THE UNITED REPUBLIC OF TANZANIA
NATIONAL EXAMINATION COUNCIL OF TANZANIA
DIPLOMA IN TECHNICAL EDUCATION EXAMINATION

789 **METAL WORKING AND MECHANICAL PRACTICE**

Time: 3 Hour. **ANSWERS** **Year: 2014**

Instructions

1. This paper consists of **eight (8)** questions.
2. Answer any **five (5)** questions.
3. Each question carries **twenty (20)** marks.
4. Non-programmable calculators may be used.
5. Communication devices, programmable calculators and any unauthorized materials are **not** allowed in the examination room.
6. Write your **Examination Number** on every page of your answer booklet(s).

1. (a) Define "cutting speed" and state its significance in machining.
Cutting speed is the rate at which the cutting tool moves relative to the surface of the workpiece, typically expressed in meters per minute (m/min). It determines how fast the material is being removed and plays a crucial role in tool life, surface finish, and production efficiency.

(b) (i) Give the formula for calculating cutting speed.
The formula for calculating cutting speed (V) is:
$V = (\pi \times D \times N) / 1000$
where V = cutting speed (m/min), D = diameter of the workpiece (mm), N = spindle speed (rpm)

(ii) Calculate the cutting speed for a workpiece of 80 mm diameter rotating at 1000 rpm.
$V = (\pi \times 80 \times 1000) / 1000 = 251.33$ m/min

(c) Explain how cutting speed affects tool life.
High cutting speeds generate more heat, leading to faster wear of the cutting edge and shorter tool life. On the other hand, very low speeds may not cut efficiently. Using the recommended speed ensures balance between performance and tool longevity.

(d) State four factors that determine cutting speed selection.
The hardness of the workpiece material affects how fast it can be cut. The type of cutting tool material, such as HSS or carbide, determines how much heat it can handle. The machining operation, whether roughing or finishing, also influences the speed. The use of cutting fluids impacts the temperature and thus the allowable cutting speed.

2. (a) What is meant by the term "filing" in metalwork?
Filing is a manual machining process that uses a file to remove small amounts of metal to smooth, shape, or size a surface. It is typically a finishing operation done after sawing or cutting.

(b) (i) State three types of files and their uses.
Flat files are used for general-purpose smoothing and squaring surfaces. Half-round files are used on both flat and curved surfaces. Round files are suitable for enlarging holes or filing concave shapes.

(ii) Describe the proper filing technique to achieve a flat surface.
Hold the file with both hands, apply pressure during the forward stroke, and lift it during the return stroke. Maintain a consistent angle and apply even pressure to avoid rounding or uneven surfaces.

(c) Explain the importance of using a file card during filing.

A file card is used to clean metal particles from between the teeth of the file. This prevents clogging, which can scratch the workpiece and reduce the cutting efficiency of the file.

(d) List four safety precautions when using files.

Always use a handle to avoid injury from the tang. Do not use files on hardened materials unless they are specially hardened. Avoid excessive pressure that can cause the file to slip. Keep fingers away from the cutting area to avoid cuts.

3. (a) Define "chain drilling" and explain its purpose.

   Chain drilling involves drilling a series of overlapping holes along a marked path, usually to remove material from the interior of a shape that cannot be cut directly. It is often used in preparation for internal cutting.

(b) (i) Identify two tools used in chain drilling.

A twist drill is used to drill the holes. A centre punch is used to mark the hole positions before drilling.

(ii) State two advantages of chain drilling in workshop operations.

It allows removal of internal material where a saw cannot reach. It minimizes material wastage and enables cutting of complex internal shapes without distortion.

(c) Describe the process of removing internal material using chain drilling.

Mark the internal profile on the workpiece. Use a centre punch to mark hole locations along the path. Drill closely spaced holes along the marked path, ensuring they overlap. Break out the web between holes using a chisel or file.

(d) State four common errors in chain drilling and how to avoid them.

Incorrect hole spacing can leave excess material, so measure carefully. Misalignment of holes can result in poor finish—use marking out tools precisely. Drill wandering can be prevented by center punching accurately. Excessive feed rate may damage drills—use proper speed and feed.

4. (a) What is meant by "tool post" in a lathe machine?

   A tool post is the part of a lathe used to mount and hold the cutting tool. It allows the tool to be positioned and secured during machining operations.

(b) (i) List three types of tool posts.

The single screw tool post, the four-way tool post, and the quick-change tool post.

(ii) Explain how each type supports the cutting tool.

A single screw tool post holds one tool and allows simple angular adjustments. A four-way tool post can hold up to four tools at once and is rotated to use the desired tool. A quick-change tool post allows for rapid swapping of tools using preset tool holders.

(c) Describe the procedure of setting a cutting tool to center height.

Mount the tool in the holder and bring the tool tip to the height of the lathe center. Use a center gauge or align it visually with the live center. Adjust using shims or tool post height adjustment until perfectly aligned.

(d) State four consequences of incorrect tool height setting.

If too high, the tool may rub instead of cut. If too low, it may dig in and break. Poor height leads to poor surface finish. Incorrect height can also result in uneven wear or tool breakage.

5. (a) Define “countersinking” in drilling.

   Countersinking is the process of enlarging the top part of a drilled hole to allow a screw head or bolt head to sit flush with or below the surface.

(b) (i) State two differences between countersinking and counterboring.

Countersinking creates a conical recess while counterboring makes a flat-bottomed hole. Countersinking is used for flat-head screws, counterboring for socket-head bolts.

(ii) List three applications of countersinking in mechanical assemblies.

Used in assembling flat-head screws in panels, for removing burrs from drilled holes, and to provide aesthetic finish in visible surfaces.

(c) Describe how to perform countersinking on a drilled hole.

Mount a countersink tool in a drill press. Align the tool with the drilled hole. Use moderate speed and feed to slowly create the conical recess. Withdraw once the desired depth is reached.

(d) Give four precautions to observe during countersinking operations.

Use appropriate tool size and angle. Avoid excessive speed to prevent chatter. Clamp the workpiece securely. Do not countersink too deep as it weakens the material.

6. (a) What is meant by "tool wear"?

Tool wear refers to the gradual removal of material from a cutting tool due to friction, heat, and mechanical forces during machining. It eventually affects tool performance and surface finish.

(b) (i) Identify four types of tool wear.

Flank wear occurs on the side of the tool. Crater wear appears on the rake face. Notch wear occurs at the cutting edge. Built-up edge is material adhering to the tool tip.

(ii) Explain how each affects machining performance.

Flank wear causes poor surface finish. Crater wear reduces tool strength. Notch wear leads to chatter and dimensional inaccuracy. Built-up edge causes uneven cutting and poor finish.

(c) Describe three methods of reducing tool wear in metal cutting.

Use appropriate cutting speeds and feeds. Apply cutting fluids to reduce heat. Use harder tool materials like carbide or coated tools for longer life.

(d) State four indicators that a cutting tool needs replacement.

Poor surface finish on the workpiece. Increased cutting forces and vibration. Visible wear on the tool edge. Unusual noise or burning smell during cutting.

7. (a) Explain the term "go and no-go gauge".

A go and no-go gauge is a measuring device used in quality control to check whether a dimension is within specified tolerances. The "go" end must fit the part, while the "no-go" end must not fit.

(b) (i) State two advantages of using go/no-go gauges in inspection.
They provide quick pass/fail decisions. They eliminate the need for detailed measurements during high-volume inspection.

(ii) List two examples of go and no-go gauges.

Plug gauges for holes. Snap gauges for shafts.

(c) Describe how to use a go and no-go plug gauge to check a hole.

Insert the "go" plug into the hole; it should fit smoothly. Insert the "no-go" plug; it should not enter. If both conditions are met, the hole is within tolerance.

(d) State four limitations of using go/no-go gauges.

They do not indicate the actual size. They cannot detect gradual wear. They require precise manufacturing of the gauges themselves. They are not suitable for recording or statistical analysis.

8. (a) Define "bending allowance" in sheet metal work.

Bending allowance is the amount of material needed to accommodate the bend in a sheet metal piece. It ensures accurate fabrication by accounting for material stretch during bending.

(b) (i) Give the formula for calculating total developed length.

Total Developed Length = Leg1 + Leg2 + Bending Allowance

(ii) A 2 mm thick sheet is to be bent 90° around a 10 mm radius. Calculate the bending allowance using K = 0.33.

Bending Allowance = $(\pi/2) \times (R + K \times T)$

$= 1.5708 \times (10 + 0.33 \times 2) = 1.5708 \times (10 + 0.66) = 1.5708 \times 10.66 = 16.74$ mm

(c) Explain the importance of bending allowance in fabrication.

It ensures that the fabricated part dimensions are accurate after bending. Without allowance, bends would distort the overall size and compromise fitting during assembly.

(d) State four factors affecting bending allowance.

Material thickness influences how much stretch occurs.

Bend radius affects the amount of material needed.

Type of material (ductile or brittle) changes how it behaves during bending.

Bend angle determines the arc length and thus the allowance needed.