Trade of Toolmaking	
Module 5:	Press Tools, Jigs & Fixtures, Mouldmaking
Unit 2:	Blanking Tool (Unguided)
	Phase 2

Published by

SOLAS
An tSeirbhís Oideachais Leanúnaigh agus Scileanna
Further Education and Training Authority

Table of Contents

Document Release History

Date	Version	Comments
25/09/2014	2.0	SOLAS transfer

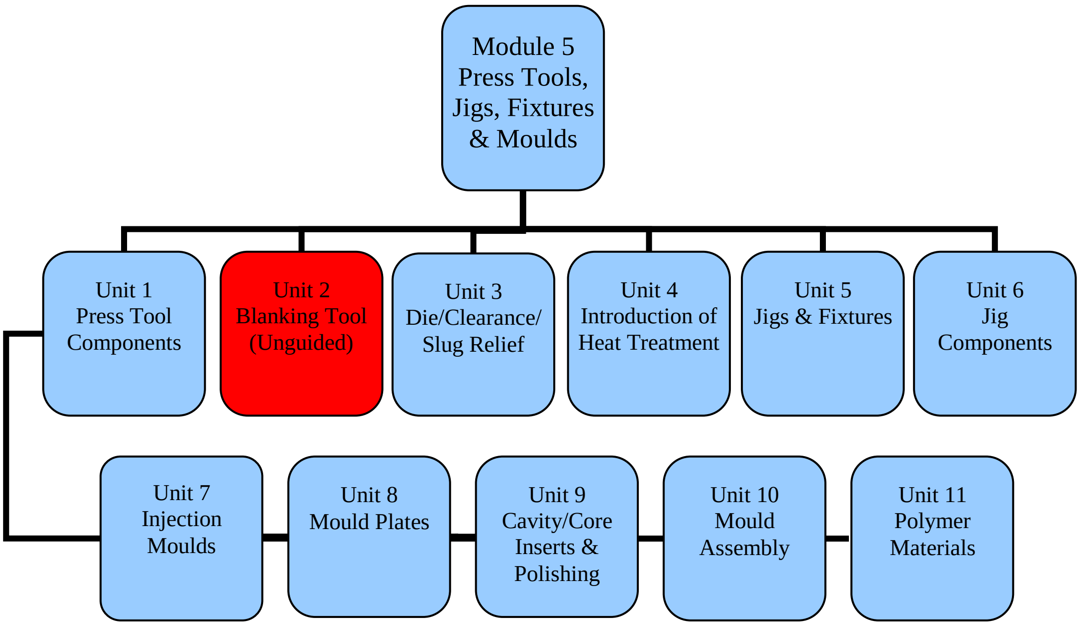
Unit Objective

On completion of this unit you will be able to select and use appropriate materials used in the manufacture of the press tool. You will also learn about the effect of carbon on steel and how heat treatment changes the properties of the steel.

Introduction

Module five of this course covers Press Tools, Jigs & Fixtures, Mouldmaking. This is the second unit in module five and identify the safety precautions to be taken when working on press tools during assembly and operation. It is important to read and interpret a drawing before starting any task

Different types of materials are used in the press tool. Mild steel can be used for components such as the base plate and stripper plate, but tool steel must be used for the punch and die components. When manufacturing the punch and die it is important to understand that these components are machined close to the finished size, but then need to be heat treated to harden the material. You should also understand the affect that the carbon content has on steel.



By the end of this unit you will be able to:

- Identify the safety precautions to be taken when working on press tools during assembly and operation.
- Read and interpret engineering assembly and component drawings in first and third angle projection.
- Select and use appropriate tooling materials.
- Plan the sequence of manufacture and machining of various parts for assembly of sample tool.
- Assemble sample tools.
- Inspect sample component from trial and identify the relationship between: a) Component and punch diameters and b) Strip and die hole diameters.

1.0 Identifying The Safety Precautions To Be Taken When Working On Press Tools

Key Learning Points

Materials handling, machine shop safety, machine operation, hand skills, assembly procedures and organisation of surrounding work areas. Safe use of jig boring machine, lathe and milling machine. Safe use of a fly press.

1.1 Materials Handling, Machine Shop Safety, Machine Operation, Hand Skills, Assembly Procedures And Organisation Of Surrounding Work Areas

When the components have been machined to size, all burrs and sharp edges need to be removed with a file. It is important to keep the work area and the workshop clean and tidy. All tools and equipment must be returned to their respective toolbox or storage area when not in use. Ensure that the floor is kept free of debris, oil and coolant spills. Clean up spills immediately. When using machines always wear safety glasses and tie back long hair or loose clothing.

1.2 Safe Use Of Jig Boring Machine, Lathe And Milling Machine

When using the jig boring machine, lathe or the milling machine, safe working procedures need to be applied. Wear safety glasses at all times. Keep hands away from swarf and use a long handed rake to remove swarf, only after the machine has been stopped. Wear suitable protective clothing. Loose clothing, long sleeves, ties and long hair are a source of danger and therefore should be securely contained. When using the lathe, remove the chuck key immediately after use. Keep the area around machine clean and tidy and free from wet or oily patches.

1.3 Safe Use Of A Fly Press

The fly press can be used for blanking, piercing and bending operations, but is suitable only for light press work and low production rates. The fly press is hand operated and as the operator needs to be close to the press tool in operation, it is important to wear safety glasses, suitable clothing and keep hands away from the working area.

2.0 Reading And Interpreting Engineering Drawings In First And Third Angle Projection

Key Learning Points

First and third angle projection, isometric/oblique drawings, abbreviations and standards to BS 8888. Partial and staggered sections on engineering drawings. Geometric tolerances and machining symbols. Limits and fits to BS 4500. Classes of fit and types of fit. Hole and shaft basis system. Definitions: i.e. tolerance, nominal size, allowance, deviation. Allowances and tolerances for assembly work.

2.1 First And Third Angle Projection, Isometric/Oblique Drawings, Abbreviations And Standards To BS 8888

Isometric and oblique drawings of components can be generated from first and third angle projected drawings. These sketches help to visualise a part or an assembly prior to manufacture. All isometric drawings are based on three axis spaced 120º apart. One axis is drawn vertical and the side walls are drawn at 30º to the right and left. In oblique drawings one of the faces of the component is parallel to the plane of projection and is drawn in its true shape. The side face is then drawn at 45º to the right.

Ref: Simmons, Colin H & Maguire, Dennis E 2004, *Manual of engineering drawing*, 2nd edn, Elsevier Science & Technology, chapter 4, *Principles of first and third angle orthographic projection*, p. 33.
ISBN-13: 9780750651202

2.2 Partial and staggered sections on engineering drawings

The assembly drawing shows the components assembled together and may be sectioned to show internal parts. The individual parts are numbered on the drawing and a table lists the components, the individual single-part drawing number, the quantity required of each component.

Ref: Simmons, Colin H & Maguire, Dennis E 2004, *Manual of engineering drawing*, 2nd edn, Elsevier Science & Technology, chapter 8, *Sections and sectional views*, p. 64.
ISBN-13: 9780750651202

2.3 Geometric tolerances and machining symbols

Geometric tolerances are used when it is necessary to control more precisely the form or shape of a feature. It can be used to define straightness, flatness, roundness, parallelism, perpendicularity, positional, concentricity etc. It is recommended to use geometric tolerances for parts that fit together.

Ref: Simmons, Colin H & Maguire, Dennis E 2004, *Manual of engineering drawing*, 2nd edn, Elsevier Science & Technology, chapter 20, *Geometric tolerancing and datums*, p. 160.
ISBN-13: 9780750651202

When the surface texture symbol is used on the drawing, the machining process, such as Grind, Ream and Polish can be written beside this symbol.

Ref: Simmons, Colin H & Maguire, Dennis E 2004, *Manual of engineering drawing*, 2nd edn, Elsevier Science & Technology, chapter 14, *Dimensioning principles; Graphical symbols*, p. 109.
ISBN-13: 9780750651202

2.4 Limits And Fits To BS 4500

The ISO and BS limits and fits system is used to manufacture components that need to be assembled together and be interchangeable with each other. The limits and fits system was introduced whereby all components are manufactured to a specific size within narrow limits. Components cannot be made to exactly the same dimensions, but the use of limits allows the components to vary slightly from the nominal dimension, but within the set limit.

Ref: Simmons, Colin H & Maguire, Dennis E 2004, *Manual of engineering drawing,* 2nd edn, Elsevier Science & Technology, chapter 19, *Limits and fits*, p. 154.
ISBN-13: 9780750651202

2.5 Classes Of Fit And Types Of Fit

The limits and fits system has a range of (i) clearance, (ii) transition and (iii) interference fits.

Ref: Simmons, Colin H & Maguire, Dennis E 2004, *Manual of engineering drawing,* 2nd edn, Elsevier Science & Technology, chapter 19, *Limits and fits*, p. 154.
ISBN-13: 9780750651202

2.6 Hole And Shaft Basis System

There are two bases of the limits and fits system, (i) the **Hole Basis** (Ref.: BSI data sheet 4500A) and (ii) the **Shaft Basis** (Ref.: BSI data sheet 4500B). In the Hole Basis system the hole size is kept constant and the shaft size varied to provide the required fit. Whereas in the Shaft Basis system the shaft is kept constant and the hole is varied to provide the required fit. The Hole Basis system is the most widely used system and is more cost effective. In this system one size reamer can be used and a range of fits can be produced by varying the shaft limits.

Ref: Simmons, Colin H & Maguire, Dennis E 2004, *Manual of engineering drawing,* 2nd edn, Elsevier Science & Technology, chapter 19, *Limits and fits; Bases of fits*, p. 155.
ISBN-13: 9780750651202

2.7 Definitions: I.e. Tolerance, Nominal Size, Allowance, Deviation

Definitions: *Tolerance*: This is the difference between the upper and lower limits of size. *Nominal size*: The size that a component is referred to, but may not be exactly be that dimension, e.g. a Ø10mm hole may be slightly larger in order to provide adequate clearance. *Allowance*: This is related to mating parts and is the difference between the high limit of size of the shaft and the low limit of size of the mating hole. Depending on whether the fit is a clearance or interference, the allowance can be positive or negative. *Deviation*: This is the difference between the maximum, minimum, or actual size of a shaft or hole and the basic size.

2.8 Allowances And Tolerances For Assembly Work

All dimensions on a drawing need a tolerance. The traditional method of tolerancing a drawing is to write the tolerance band beside each dimension and explain the design requirements e.g. squareness, flatness, concentricity etc, by adding notes to the drawing. Another method is to use geometric tolerancing, which uses symbols on drawings to define the form and size of a tolerance zone within which the feature needs to be contained.

3.0 Selecting And Using Appropriate Tooling Materials

Key Learning Points

Plain carbon steel manufacture. Effect of carbon content on steel and heat treatments applied to change the steel properties.

3.1 Plain Carbon Steel Manufacture

Iron is an element that is extracted from iron ore, which is mined in various parts of the world. In order to produce steel, the iron ore is processed in two main stages. In the first stage *Pig Iron* is produced in a blast furnace. This type of iron is weak and brittle due to the high carbon content (4%) and is of little use in industry. The Pig Iron therefore needs to be processed further to reduce carbon in the steel. The main processes used are: (i) the Bessemer process, (ii) the basic oxygen process and (iii) the open hearth process. Mild Steel, Medium and High Carbon Steels can be produced from these processes.

3.2 Effect Of Carbon Content On Steel And Heat Treatments Applied To Change The Steel Properties

The most common steel used in the Toolroom is Mild Steel, which has a carbon content of approx. 0.15% to 0.5%. This material is easily machined and can be used for parts of the press tool that are not subject to high stresses and wear, e.g. the stripper plate and base plate. For components such as the punch and die, tool steel is used that has a carbon content of 1%. It also contains tungsten, manganese and chromium. These components are then heat treated to bring them up to the required hardness and then ground to size.

4.0 Plan Sequence Of Manufacturing And Machining Operations

Key Learning Points

Initiative, communication, job sequencing and planning. Calculating speeds and feeds for machining sequencing to suit cutting tools and materials being machined. Accurate hole transfer between various steel plates. Determination of the clearances for punch and die.

4.1 Initiative, Communication, Job Sequencing And Planning

Drawings should be read carefully prior to carrying out any task and it is important to plan the sequence of operations. Tools and material should be laid out and organised in a neat and logical order. If you have any questions or not sure about something ask your Instructor.

4.2 Calculating Speeds And Feeds For Machining Sequencing To Suit Cutting Tools And Materials Being Machined

The correct spindle speed needs to be set for each cutter. This is calculated by entering the cutting speed and the cutter diameter into the RPM formula, where the *Cutting Speed* is expressed in meters per minute. Charts are available that recommend the correct cutting speed for a particular material, e.g. a typical cutting speed of 30 meters/min is used for mild steel and 20 metres/min for tool steel. Therefore the spindle speed will be lower for the tool steel when compared to that of mild steel.

Recommended cutting speed in metres per minute:

Material	High Speed Steel (metres/min)	Carbide Cutter (metres/min)
Tool Steel	21 - 33	68 – 158
Mild Steel	33 – 42	98 – 188
Cast Iron	21 – 30	68 – 98
Brass	42 – 75	120 – 180
Aluminium	60 – 105	180 - 350
Plastic	60 - 450	unlimited

To find the correct RPM (revs per minute) setting of the spindle the following formula should be used;

$$RPM = \frac{\text{Cutting Speeds in metres per minute x 1000}}{\text{Circumference of cutter in millimetres}}$$

$$= \frac{S \times 1000}{\pi \times D}$$

For example, for a mild steel part a cutting speed of 30 m/min is chosen from the above table. If it is to be machined with a Ø20mm High Speed Steel end mill, the RPM is calculated as follows:

$$RPM = \frac{30 \times 1000}{3.14 \times 20} = 478$$

Feed Rate

Feed rate is the speed at which the workpiece is moved relative to the cutter. 0.05 is chosen from the cutter manufacturers tables. The feed rate is calculated as follows:

Feed Rate = feed/tooth x No. of cutting teeth x RPM

= 0.05 x 4 x 478

= 96 mm/min

Time to Cut

If the workpiece is 100mm long, then the time to cut is calculated as follows:

$$\text{Time to cut} = \frac{\text{length of cut in mm}}{\text{Feed rate in mm/min}}$$

$$= \frac{100}{96}$$

= 1.04 minutes

= 62.5 seconds

Ref: Black, Bruce J 2004, *Workshop processes, practices and materials*, 3rd edn, Elsevier Science & Technology, chapter 9, p. 153.
ISBN-13: 9780750660730

4.3 Accurate Hole Transfer Between Various Steel Plates

When two metal plates need to be screwed together, the top plate is first drilled. The bottom plate is then clamped to the drilled top plate with toolmakers clamps and the drilled holes are used to guide a drill of the same size to spot the bottom plate.

4.4 Determination Of The Clearances For Punch And Die

The clearance between the punch and the die is dictated by the thickness and hardness of the material, e.g. for punching mild steel strip, the clearance should be 5-10% of the thickness of the strip.

Ref: Black, Bruce J 2004, *Workshop processes, practices and materials*, 3rd edn, Elsevier Science & Technology, chapter 16, *Presswork*, sec. 16.2, *Press-tool design*, p. 269.
ISBN-13: 9780750660730

5.0 Assembly Of Sample Tools

Key Learning Points

Assembly and disassembly techniques.

5.1 Assembly And Disassembly Techniques

The press tool components are assembled as follows: assemble the *base plate* and *die block* together with the dowel pins. Insert the *strip stop* into die. Locate the *stripper plate* onto the die by locating it onto the same dowels. Secure the three plates with the locking screws. Finally assemble the *punch* with the *spigot*.

6.0 Inspecting Sample Component From Trial

Key Learning Points

Blanking tool performance: inspection of size, shape and finish on component.

6.1 Blanking Tool Performance: Inspection Of Size, Shape And Finish On Component

As mentioned above, the clearance between the punch and the die is very important. If the clearance is too small, then the edges of the blank will look rough and uneven. If the clearance is too big, the blank will be dished on the punched side and the edge will be radiused on the far side. Also the edge of the component will be tapered.

The blank or a hole will always have a taper but this needs to be kept to a minimum. Upon inspection of the blank it will be found that (i) the size of the punch will be the same size as the hole it produced and (ii) the size of the die will be the same size as the blank.

Ref: Black, Bruce J 2004, *Workshop processes, practices and materials*, 3rd edn, Elsevier Science & Technology, chapter 16, *Presswork*, sec. 16.2, *Press-tool design*, p. 269.
ISBN-13: 9780750660730

Summary

Identifying the safety precautions to be taken when working on press tools: When the components have been machined to size, all burrs and sharp edges need to be removed with a file. It is important to keep the work area and the workshop clean and tidy. All tools and equipment must be returned to their respective toolbox or storage area when not in use. Ensure that the floor is kept free of debris, oil and coolant spills. Clean up spills immediately. When using machines always wear safety glasses and tie back long hair or loose clothing.

Reading and interpreting engineering drawings in first and third angle projection: There are two systems of projection, First Angle and Third Angle, which are based on a framework of planes at right angles. In first angle projection, each view shows what would be seen by looking on the far side of an adjacent view. In Third angle projection, each view shows what would be seen by looking on the near side of an adjacent view. Grid paper is used for drawing isometric and oblique drawings.

Selecting and using appropriate tooling materials: The most common steel used in the Toolroom is Mild Steel, which has a carbon content of approx. 0.15% to 0.5%. This material is easily machined and can be used for parts of the press tool that are not subject to high stresses and wear, e.g. the stripper plate and base plate. For components such as the punch and die, tool steel is used that has a carbon content of 1%. It also contains tungsten, manganese and chromium. These components are then heat treated to bring them up to the required hardness and then ground to size.

Plan sequence of manufacturing and machining operations: Drawings should be read carefully prior to carrying out any task and it is important to plan the sequence of operations. Tools and material should be laid out and organised in a neat and logical order. If you have any questions or not sure about something ask your Instructor.

Assembly of sample tools: The press tool components are assembled as follows: assemble the *base plate* and *die block* together with the dowel pins. Insert the *strip stop* into die. Locate the *stripper plate* onto the die by locating it onto the same dowels. Secure the three plates with the locking screws. Finally assemble the *punch* with the *spigot*.

Inspecting sample component from trial: As mentioned above, the clearance between the punch and the die very important. If the clearance is too small, then the edges of the blank will look rough and uneven. If the clearance is too big, the blank will be dished on the punched side and the edge will be radiused on the far side. Also the edge of the component will be tapered.

The blank or a hole will always have a taper but this needs to be kept to a minimum. Upon inspection of the blank it will be found that (i) the size of the punch will be the same size as the hole it produced and (ii) the size of the die will be the same size as the blank.

Suggested Exercises

1. Draw an isometric sketch of a fly press and label its main features.
2. What safety precautions should you take when using a fly press.
3. What is the difference between the Hole and Shaft based systems.
4. Calculate the spindle speed to mill a mild steel plate with a 12mm high speed steel end mill.
5. Mill two mild steel plates to the same size. Drill a Ø6 millimetre at each corner in one of the plates. Now accurately transfer the holes into the second plate.

Questions

1. Why is Geometric Tolerancing used in Technical Drawing.
2. In the limits and fits system, what are the three classes of fits?
3. What affect does carbon have on steel and give examples?
4. What affect does the following have on a punched component:
   (i) When the clearance between the punch and die is too small.
   (ii) When the clearance between the punch and die is too big.
5. When a blank is measured, is it the same size as the punch or the die? Also, the same question applies to a pierced hole.

Answers

1. Geometric tolerances are used when it is necessary to control more precisely the form or shape of a feature.
2. The limits and fits system has a range of (i) clearance, (ii) transition and (iii) interference fits.
3. The higher the carbon content in the steel the harder the steel. Mild Steel, has a carbon content of approx. 0.15% to 0.5%. It can be used for parts of the press tool that are not subject to high stresses and wear, e.g. the stripper plate and base plate. For components such as the punch and die, tool steel is used that has a carbon content of 1%.
4. If the clearance is too small, then the edges of the blank will look rough and uneven. If the clearance is too big, the blank will be dished on the punched side and the edge will be radiused on the far side. Also the edge of the component will be tapered.
5. When a blank or a pierced hole is measured it is found that, (i) the size of the die will be the same size as the blank and (ii) the size of the punch will be the same size as the hole it produced.

Recommended Additional Resources

Reference Books

Black, Bruce J 2004, *Workshop processes, practices and materials*, 3rd edn, Elsevier Science & Technology.

ISBN-13: 9780750660730

Simmons, Colin H & Maguire, Dennis E 2004, *Manual of engineering drawing*, 2nd edn, Elsevier Science & Technology.

ISBN-13: 9780750651202

Bird, John 2005, *Basic engineering mathematics*, 4th edn, Elsevier Science & Technology.

ISBN-13: 9780750665759