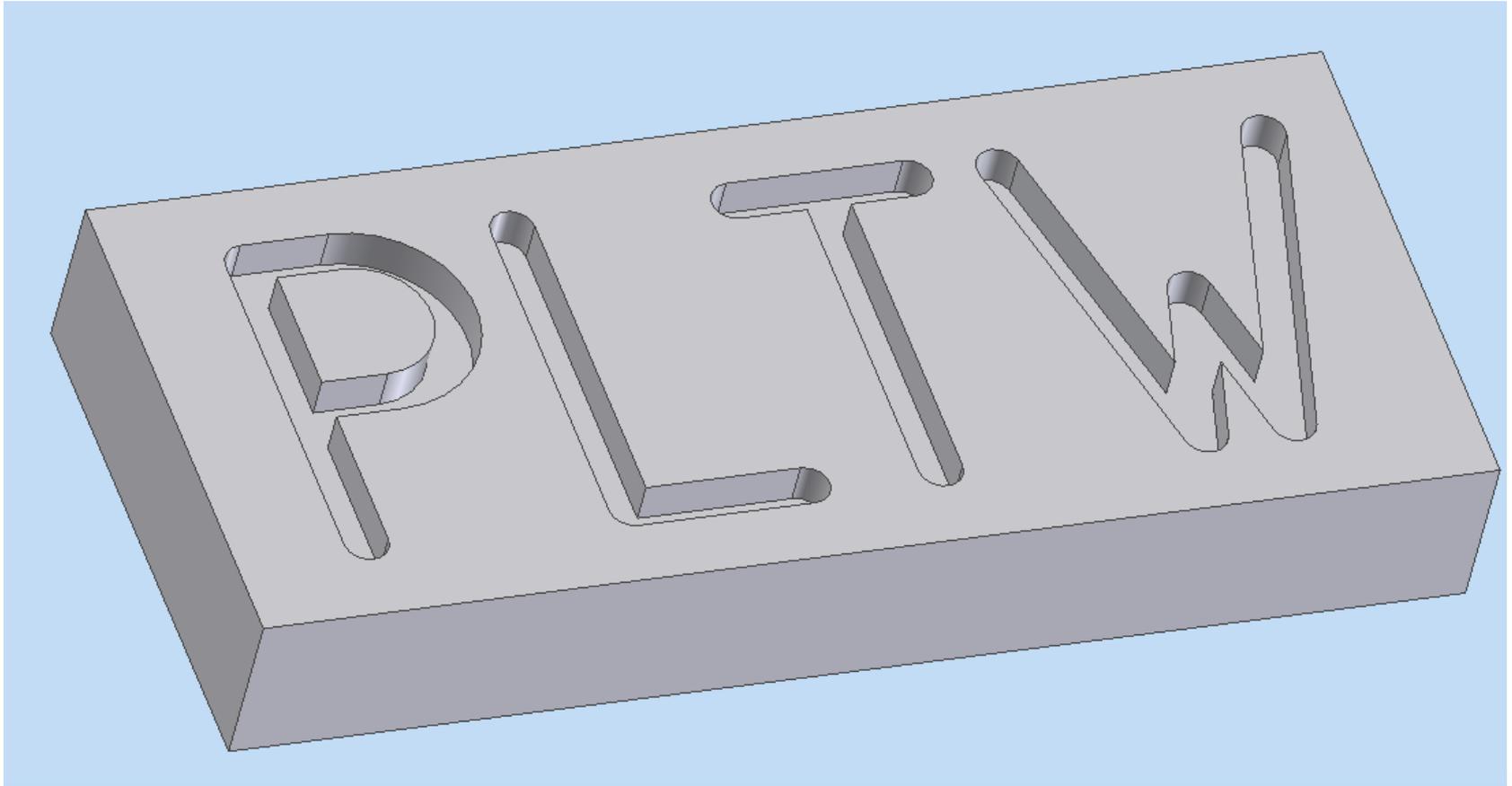
Worksheet Problem 2

.250 GRID SPACE
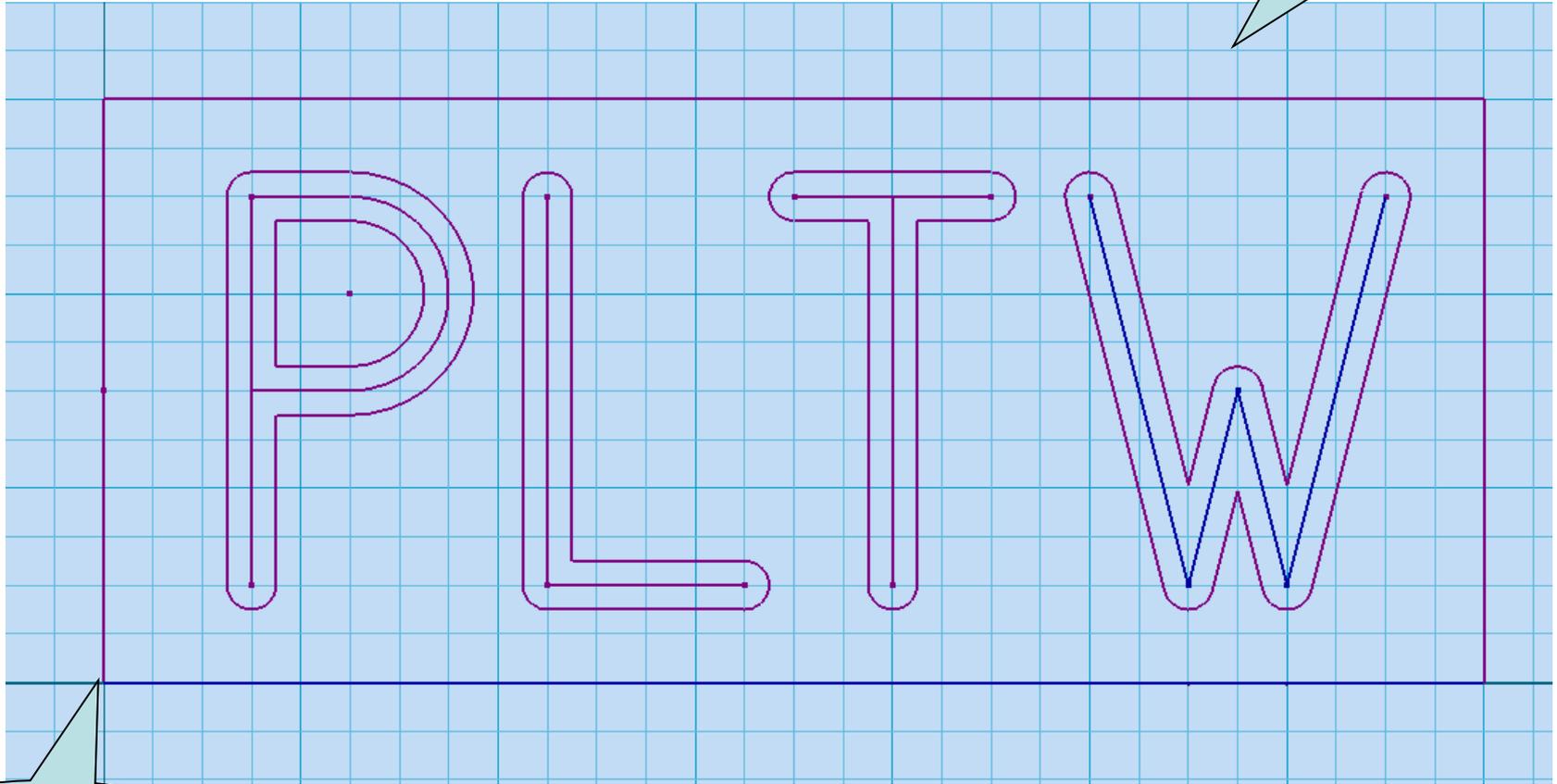
Origin

Example 2: Top View

M(,)

F(,)

J(,)

I(,)

K(,)

O(,)

C(,)

B(,)

Q(,)

E(,)

A(,)

D(,)

G(,)

H(,)

L(,)

N(,)

P(,)

Origin
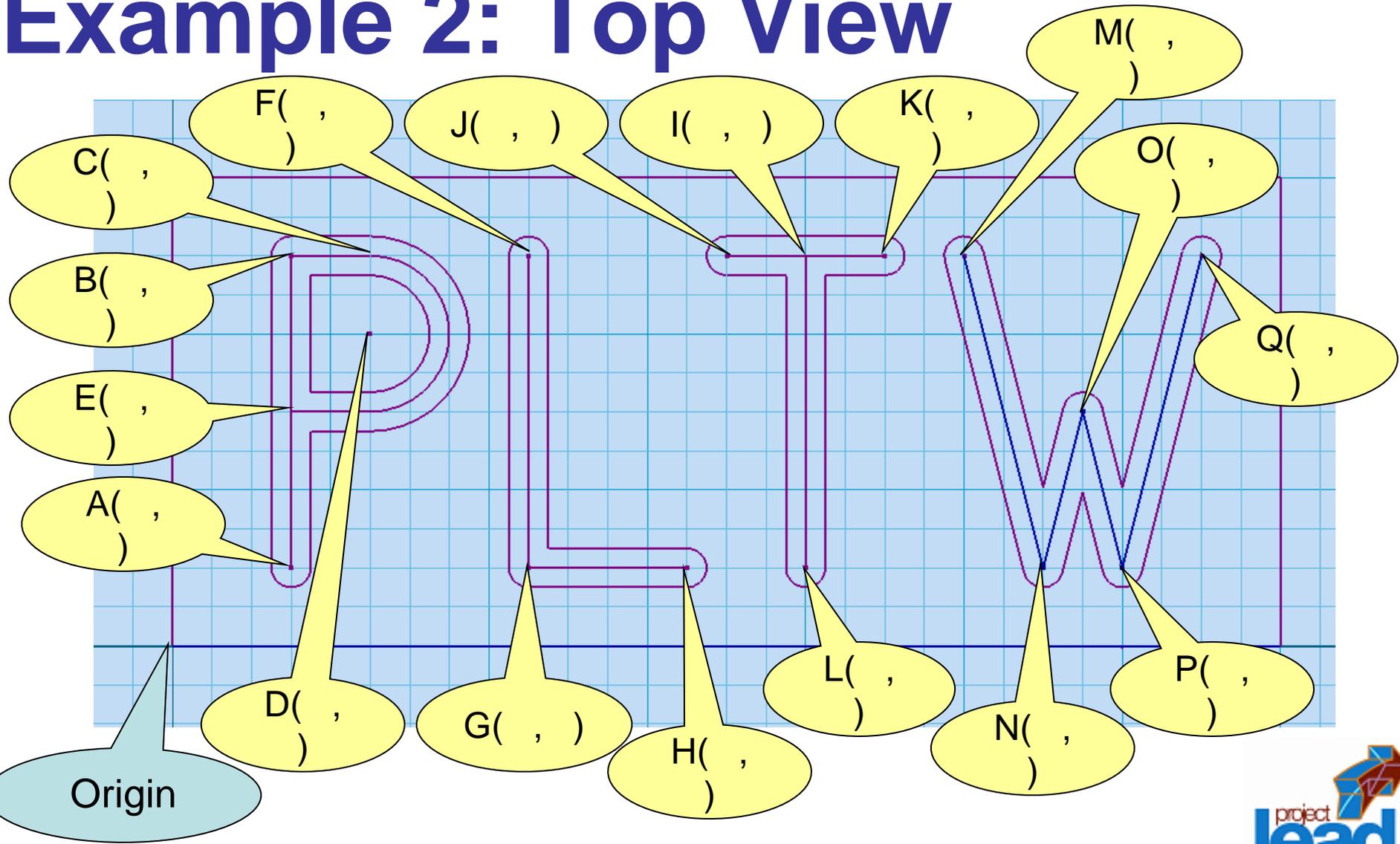
.250 Grid Space

Pause Lesson
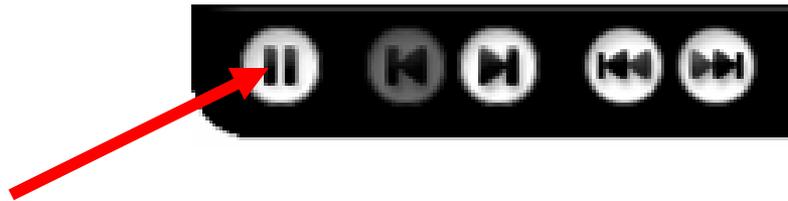
Example 2: PLTW

N01 G90 G80 T01

N02 G00 X0 Y0 Z1.000

N03 M03 S1000

N04 X.750 Y.500 Z1.000

N05 Z.100

N06 G01 Z-.250 F5.00

N07 Y2.500

Example 2: PLTW cont'd

N08 X1.250

N09 G02 X1.250 Y1.500 I1.250 J2.000

N10 G01 X.750

N11 G00 Z.100

N12 X2.250 Y2.500

N13 G01 Z-.250

N14 Y.500

Example 2: Program cont'd

N15 X3.250

N16 G00 Z.100

N17 X4.000

N18 G01 Z-.250

N19 Y2.500

N20 X3.500

N21 X4.500

Example 2: PLTW cont'd

N22 G00 Z.100

N23 X5.000 Y2.500

N24 G01 Z-.250

N25 X5.500 Y.500

N26 X5.750 Y1.500

N27 X6.000 Y.500

N28 X6.500 Y2.500

Example 2: PLTW cont'd

N29 G00 Z1.000

N30 X0 Y0

N31 M05

N32 M30

Curriculum Alignment:

Unit 2: CNC Machining

Section 2.3 – CNC Machining

PowerPoint – Introduction to CNC

PowerPoint – CNC Programming

References:

Oberg, E. & Jones F. D. & Horton, H. L. & Ryffell, H. H. (2000). *Machinery's Handbook, 26th ed.,* New York, NY: Industrial Press Inc.

Valentino, J.V. & Goldenberg, J. (2003). *Introduction to Computer Numerical Control (CNC), 3rd ed.*, Upper Saddle River, NJ: Prentice Hall

Credits:

Writer: Bob Arrendondo

Content Editor: Donna E. Scribner

Narration: Donna E. Scribner

PLTW Editor: Ed Hughes

Production: CJ Amarosa

Video Production: CJ Amarosa

Audio: CJ Amarosa

Project Manager: Donna E. Scribner