Course Outline

Precision Measurements for Machinist

Lesson 1: Basic Print Reading.

After completing this lesson the operator will be able to:

1 .Identify the following on a print:

a. multi-view print and explain its relationship to part features,
b. Workpiece features and their locations on a print.
c. A half section and full section view.
d. A detail view.
e. An auxiliary view.
f. A center line.
g. Both object lines and hidden lines.
h. An extension line.
i. A dimension line.
j. Leader line.
k. A cutting plane line and its reference notation.
l. A line.
m. A datum feature.
n. A co-axial and co-planar datum feature
o. A datum target feature.

2. Explain the following abbreviations and symbols on a print:

a. DIA, D, Ø

b. R, RIA, RAD

c. TIR, FIR, FIM

d. TYP, 2X, 3X

e. MAX

f. MIN

g. C, CFR, CHAM

h. Surface Finish Symbol

i. Counter Bore, Counter Sink, Depth Symbols

3. The Title Block and Tolerances.

a. Identify the Title Block.

b. Identify the Notes Area.

c. Explain Nominal Dimension

e. Identify Basic Dimensions.

f. Datum vs. Chain Dimensioning.

g. Find Missing Dimensions.

4. Explain the two common systems of measurement.

a. Inch and Metric System principles.

b. Elements of the Inch System.

c. Elements of the Metric System.

d. Visualize dimensions and estimate sizes.

Lesson 2: Geometric Dimensioning and Tolerancing

After completing this lesson the operator will be able to:

1. Describe practical benefits of GDT in manufacturing.

2. Explain concepts and terms of basic geometry as used in manufacturing, including:

a. Point.
b. Line.
c. Plane.
d. Two dimensional coordinate grid.
e. Three dimensional coordinate grid.
f. Origin of coordinate grid.
g. Positive and negative coordinates.
h. Circle, circumference, diameter and radius.
i. Cylinder.
j. Parallelism.
k. Perpendicularity.

3. Name and describe the five types of geometric tolerances.

a. Form
b. Orientation
c. Location
d. Runout
e. Profile

4. Name and describe the three types of Material Condition Modifiers.

a. Maximum Material Condition (MMC).
b. Least Material Condition (LMC).
c. Regardless of Feature Size (RFS).

5. Describe the benefits of true positioning location over "coordinate" location tolerancing.

a. Explain True Position tolerancing.
b. Explain "coordinate" location tolerancing.

6. Describe how GDT location tolerances also control the form of part features.
7. Read and interpret a GDT Feature Control Frame.

Lesson 3: Datums and Bonus Tolerance

After completing this lesson the operator will be able to:

1. Describe the major methods of measuring Geometric dimensions.

a. Functional gages.
b. Open setup inspection.
c. Optical comparator.
d. Air gages.
e. Coordinate measuring machine (CMM).

2. Describe why Datums are used for measuring Geometric dimensions.
3. Determine the correct order in which to establish Datums, given a Feature Control Frame callout.
4. Establish the correct number of points of contact when establishing Datums, based on the type of feature and the Datum order.
5. Establish Datums for measurement, including flat surfaces, linear elements along a cylindrical feature, rotational axes, slots or tabs.

6. Describe how Bonus Tolerances are possible based on material condition modifiers.
7. Given a part print and workpiece, calculate a bonus tolerance.

a. State the formula used to calculate bonus tolerances.
b. Calculate the MMC size of a feature.
c. Measuring the feature on a workpiece.
d. Calculate the bonus tolerance based on the previous

calculations.

e. Calculate additional bonus tolerance from the material condition of Datum features.

8. Calculate with Signed Numbers.

a. Explain the rules for adding signed numbers.

b. Explain the rules for subtracting signed numbers.

c. Calculate with signed numbers.

NOTE: Geometric Dimensioning and Tolerancing Job Aids are provided with the training system. These Job Aids cover each GDT tolerance in detail, including suggested measuring equipment and techniques.

Lesson 4: Using the Inch and Metric Systems.

After completing this lesson the operator will be able to:

1. Explain the elements of Metric measuring system.

a. Identify the thousandths, hundredths, tenths and whole millimeter places in a dimensional value.

b. Relate the centimeter and millimeter to the inch.

c. Estimate Metric measurements based on relations to the inch.

d. Locate Centimeter divisions on a scale.

e. Locate Millimeter divisions.

2. Locate and identify the following features on an Inch dimension.

a. One inch, decimals, fractions.

b. When fractions and decimals are used.

c. Half, quarter, eighths, sixteenths, thirty seconds and sixty fourths.

d. Thousandths, hundredths, and tenths of an inch and how they relate.

e. How decimals and fractions differ and converting between them.

f. Converting between the inch and metric systems.

g. Rounding to the correct number of decimal places.

h. Calculate Tolerance limits.

i. Running Tolerance limits.

3. Explain how to avoid Common errors in calculation.

a. Using mental estimation before calculation.

b. Avoiding Transposition of numbers.

c. Avoiding mistakes when entering values.

d. Communicating values correctly in inch and metric.

4. Factors which effect a measurement.

a. Cleanliness.

b. Temperature.

c. Gage accuracy.

d. Gauge setting.

e. Workpiece Deformation.

f. Gage deformation

g. Proper measuring technique.

5. The Steel Rule.

a. Tolerances that allow the use of a Steel Rule.

b. Smallest measurement = to .010 or 1/64, or .5 mm.

c. Sizes of rules.

d. Dual dimensioned Steel Rules.

e. Three causes of errors: Worn edge, rule not parallel, parallax error.

f. Count whole units to feature end point.

g. Count remaining unit spaces to end point.

h. Total the measurement.

i. Write down the value correctly for the feature measured.

j. Care and handling of the steel rule.

Lesson 5: Analog and Digital Micrometers

After completing this lesson the operator will be able to:

1. Identify features of Metric and Inch micrometers.

a. Identify major parts of the micrometer.

b. Locate the baseline and zero point.

c. Locate 100 thousandths division lines.

d. Locate the 25, 50, 75 thousandths lines.

e. Locate the thimble and thousandths division lines.

f. Locate whole and half mm divisions on metric micrometer.

g. Locate thimble scale on metric Micrometer.

h. OD & ID

i. Depth

j. Thread

k. Special mics.

2. Use and care of a conventional micrometer.

a. Clean Workpiece and Micrometer.

b. Rapidly open and close the micrometer

c. Different ranges of micrometers.

d. Close with proper tension a micrometer using a friction thimble or ratchet stop

e. Close with proper tension a micrometer which has a friction

knob.

f. Check zeroing of micrometer.

g. Explain thimble rotation and resulting spindle travel.

h. Hold micrometer properly.

i. Properly fit the micrometer to the workpiece to read the required dimension.

j. Changes in feel for round of cylindrical workpieces (reduce pressure to avoid compression of the piece.)

3. Read and total micrometer measurements.

a. Determine whole inch spaces.

b. Determine 100 thousandths spaces.

c. Determine 50 thousandths spaces.

e. Determine number of the 25 thousandths.

d. Write down values starting with micrometer size.

e. Align decimal points and total.

f. Identify the vernier scale and the tenth.

g. Read the vernier scale correctly.

h. Correctly total a measurement using a vernier instrument.

i. Ten-to-one rule.

j. Determine whole mm.

k. Determine half mm spaces.

l. Determine hundredths on sleeve.

m. Locate vernier scale on metric mic.

4. Identify and use Digital Mics.

a. Identify the types of digital micrometers.

b. Features of mechanical digital mics.

1. Measuring with a mechanical digital mic.

2. Reading the measurement.

3. Features of electronic digital mics.
4. Measuring with an electronic digital mic.
5. Reading the electronic measurement.
6. Storing and maintaining the instrument.

Lesson 6: Using Calipers and Go, No-Go Gages.

After completing this lesson the operator will be able to:

1. Identify caliper components and types of measurements.

    a. Find lengths or outside diameters using the Outside Jaws.
    b. Find inside diameters using the caliper blades.
    c. Find recesses and depths using the depth rod.
    d. Locate and explain the use of the Beam.
    e. Locate and explain the use of the Outside and Inside Jaws.
    f. Locate and explain use of the Depth Rod.
    g. Locate and explain the use of the Slide.
    h. Locate and explain the use of the Dial.
    i. Locate and explain the use of the Vernier scale.
    j. Explain the relationship between the divisions on the Beam and the Vernier scales.
    l. Explain the "offset" jaw caliper and its two vernier scales.

2. Measure a workpiece feature using a Dial Caliper.

    a. Check the calipers for proper adjustment and operation.
    b. Clean the workpiece and calipers, and fit the calipers to the workpiece.
    c. Apply the proper pressure during the measurement.
    d. Use the Knife Edges properly for measuring grooves.
    e. Rotate workpiece to find true diameter.

f. Read the number of whole inches on the Beam.

g. Read the number of hundreds of thousandths on the Beam.

h. Read the dial to determine the number of thousandths.

i. Total the measurement.

j. On a Metric Caliper, determine if centimeters or millimeters are used on the Beam.

k. Read the number of Beam units as millimeters.

l. Read the number of hundredths of a millimeter on the dial.

m. Add the readings to determine the measurement.

n. Clean and store the calipers when finished.

3. Measure a workpiece feature using a Vernier Caliper.

a. Explain the process of locating the matching lines on vernier scales.

b. Explain the beam and vernier divisions on a vernier caliper.

c. Locate the start point of the measurement.

d. Make an accurate measurement using an inch vernier caliper.

e. Locate the scales and divisions on an inch/metric caliper.

f. Using the metric scale, make an accurate measurement.

g. Describe how to determine the match line as it relates to the relative location to the tolerance allowed.

4. Understand digital calipers.

a. Locate and identify the features.

b. Explain the auto-shutoff feature.

c. Converting between metric and inch measurements.

d. Zeroing the caliper at any point.

e. Reading measurements with signed numbers.

5. Using Go and No-Go gages

a. Identify and explain the use of Go, No-Go gages.

b. Explain the components of, and properly use, snap gages.

c. Explain the use of Functional gages.

d. Explain and properly use Plug and Pin gages.

e. Explain and properly use Ring gages.

f. Explain what to do if a feature fails a Go, No-Go test.

Lesson 7: Measuring Surface Finish

After completing this lesson the operator will be able to:

1. Identify Characteristics of Surface Finish.

    a. Roughness

    b. Waviness

    c. Lay

    d. Flaws

2. Identify Units of measure for Surface Finish.

    a. Explain Profile and Nominal Profile.

    b. Identify and explain Microinches and Microns..

    c. Identify peaks and valleys.

    d. Identify and explain Ra and Aa.

    e. Identify and explain the surface finish symbol and its components.

3. Comparison Gages.

    a. Understanding Comparison Gages

    b. Range of Comparison Gages.

    c. Use of Comparison Gages to Measure Surface Finish.

4. Analog and Digital Profilometers

    a. Components of analog profilometers.
    b. Explain cutoff.
    c. Range of analog profilometers.
    d. Set up and measure with analog profilometers.
    e. Set up and measure with digital profilometers.

5. Portable Surface Roughness Gage

    a. Uses for the Roughness Gage.
    b. Identify components of the gage.
    c. Calibrate the gage.
    d. Measure Workpiece roughness.

Lesson 8: Optical Comparators and Thread Measurement

After completing this lesson the operator will be able to:

1. Identify the Elements of Threads and Thread Notation.

a. Locate and explain the elements of threads.

    1. Thread Crest.
    2. Thread Root.
    3. Pitch versus Pitch Diameter.
    4. Lead of a thread.
    5. Major Diameter.

        a. ID Threads
        b. OD Threads

6. Pitch Diameter
7. Minor Diameter.

    a. ID Threads
    b. OD Threads

8. How Geometric Tolerances apply.

b. Read and explain Print Notation of threads.

1. The Unified Inch Screw standard.
2. UNC, UNF, UNEF, UNJC, NPT and ACME designations.
3. Understanding Major Diameter notation and actual class size.
4. Locating the tolerances of Pitch Diameters on a chart.
5. Identifying the Threads Per Inch.
6. Calculating the Pitch from the TPI value.
7. Class of Fit and Hand designations.

c. Read and explain Metric Notation on a shop print.

1. Identify "M" (metric) symbol.
2. The nominal major diameter in millimeters.
3. The Pitch of the metric thread notation.
4. Metric Coarse Threads: Look up and read the thread Pitch from a chart of metric thread standard dimensions.
5. Metric Fine Threads: Read the thread Pitch behind the "times sign".
6. Describe the class of fit and g & h allowance notations.
7. State whether internal or external threads.
8. Read the Pitch Diameter if shown.

2. Explain the Methods of Measuring Threads.

a. Explain the steps involved in thread measurement.

b. Clean threaded features and measuring instruments.

c. Explain the features that a GO gage is checking on an OD thread.

d. Based on print specifications, select the measuring instrument.

f. Use a Go, No-Go Snap gage for Major Diameter and Pitch Diameter.

g. Detecting no-go conditions for inch and metric gages.

h. Use a go, no-go Plug gage for ID threads.

i. Correctly use Ring gages for OD threads.

j. Measure Pitch Diameter using a Pitch Micrometer.

3. Use a Tri-Roll gage to measure Pitch Diameter.

a. Identify the parts of a tri-roll gage.

b. The effect of multi-thread rolls.

c. Read the values on the face of the dial indicator.

d. Locate the tolerance limit dimensions in reference materials.

e. Identify the tolerance limit represented by a Master gage.

f. Place the master in the gage.

g. Set the tolerance limit on the dial face. (zero the dial)

h. Calculate and mark the other tolerance limit on the dial.

i. Note the full rotations when Mastering the gage.

j. Place a workpiece into the gage.

k. Complete a measurement at several locations.

l. Visually check thread quality and finish.

m. Check thread crests and identify faulty conditions.

4. Use an Optical Comparator.

a. Identify the components of an optical comparator with digital display.
b. Identify the axes of motion.
c. Identify the rotational axis of the table.
d. Explain the divisions on the screen grid.
e. Explain how the grid divisions change with changes in magnification.
f. Mount a threaded workpiece to the stage.
g. Set the angle of the table to the Helix Angle.
h. Focus the shadow.
i. Complete a root radius measurement.
j. Complete a feature length measurement.
k. Complete a angle measurement.
l. Measure a radius by matching to the grid arcs.
m. Identify Gage Charts and their magnification settings.
n. Read the limit lines scribed on a Gage Chart.

Lesson 9: Open Setup Inspection

After completing this lesson the operator will be able to:

1. Use an Indicator.

a. Identify components of dial and test indicators.
b. Interpret the reading on a dial face.
c. Explain the difference between balance dials and continuous dials.
d. Select the correct contact point for a dial indicator.
e. Set up an indicator properly for measurement.
f. Select the correct indicator (range, inch/metric).
g. Explain typical applications of indicators in inspection.

2. Use Surface Plates and Holding Devices

a. Describe surface plates, when they are used.
b. How to work with surface plates to avoid damaging them.
c. Fixture workpieces using the appropriate equipment.
d. Fixture workpieces by the proper feature for measurement.
e. Identify and use V-blocks.
f. Identify and use a Tri-Roll.
g. Explain Coaxial datum features.
h. Identify and use Parallel bars.
i. Identify and use Gage blocks for workpiece fixturing.

3. Use Gage Blocks

a. Describe uses of gage blocks.
b. Identify a typical set of shop-grade gage blocks.
c. Explain how to select gage blocks for a given dimension.
d. Explain how to properly clean and wring gage blocks.

4. Use Mechanical Height Gages.

a. Identify components of mechanical height gages.
b. Types of workpiece contact devices: test indicators, depth rod, scriber.
c. Rough and fine adjustment methods.
d. Vernier scales, dials, and digital readouts on height gages.
e. Using a Master height gage to create or measure a height setting.
f. How to use a Sweep height gage.

Lesson 10: Coordinate Measuring Machines

After completing this lesson the operator will be able to:

1. Identify and explain the purpose of CMMs and their components.

    a. Identify the types of CMMs found in shops.
    b. Explain the advantages of a CMM
    c. Identify the basic components of a CMM.
    d. Identify the axes of motion.
    e. Locate the Ways and their need for cleanliness.
    f. Locate the Axis Control Switches and explain their operation.
    g. Locate and explain the Manual Axis Controls.
    h. Locate and explain the features of the Probe and Probe Tip.
    i. Explain the use of an Indexable head.
    j. Explain the purpose of the Computer.
    k. Locate the Icons and their purpose.

2. Prepare for Measurement.

    a. Explain when it is necessary to HOME the CMM.
    b. Locate the Surface Plate and its mounting holes.
    c. Locate and explain the purpose of the Qualification Sphere.
    d. Clean the Surface Plate.
    e. Prepare the workpiece for measurement.
    f. Locate the piece and align a major axis to a CMM axis.
    g. Clamp the piece correctly.
    h. Locate and explain the coordinate display.
    i. Locate the origin and datums of a feature.
    j. Explain why the grid system is aligned to the origin of the features.
    k. Explain when a probe tip must be changed.
    l. Remove and replace a probe tip using the correct wrench.

m. Explain why the new tip must be qualified.

n. Start the Qualification process on the computer and

o. Explain when Multi-tip or Single Tip is to be used.

p. Move the probe in all three axes.

q. Properly touch-off the probe on the qualification sphere.

r. Explain the screen display and delete a hit..

s. Click DONE and explain the measured diameter value.

t. Locate and explain Standard Deviation values.

u. Locate and explain the Status Line values.

3. Align the CMM Coordinate System to a Feature's Datums.

a. Explain the auto-naming process of features and datums.

b. Clear any existing auto named features.

c. Identify stored measurement sequences from the file list.

d. Check the Status Line to confirm Alignment status

e. Perform a CLEAR ALIGNMENT sequence.

f. Locate the datums assigned to the features to be measured.

g. Use the PLANE function to locate a datum surface.

h. Auto-name the Plane and check standard deviation.

i. Explain what to do when Standard Deviation is incorrect.

j. Use the LINE function to locate a datum and auto-name it.

k. Use the CIRCLE function to locate a hole and auto-name it.

l. Complete an alignment process using the assigned datums.

4. Perform Common Measurement Procedures.

a. Explain and select a TRUE POSITION measurement sequence.

b. Touch-off a hole to find True Position.

c. Set the screen display to True Position MMC.

d. Locate and enter the Nominal print coordinates of X and Y.

e. Locate and enter the Nominal diameter value.

f. Locate and enter the Upper and Lower tolerance limits.

g. Locate and explain the True Position measured values.

h. Complete the sequence on other features making adjustments to the nominal values as required.

i. Explain and select an ANGLES measurement sequence.

j. Use a line-to-line sequence.

k. Touch-off the first line feature in the correct direction.

l. Touch-off the second line feature in the correct direction.

m. Complete the sequence and display the correct angle and deviation.

n. Explain and select a PERPENDICULARITY measurement sequence.

o. Use a Bore / Plane sequence.

p. Touch-off circles within the bore at two locations.

q. Recall the named Plane.

r. Complete the sequence and display the correct value and deviation.