Trade of Toolmaking	
Module 1:	Induction & Bench Fitting
Unit 7:	Precision Internal Filing
	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

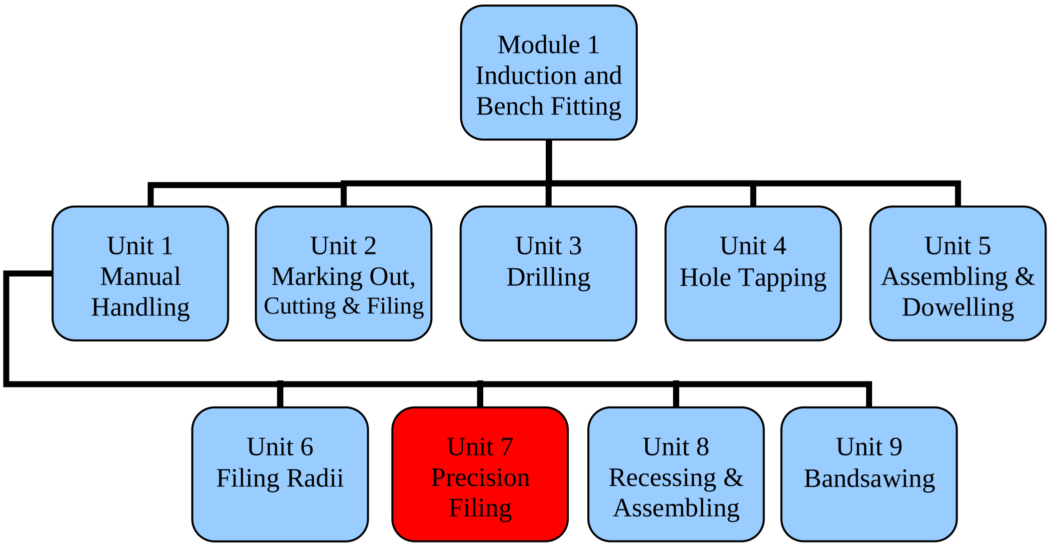
Unit Objective

On completion of this unit you will be able to file an internal square, produce freehand isometric sketches, explain the significance of geometric tolerancing and calculate the clearance allowance between the mating components.

Introduction

Module one of this course covers induction and bench fitting. This is the seventh unit in module one and introduces the techniques associated with precision internal filing, how to produce freehand isometric and oblique sketches, understand geometric tolerancing and learn how to calculate the maximum and minimum clearance allowances between the mating parts.

Internal shapes can be created to a high degree of accuracy and finish by using bench fitting techniques. Marking out, drilling, hack-sawing and filing metal plates has already been explained in previous units. In order to produce high precision parts using hand files, great care and patience is required. It is also important to learn how to use precision equipment such as micrometers and slip gauges.



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

- Use precision filing techniques to produce an internal square in mild steel to the required accuracy and finish specified in workshop drawings.
- Produce neat freehand, isometric and oblique sketches of components from first and third angle drawings to aid visualisation.
- Explain the significance of geometric tolerance applied to first and third angle projection drawings.
- File and assemble an external square to the required size and finish to provide a clearance fit with an external square.
- Calculate the maximum and minimum clearance/allowance between the mating components.

1.0 Precision Filing Techniques

Key Learning Points

Reading of workshop drawing to plan the sequence of operations prior to manufacturing. Importance of datum surfaces to hold tolerances. Precision filing, flat filing, square to relevant surfaces, parallel to reference surfaces.

1.1 Reading Of Workshop Drawing To Plan The Sequence Of Operations Prior To Manufacturing

Job planning is important prior to starting any task. The drawing should first be studied and understood. The drawing can initially be used to calculate the material requirement for the component to be manufactured. The workpiece is marked out using the dimensions and datum's as specified on the drawing. A basic level of maths is required such as addition, subtraction, multiplication, division, fractions and decimals.

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.*
ISBN-13: 9780750651202

1.2 Importance Of Datum Surfaces To Hold Tolerances

Parts should be marked out using the same datum edges as those specified on the drawing. It means that all features are marked out and measured from one end of the part, which is more accurate and easier to hold the required tolerance. Normally there are two datum edges, which are at right angles to each other.

Ref: Black, Bruce J 2004, *Workshop processes, practices and materials*, 3rd edn, Elsevier Science & Technology, chapter 3, *Marking out*, p. 44.
ISBN-13: 9780750660730

1.3 Precision Filing, Flat Filing, Square To Relevant Surfaces, Parallel To Reference Surfaces

When the component has been cut roughly to size using a hack-saw, the file is used to finish the component to the specified drawing dimensions. A range of files can be used, which are available in various shapes are Flat, Hand, Warding, Square, Three Square and Half Round, depending on the shape of the component and the precision and finish required. Files such as smooth or dead smooth are used to finish the feature close to the specified drawing dimensions. A vernier callipers is used to measure the internal shape of the component as it is filed close to the required dimension. At this stage slip gauges assembled to the maximum and minimum values for a specific dimension. Needle files are used to remove small amounts of material and can be used to remove material to form sharp corners. While filing it is important to continuously check the dimensions and also check that the filed surfaces are square.

Ref: Black, Bruce J 2004, *Workshop processes, practices and materials*, 3rd edn, Elsevier Science & Technology, chapter 2, *Hand processes*, p. 30.
ISBN-13: 9780750660730

2.0 Producing Isometric And Oblique Sketches Of Components

Key Learning Points

First angle projected drawings used to generate isometric and oblique sketches of individual and assembled components.

2.1 First Angle Projected Drawings Used To Generate Isometric And Oblique Sketches Of Individual And Assembled Components

Isometric and oblique sketches 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

3.0 The Significance Of Geometric Tolerance

Key Learning Points

Identification of key features on which geometric tolerance should be applied. Application of geometric and general tolerances. Dimensioning to BS 8888 standards. Areas: maximum and minimum material conditions.

3.1 Identification of key features on which geometric tolerance should be applied

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

3.2 Application Of Geometric And General Tolerances

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. by adding notes to the drawing. Another method used to define straightness, flatness, roundness, parallelism, perpendicularity, positional, concentricity etc, 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.

Ref: Simmons, Colin H & Maguire, Dennis E 2004, *Manual of engineering drawing*, 2nd edn, Elsevier Science & Technology, chapter 21, *Application of geometrical tolerances*, p. 168.
ISBN-13: 9780750651202

3.3 Dimensioning To BS 8888 Standards

The drawing should contain adequate information such as, dimensions, tolerances surface finish, etc., to allow the part to be manufactured. This information is also used to inspect the finished part. BS 8888 covers all aspects of dimensioning.

Ref: Simmons, Colin H & Maguire, Dennis E 2004, *Manual of engineering drawing*, 2nd edn, Elsevier Science & Technology, chapter 14, *Dimensioning principles*, p. 100.
ISBN-13: 9780750651202

3.4 Areas: Maximum And Least Material Conditions

The *maximum material condition* (MMC) is when the part or a feature contains the maximum amount of material, e.g. a block of metal with a hole drilled through, where the external surfaces are at the maximum size allowed and the hole is at the minimum size. The *least material condition* (LMC) is where the external surfaces are on at the minimum size allowed and the hole is at the maximum size.

Ref: Simmons, Colin H & Maguire, Dennis E 2004, *Manual of engineering drawing*, 2nd edn, Elsevier Science & Technology, chapter 22, *Maximum material and least material principles*, p. 179.
ISBN-13: 9780750651202

4.0 Filing And Assembling Components With The Required Clearance Fit

Key Learning Points

Speeds and feeds for centre drilling. Drilling corners on internal features. Use of needle files. Deburring sharp edges. Removal of waste material on internal features. Assembly techniques, use of feeler gauges.

4.1 Speeds And Feeds For Centre Drilling

In order to use the drilling machine efficiently it is important to calculate the correct cutting speeds and feeds using the relevant formulae. The correct speeds and feeds will ensure more efficient machine operation and will result in reduced tool wear. When drilling using a bench or pillar drill, the feed rate is normally controlled by hand.

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. for mild steel, a typical cutting speed of 30 meters/min is used and for tool steel the cutting speed is 20 metres/min. Therefore the spindle speed will be lower for the tool steel when compared to that of mild steel.

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

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

`

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

Ref: Black, Bruce J 2004, *Workshop processes, practices and materials*, 3rd edn, Elsevier Science & Technology, chapter 7, *Cutting tools and cutting fluids*, sec. 7.4, *Cutting speed*, p. 122.
ISBN-13: 9780750660730

4.2 Drilling Corners On Internal Features

When an internal feature such as a square needs to be filed it is first marked out as explained in previous units. The corners are centre punched and then a centre drill is used prior to drilling the holes at each corner. The holes provide clearance in the corners when filing and for the mating part.

4.3 Use Of Needle Files

The use of needle files as explained above is used when removing small amounts of metal and for adding chamfers and deburring sharp edges.

4.4 Deburring Sharp Edges

For removing burrs from holes a hand held deburring tools can be used. Needle files as explained above can be used when removing small amounts of metal and for adding chamfers and deburring sharp edges.

Ref: Black, Bruce J 2004, *Workshop processes, practices and materials*, 3rd edn, Elsevier Science & Technology, chapter 2, *Hand processes*, p. 30.
ISBN-13: 9780750660730

4.5 Removal Of Waste Material On Internal Features

A series of holes can be drilled inside the scribed lines and the remaining material can be removed by using a hacksaw or junior hacksaw. Hand files are then used to remove remaining material. Machine Files, which have attachments at both ends, can be clamped in a filing machine. The file is held in the middle of the machine table and reciprocates vertically while the workpiece is pushed against the file face to remove the required material.

4.6 Assembly Techniques, Use Of Feeler Gauges

Feeler gauges are normally available in a sets can be used to check the gap between the mating parts.

5.0 The Calculation Of Clearance Fits Between Mating Components

Key Learning Points

Use and care of the external micrometer. Use of slip gauge piles to gauge go and no-go conditions. Clearance fits, max and minimum allowances. Creation of slip gauge piles for gauging upper and lower limits of size of internal slots/features.

5.1 Use And Care Of The External Micrometer

The external micrometer is a precision instrument and is used to measure outside diameters, material thickness and length.

Ref: Black, Bruce J 2004, *Workshop processes, practices and materials*, 3rd edn, Elsevier Science & Technology, chapter 6, *Measuring equipment*, sec. 6.2, *Micrometers*, p. 101.
ISBN-13: 9780750660730

5.2 Use Of Slip Gauge Piles To Gauge Go And No-Go Conditions

A set of slip gauges consist of a range or varying size blocks that can be built up to create the upper and lower limits of the slot be checked.

Ref: Black, Bruce J 2004, *Workshop processes, practices and materials*, 3rd edn, Elsevier Science & Technology, chapter 5, *Standards, measurements and gauging*, p. 70.
ISBN-13: 9780750660730

5.3 Clearance Fits, Max And Minimum Allowances

A *clearance fit* is when, for example, a shaft at the maximum tolerance is smaller than a hole that is at the minimum tolerance allowed.

Minimum allowance: This is the difference between the high limit of size of the shaft and the low limit of size of the hole.

Maximum allowance: This is the difference between the low limit of size of the shaft and the high limit of size of the hole.

5.4 Creation Of Slip Gauge Piles For Gauging Upper And Lower Limits Of Size Of Internal Slots/Features

The polished surfaces allow the blocks to be 'wrung' together to form a stack. The slip gauges that make up the lower limit can be used for the go gauge and upper limit stack of slip gauges are used as the no-go gauge.

Ref: Black, Bruce J 2004, *Workshop processes, practices and materials*, 3rd edn, Elsevier Science & Technology, chapter 5, *Standards, measurements and gauging*, p. 70.
ISBN-13: 9780750660730

Summary

Precision filing techniques: As explained in previous Units, all the basic filing techniques are used for precision filing. The hacksaw is used to cut the metal close to the scribed lines. Files such as smooth or dead smooth are used to file the internal shape close to the specified drawing dimensions. The vernier callipers is used to measure the internal shape of the component as it is filed close to the required dimension. At this stage slip gauges assembled to the maximum and minimum values for a specific dimension. Needle files are used to remove small amounts of material and can be used to remove material to form sharp corners. While filing it is important to continuously check the dimensions and also check that the filed surfaces are square.

Producing isometric and oblique sketches of components: Isometric and oblique sketches 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.

The significance of geometric tolerance: 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.

Filing and assembling components with the required clearance fit: When an internal feature such as a square needs to be filed it is first marked out as explained in previous units. The corners are centre punched and then a centre drill is used prior to drilling the holes at each corner. The holes provide clearance in the corners when filing and for the mating part. The use of needle files as explained above is used when removing small amounts of metal and for adding chamfers and deburring sharp edges. A series of holes can be drilled inside the scribed lines and the remaining material can be removed by using a hacksaw or junior hacksaw. Hand files are then used to remove remaining material. Feeler gauges can be used to check the gap between the mating parts**.** Machine Files, which have attachments at both ends, can be clamped in a filing machine. The file is held in the middle of the machine table and reciprocates vertically while the workpiece is pushed against the file face to remove the required material.

The calculation of clearance fits between mating components: A set of slip gauges consist of a range or varying size blocks that can be built up to create the upper and lower limits of the slot be checked. The polished surfaces allow the blocks to be wrung together to form a stack. The slip gauges that make up the lower limit can be used for the go gauge and upper limit stack of slip gauges are used as the no-go gauge.

Suggested Exercises

1. Explain the difference between the maximum and minimum material conditions.
2. Explain how calculate and build up a stacks of slip gauges to inspect a slot at the upper and lower limits.
3. Sketch the geometric tolerance symbols for Straightness, Flatness, Roundness, Parallelism, Angularity, Position, Symmetry and Profile of a line.
4. For what purpose are needle files used for.

Questions

1. Files are available in various shapes, list three types.
2. List three grades of files.
3. What is the purpose of Geometric Tolerancing?
4. Explain the term ‘maximum material condition’.
5. Explain the terms, Clearance Fit, Minimum Allowance and the Maximum Allowance.

Answers

1. A range of files are available in various shapes such as Flat, Hand, Warding, Square, Three Square and Half Round.
2. Files are cut to the following grades: Rough, Bastard, Second Cut, Smooth and Dead Smooth.
3. 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.
4. The maximum material condition (MMC) is when the part or a feature contains the maximum amount of material, e.g. a block of metal with a hole drilled through, where the external surfaces are at the maximum size allowed and the hole is at the minimum diameter allowed by the tolerance.
5. A clearance fit is when, for example, a shaft at the maximum tolerance is smaller than a hole that is at the minimum tolerance allowed.
   - *Minimum allowance:* This is the difference between the high limit of size of the shaft and the low limit of size of the hole.
   - *Maximum allowance:* This is the difference between the low limit of size of the shaft and the high limit of size of the hole.

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