

Trade of Toolmaking	
Module 2:	Turning
Unit 3:	Drilling, Reaming & Tapping
	Phase 2

Published by



Table of Contents

Document Release History	3
Unit Objective.....	4
Introduction.....	4
1.0 Drilling To Correct Spindle Speed And Feed Rate.....	5
1.1 Drilling Of Core Diameters For Tapping (Tapping Size).....	5
2.0 Tapping Holes On A Lathe	6
2.1 Use Of Zeus Chart Calculations For Tapping Sizes, Metric Threads	6
2.2 Use Of Tap Wrench On Lathe Tailstock And Chuck Plus Accessories, Thread Cutting Fluids	6
2.3 Sectional Views (Including Sectional Views Of Threads), End View, Symbols	6
2.4 Centre Lathe Operations: Concentricity And Alignment	7
3.0 Drilling And Reaming To Correct Spindle Speed And Feed Rate	8
3.1 Calculation Of Machine Speeds And Feed Rate For Drilling And Reaming	8
3.2 Drilling And Reaming Of Holes On The Centre Lathe, Using Parallel And Taper Shank Drills And Reamers	9
3.3 Reaming Allowance Appropriate To Hole Size	9
3.4 Use Of Coolants And Lubricants	9
Summary.....	10
Suggested Exercises	11
Questions.....	12
Answers.....	13
Recommended Additional Resources.....	14
Reference Books	14

Document Release History

Date	Version	Comments
25/09/2014	2.0	SOLAS transfer

Unit Objective

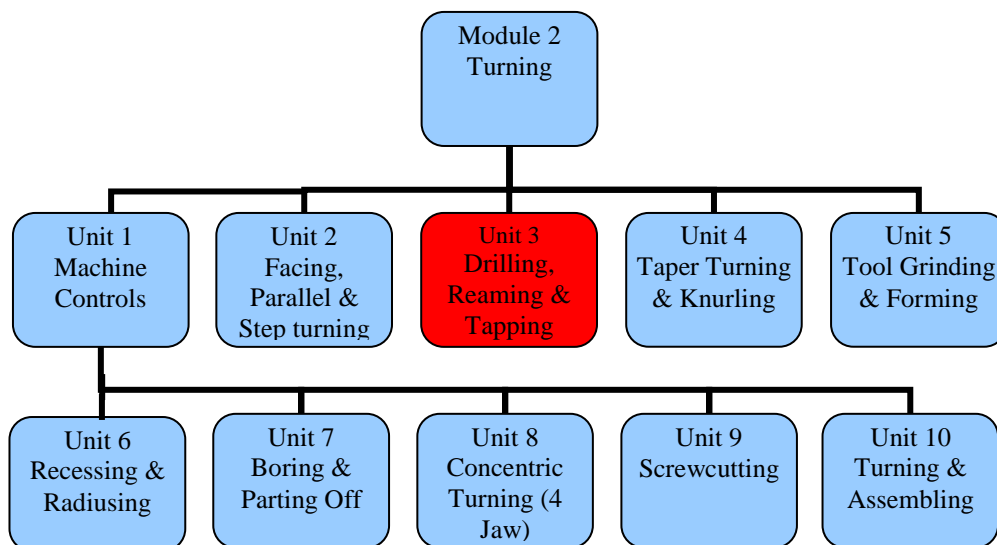
On completion of this unit you will be able to calculate the spindle speed for drilling, determine the correct tapping drill size and perform the tapping operation. You will also be able to drill and ream to the required size.

Introduction

Module two of this course covers turning. This is the third unit in module two and explains how to calculate the correct spindle speed for drilling on a lathe and the techniques associated with using the tailstock for drilling into the work piece to the required depth. This unit also explains how to determine the tapping hole size and how to perform the tapping operation on a lathe.

Where a higher degree of accuracy and surface finish is required, reamers are used. This unit explains how to determine the drill size, the spindle speed and feed prior to reaming the hole to the required dimension.

Tapping and reaming operations are performed throughout industry to allow mating components to be assembled accurately and securely. Operations explained in this unit are used for one off and low volume work, whereas for high volume requirements, CNC machines are generally used nowadays.



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

- Drill core (tapping size) diameters to depth using the correct spindle speed and feed rate.
- Tap holes on components while held in the lathe using the given tap wrench, taps and tailstock.
- Drill and ream to the required depths and diameters using the correct spindle speed and feed.

1.0 Drilling To Correct Spindle Speed And Feed Rate

Key Learning Points

Drilling of core diameters for tapping (tapping size).

1.1 Drilling Of Core Diameters For Tapping (Tapping Size)

Cutting speed is expressed in meters per minute. This refers to the distance covered by the tool across the material when machining. A data chart is available listing various materials and their corresponding cutting speeds. The revolutions per minute (RPM) is calculated by entering the cutting speed and the diameter of the material into the RPM formula.

The centre drill is held in the drill chuck, which in turn is held in the tailstock. It is important to use coolant and feed the center drill slowly into the work piece. Graduations on the side of the barrel can be used to drill to the required depth. A drill or a series of drills are then used to bore out the work piece to the required diameter. It is important to withdraw the drill frequently to remove swarf and allow coolant to enter the drill hole.

Ref: Black, Bruce J 2004, *Workshop processes, practices and materials*, 3rd edn, Elsevier Science & Technology, chapter 18, *Drilling*, p. 129.
ISBN-13: 9780750660730

2.0 Tapping Holes On A Lathe

Key Learning Points

Use of Zeus Chart calculations for tapping sizes, metric threads. Use of tap wrench on lathe tailstock and chuck plus accessories, thread cutting fluids. Sectional views (including sectional views of threads), end view, symbols. Centre lathe operations: concentricity and alignment.

2.1 Use Of Zeus Chart Calculations For Tapping Sizes, Metric Threads

The Zeus Chart can be used to determine the tapping drill size prior the tapping the thread. If, for example, the drawing calls for a 6mm internal thread, then this will be shown on the drawing view as M6 x 1.0. This is a Ø6mm metric thread with a 1mm pitch. The pitch is the distance from the top of one thread to the next thread. The tapping drill size is the core diameter of an internal thread, which needs to be drilled prior to performing the tapping operation. The next step is to go to the metric thread section on the Zeus Chart, which will show the screw thread and pitch required and also the tapping drill size.

Ref: Zeus Precision Charts Ltd. 2007, Zeus precision data charts and reference tables for drawing office, toolroom & workshop, 2007 edn, Zeus Precision Charts Ltd.
ASIN: B0000CLZUO

2.2 Use Of Tap Wrench On Lathe Tailstock And Chuck Plus Accessories, Thread Cutting Fluids

The tapered end of the tap allows the tap to be entered in to the pre-drilled hole and makes it easier to start the thread. It is important use a suitable lubricant on the tap. A tap handle is used to rotate the tap to cut the thread and the centre held in the tailstock will help to keep the tap centred. It is important to reverse and withdraw the tap frequently in order to clear out and break the metal chips. The above steps are repeated with the Plug and Finishing taps until the required hole is tapped.

Ref: Black, Bruce J 2004, *Workshop processes, practices and materials*, 3rd edn, Elsevier Science & Technology, chapter 2, Hand processes, p. 39.
ISBN-13: 9780750660730

2.3 Sectional Views (Including Sectional Views Of Threads), End View, Symbols

When a feature on a component is hidden or is inside the component and cannot be shown clearly on one of the projected views, then a sectioned view is taken through the part, showing the sectioned view at a right angle to the view.

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

2.4 Centre Lathe Operations: Concentricity And Alignment

For parts that require a number of different diameters that need to be concentric, then turning should be performed in one setting without removing the part from the chuck. If a part is removed from a chuck and put back in again, then accuracy is lost. If a part needs to be aligned accurately it can then setup in a 4-jaw chuck and a dial indicator is used to centre it.

Ref: Black, Bruce J 2004, *Workshop processes, practices and materials*, 3rd edn, Elsevier Science & Technology, chapter 9, Turning., p. 151.
ISBN-13: 9780750660730

3.0 Drilling And Reaming To Correct Spindle Speed And Feed Rate

Key Learning Points

Calculation of machine speeds and feed rate for drilling and reaming. Drilling and reaming of holes on the centre lathe, using parallel and taper shank drills and reamers. Reaming allowance appropriate to hole size. Use of coolants and lubricants.

3.1 Calculation Of Machine Speeds And Feed Rate For Drilling And Reaming

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 metres/min is used for mild steel and cutting speed is 20 metres/min is used 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

Drills

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

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

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

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 20mm High Speed Steel Drill, the RPM is calculated as follows:

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

Feed Rate

Feed rate is the speed at which the workpiece is moved relative to the cutter.

A value of 0.05 is chosen from the manufacturers tables. The feed rate is calculated as follows:

$$\begin{aligned}\text{Feed Rate} &= \text{feed/tooth} \times \text{No. of cutting teeth} \times \text{RPM} \\ &= 0.05 \times 2 \times 478 = 48 \text{ mm/min}\end{aligned}$$

Reamers

When using *reamers* use half the speed of a corresponding size drill and double the feed rate.

3.2 Drilling And Reaming Of Holes On The Centre Lathe, Using Parallel And Taper Shank Drills And Reamers

Drills and reamers are available with straight shank and taper shanks. The straight shank is held in a drill chuck. The chuck is fitted with a Morse tapered shank, which fits into a corresponding morse taper in the spindle of the tailstock. A tapered drill shank can be fitted directly into the tapered spindle of the tailstock. To remove the Morse tapered shank the quill is wound back with the feed handle until tang at the top of the tapered shank hits the back of the tailstock, which caused the tapers to disengage.

Ref: Black, Bruce J 2004, *Workshop processes, practices and materials*, 3rd edn, Elsevier Science & Technology, chapter 8, *Drilling*, p. 131.
ISBN-13: 9780750660730

3.3 Reaming Allowance Appropriate To Hole Size

The purpose of a reamer is to produce a hole of high accuracy and to a good finish. The holes are drilled to a diameter, which is slightly less than the finished dimension required, e.g. for up to Ø10mm holes the amount of stock that needs to be removed by the reamer is approximately 0.3mm.

3.4 Use Of Coolants And Lubricants

When the component is being drilled, heat will be generated at the cutting edge, due to frictional forces. This can be avoided by using coolant which cools and lubricates the work and drill. The advantages of coolant are: reduced wear on the drill, drilling speeds and feeds can be increased and a better surface finish is produced.

Ref: Black, Bruce J 2004, *Workshop processes, practices and materials*, 3rd edn, Elsevier Science & Technology, chapter 7, *Cutting tools and cutting fluids*, p. 124.
ISBN-13: 9780750660730

Summary

Drilling to correct spindle speed and feed rate: Cutting speed is expressed in feet per minute or meters per minute. This refers to the distance covered by the tool across the material when machining. A chart is available listing various materials and their corresponding cutting speeds. The revolutions per minute (RPM) is calculated by entering the cutting speed and the diameter of the material into the RPM formula.

The centre drill is held in the drill chuck, which in turn is held in the tailstock. It is important to use coolant and feed the center drill slowly into the work piece. Graduations on the side of the barrel can be used to drill to the required depth. A drill or a series of drills are then used to bore out the work piece to the required diameter. It is important to withdraw the drill frequently to remove swarf and allow coolant to enter the drill hole.

Tapping holes on a lathe: The Zeus Chart can be used to determine the tapping drill size prior the tapping the thread. If, for example, the drawing calls for a 6mm internal thread, then this will be shown on the drawing view as M6 x 1.0. This means that is a Ø6mm metric thread with a 1mm pitch. The pitch is the distance from the top of one thread to the next thread. The tapping drill size is the core diameter of an internal thread, which needs to be drilled prior to performing the tapping operation. The next step is to go to the metric thread section on the Zeus Chart, which will show the screw thread and pitch required and also the tapping drill size.

The tapered end of the tap allows the tap to be entered in to the pre-drilled hole and makes it easier to start the thread. It is important use a suitable lubricant on the tap. A tap handle is used to rotate the tap to cut the thread and the centre held in the tailstock will help to keep the tap centred. It is important to reverse and withdraw the tap frequently in order to clear out and break the metal chips. The above steps are repeated with the Plug and Finishing taps until the required hole is tapped.

Drilling and reaming to correct spindle speed and feed rate: A reamer is used to produce a hole to a higher degree of accuracy and surface finish than that can be produced by a drill. It is used for enlarging and finishing to size after a hole has been drilled. The recommended amount of stock that a reamer can remove is 0.3 to 0.5mm depending on the size of the hole being reamed. The calculation for spindle speeds for drilling has been explained above. For reaming, a general rule is to reduce the speed by at least a half and double the feed rate to that of a similar size drill. The reamer normally has a taper shank and is held in the tapered bore of the tailstock. The reamer is fed into the drilled hole using coolant.

Suggested Exercises

1. Setup a mild steel bar in the chuck and drill an Ø8mm hole, use the RPM formula to calculate the correct spindle speed.
2. Drill the Ø8mm hole to a depth of 35mm, using the graduations on the tailstock.
3. Setup a mild steel bar in the chuck and drill a tapping hole for an M8 thread. Use the Zeus book to determine the tapping drill size.
4. Tap the workpiece using an M8 set of taps and a tap wrench.
5. What size hole is drilled when a Ø8mm reamer is being used. Also calculate the spindle speed for both the drill and the reamer.

Questions

1. Use the Zeus Book to determine the tapping drill size for an M6 screw thread.
2. What type of shanks do drills and reamers have?
3. Explain how drills and reamers are held when machining.
4. What are the advantages of using coolant when drilling and reaming?
5. Explain how to tap a threaded hole on a lathe using the tailstock centre.

Answers

1. The tapping drill size for an M6 screw thread is Ø5.0mm (see Zeus Book).
2. Drills and reamers are available with straight shank and taper shanks.
3. The straight shank is held in a drill chuck. The chuck has a Morse tapered shank, which fits into a corresponding Morse taper in the spindle of the tailstock. A tapered drill shank can be fitted directly into the tapered spindle of the tailstock. For smaller drills, Morse tapered sleeves may need to be used.
4. When the component is being drilled, heat will be generated at the cutting edge, due to frictional forces. This can be avoided by using coolant which cools and lubricates the work and drill. The advantages of coolant are: reduced wear on the drill, drilling speeds and feeds can be increased and a better surface finish is produced.
5. A suitable lubricant should be used when tapping screw threads. The tapered tap is assembled first with the tapping handle, which is used to rotate the tap to cut the thread. The tailstock centre is aligned with the centre hole at the back of the tap, which will help to keep the tap centred. It is important to reverse and withdraw the tap frequently in order to clear out and break the metal chips. The above steps are repeated with the Plug and Finishing taps until the required hole is tapped.

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

Zeus Precision Charts Ltd. 2007, *Zeus precision data charts and reference tables for drawing office, toolroom & workshop*, 2007 edn, Zeus Precision Charts Ltd.

ASIN: B0000CLZUO