Hole Making

Week 4 Test

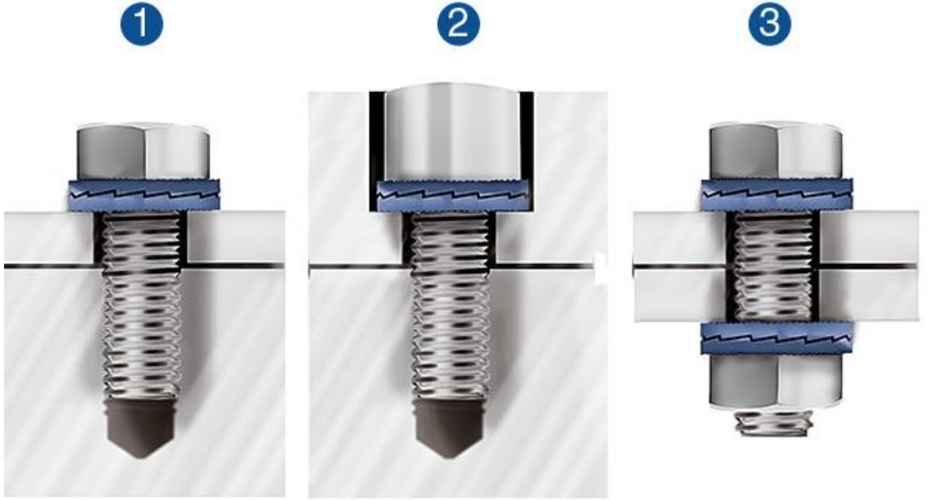
- The test paper and submission boxes will become available in Moodle at 3pm, Week 4 Friday.
  - The submission boxes will close at 5pm that same day!
- The test should take approximately 60-90 minutes to complete. However, you will have the full two hours to complete the test and submit your files.
- The topics covered in this test will be:
  - Engineering standards
  - Engineering drawings
- There will be a mix of short answer questions as well as a practical drawings component that must be completed using Solidworks.
- You will not be required to create 3D models for this test.

Topics

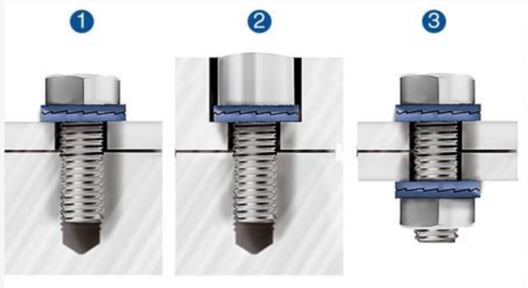
- Drilling
- Drills
- Drilling Machines
- Material-removal rate (MRR)
- Reaming and Reamers
- Drilling practice
- Design considerations for drilling
- Tapping and Taps
- ISO Metric Screws

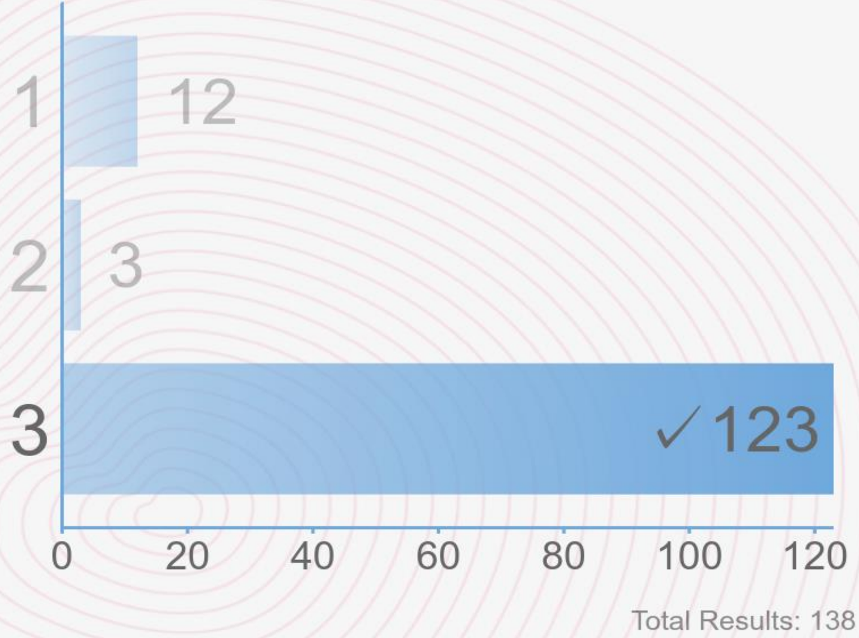
Drilling

- I want you to cast your mind and really think about any machine or product you have seen lately. How was it assembled?
- Chances are it employed holes to enable various fasteners
  - Bolts, screws, rivets



Which one is a bolt?

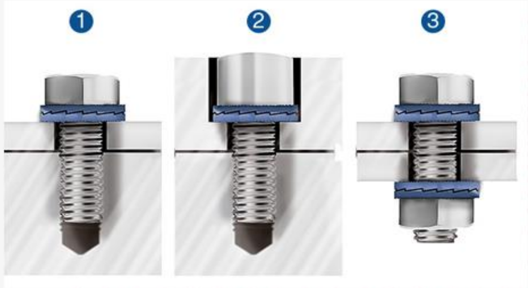


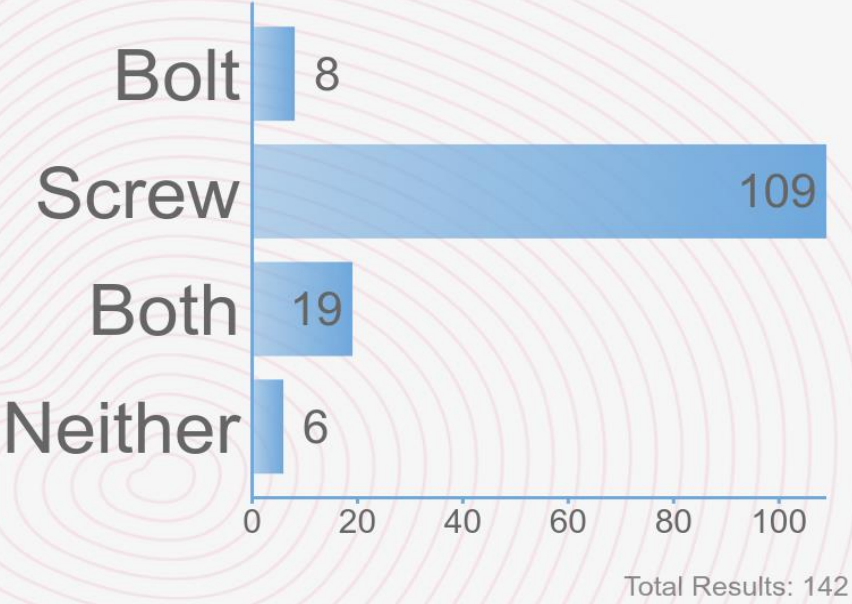


Powered by Poll Everywhere

Start the presentation to see live content. For screen share software, share the entire screen. Get help at **pollev.com/app**

Is the fastener depicted in Option (1) a bolt or a screw?





Powered by Poll Everywhere

Start the presentation to see live content. For screen share software, share the entire screen. Get help at **pollev.com/app**

Apart from enabling the use of fasteners, why else might we want to make holes in our designs?

UNSW SYDNEY

Top

42	Remove excess weight
39	running wires through
34	To save weight

New

-1	Depends on purpose
-1	reduce weight
-1	why wouldnt it be fun? why dislike?

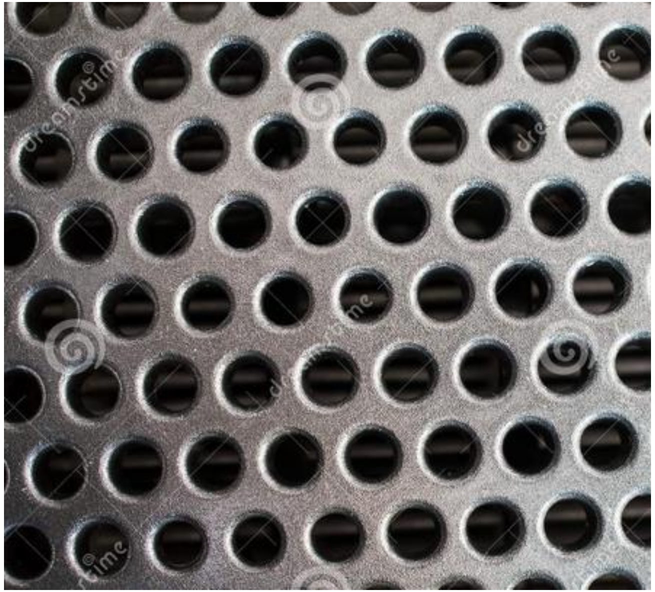
Total Results: 132

Powered by Poll Everywhere

Start the presentation to see live content. For screen share software, share the entire screen. Get help at **pollev.com/app**

Drilling

- Examples of hole making for design purposes



Drilling Machines

- Whilst it is possible to create holes using hand-held machines, typically we employ larger machinery.
- The most common machine is the drill press.
- Securing the workpiece is done either by
  - Clamping directly to the adjustable table
  - Held in a vise, then clamped to the adjustable table
- The rotating drill is lowered to the workpiece manually by the handwheel or automatically using a power feed with preset rates.

Drilling Machines

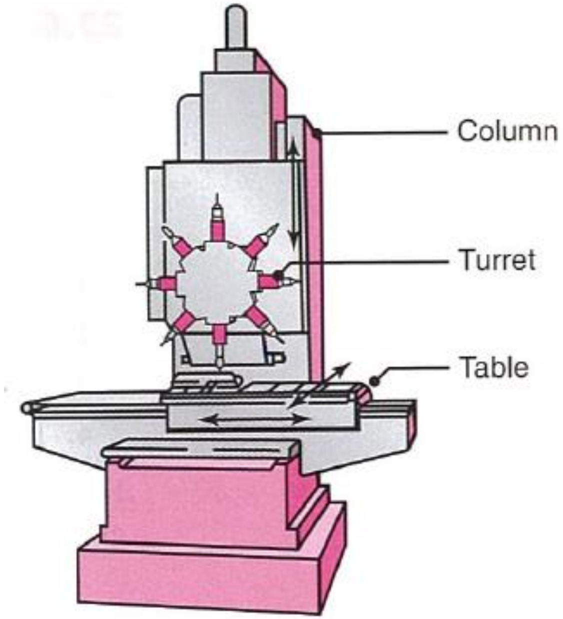
- Drill presses are usually designated by the maximum workpiece diameter
- Cutting speeds must be varied depending on drill size and material.
  - This is achieved by pulleys, gearboxes or variable speed motors.
- There are three main types of drilling machines
  - Vertical drill press

Drilling Machines

- Drill presses are usually designated by the maximum workpiece diameter
- Cutting speeds must be varied depending on drill size and material.
  - This is achieved by pulleys, gearboxes or variable speed motors.
- There are three main types of drilling machines
  - Vertical drill press
  - Radial drill press

Drilling Machines

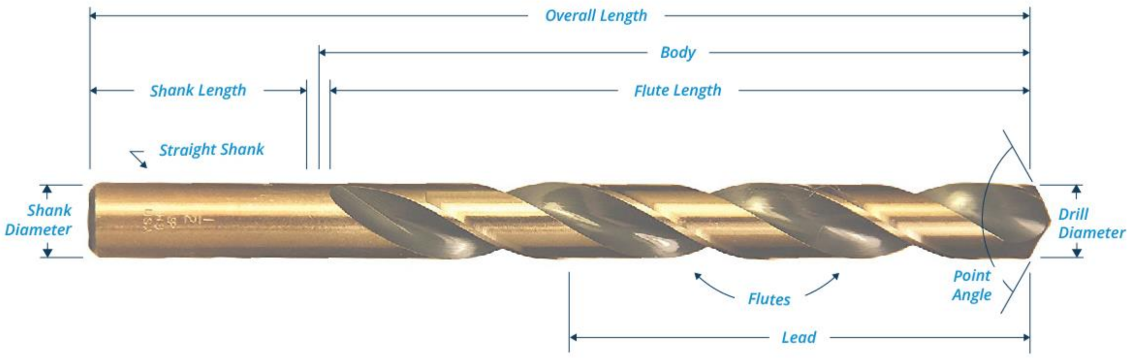
- Drill presses are usually designated by the maximum workpiece diameter
- Cutting speeds must be varied depending on drill size and material.
  - This is achieved by pulleys, gearboxes or variable speed motors.
- There are three main types of drilling machines
  - Vertical drill press
  - Radial drill press
  - CNC drilling machines



Three axis CNC drilling machine

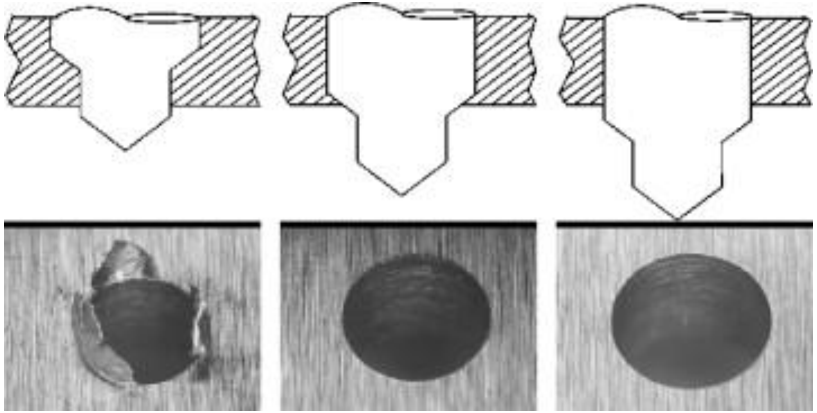
Drills

- Drills have a high length-to-diameter ratio and thus are capable of making deep holes.
- Drills have three main features I'd like you to know
  - Drill tip (where the cutting happens)
  - Flute (chip guide/cutting fluid pathway)
  - Shank (where the machine/chuck grips the drill)

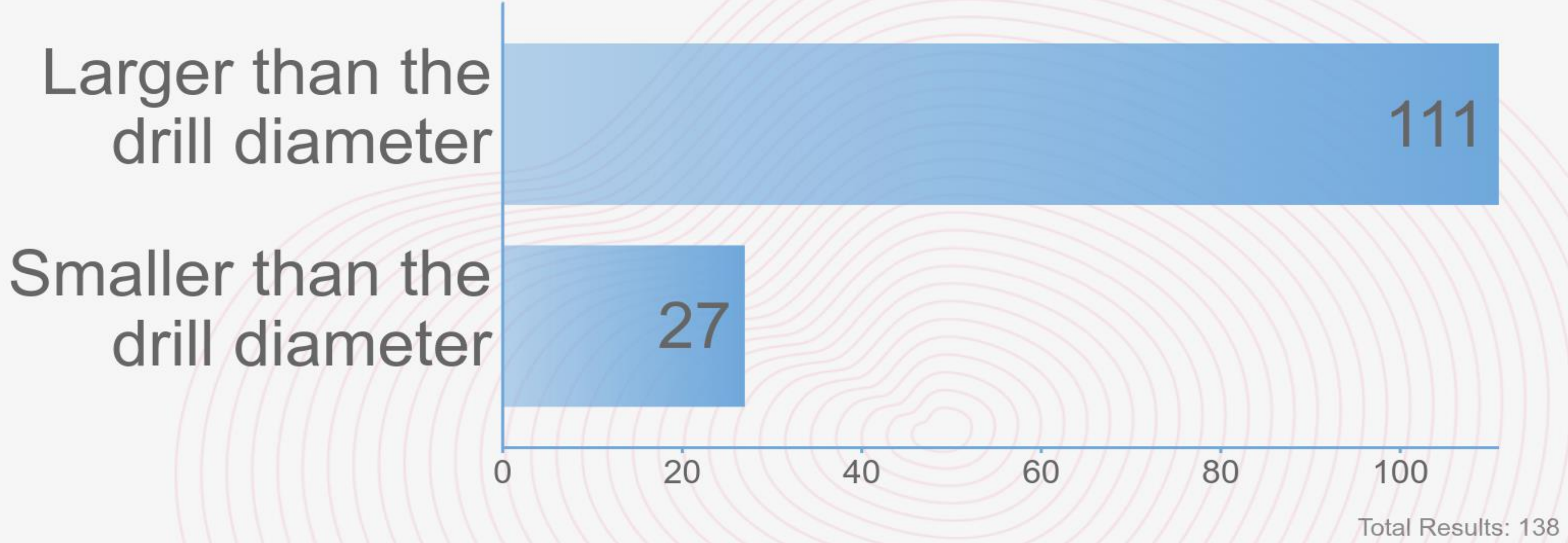


Drills

- Chips that are produced move in the opposite direction to the forward motion of the drill
  - This can cause difficulties for effective chip removal and cutting-fluid application
  - https://www.youtube.com/watch?v=3Lytkc2HXj8
- Drills generally leave a “burr” on the bottom surface upon breakthrough.
- Note, drills can bend when in use so care must be taken to drill holes accurately and prevent breakage



After the drilling operation, what would you expect the size of the hole to be?



Powered by Poll Everywhere

Start the presentation to see live content. For screen share software, share the entire screen. Get help at **pollev.com/app**

Drills

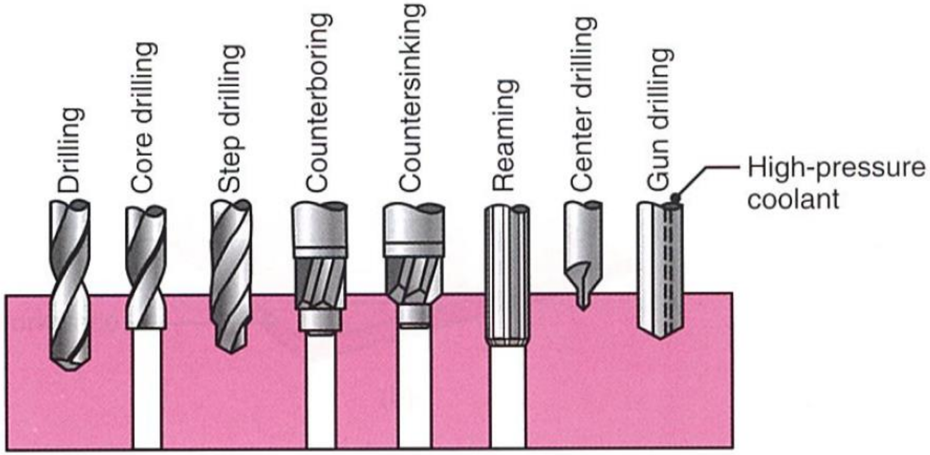
Twist drill

- Most common drill is a twist drill.
- Two spiral grooves (flutes) run the length of the drill.
  - chips produced are guided upwards
  - cutting fluid is allowed passage downwards
- Some drills have internal channels for cutting fluids to assist cooling and lubrication
- Drills that are on automated machines often employ chip-breakers.
- https://www.youtube.com/watch?v=60052AWpRkk

Drills

<u>Other types of drills</u>

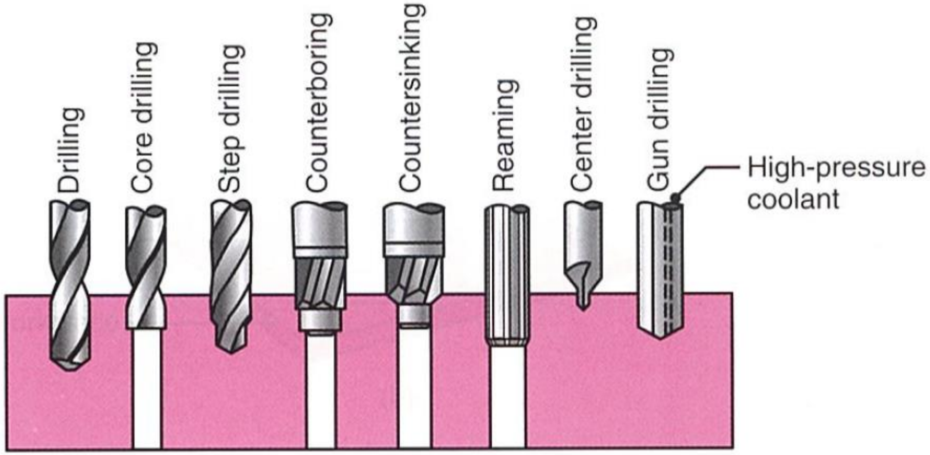
- Step drill
  - Produces hole with two or more diameters
- Core drill
  - Enlarges existing hole
- Counterboring and countersinking
  - Creates depressions to accommodate bolt and screw heads



Drills

<u>Other types of drills</u>

- Center drill
  - Used to create a mounting point for a workpiece due for turning
  - Starts holes to avoid drill walk



Drills

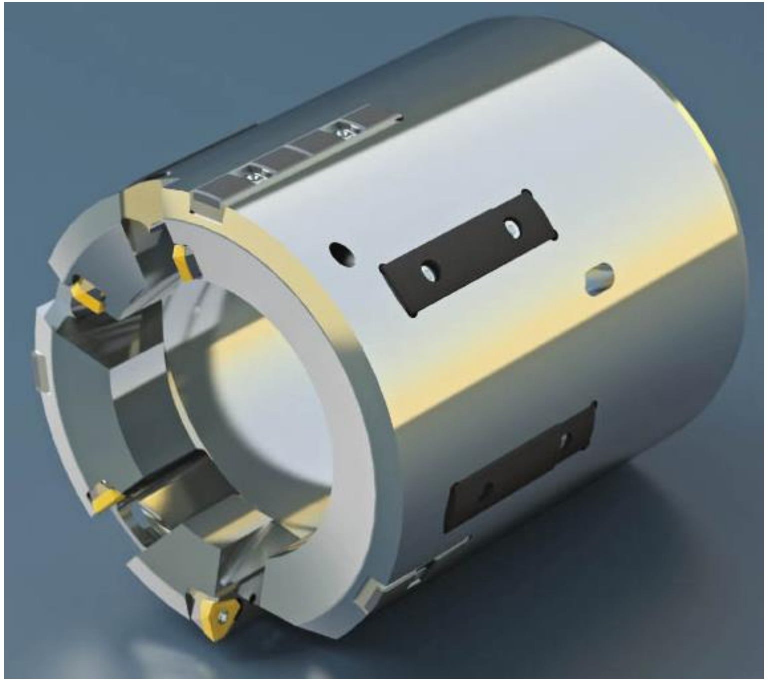
Other types of drills

- Center drill
  - Used to create a mounting point for a workpiece due for turning
  - Starts holes to avoid drill walk
- Spade drill
  - Has removable tips or bits
  - Create large diameter and deep holes
  - Stronger than twist drills due to lack of flutes

Drills

<u>Other types of drills</u>

- Trepanning
  - Uses a cutting tool that removes a disc shaped piece (core
  - Diameters up to 250mm can be created
  - Can also be used to create circular grooves for O-rings
  - https://www.youtube.com/watch?v=G0iz5uKVe9M

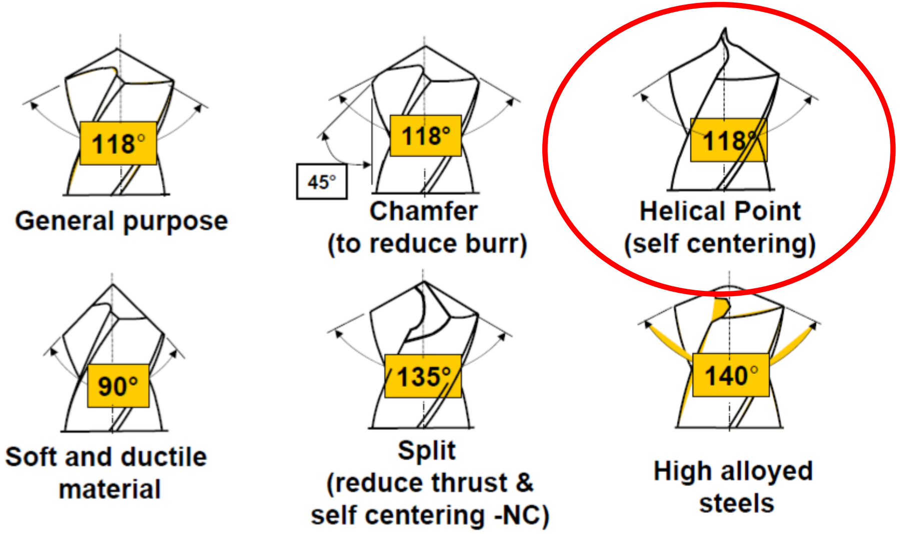


Drilling Practice

- Drill are held in “drill chucks” and may be tightened with or without keys
- A typical drill does not have a centering action when it first begins cutting
  - This causes the drill to “walk”.
- This can cause accuracy issues and for small-diameter drills this can ultimately cause drill failure
- To avoid “walk”, we often employ a starting hole created with a center drill.
  - Center drills usually employ point angles of 60 degrees

Drilling Practice

- Drill points may also be ground to an S-shape (helical or spiral point).
  - These drills are self-centering.
  - Very important for automated production with CNC machines.



Drilling Practice

- The cutting speed and feed is very important in metalworking.
- The mathematical analysis is very involved involving energy density of materials, solid mechanics, power analysis and material science.
  - Don’t worry, this is outside the scope of this course.

General Recommendations for Speeds and Feeds in Drilling

		Drill diameter			
		Feed, mm/rev		Speed, rpm	
Workpiece material	Surface speed m/min	1.5 mm	12.5 mm	1.5 mm	12.5 mm
Aluminum alloys	30–120	0.025	0.30	6400–25,000	800–3000
Magnesium alloys	45–120	0.025	0.30	9600–25,000	1100–3000
Copper alloys	15–60	0.025	0.25	3200–12,000	400–1500
Steels	20–30	0.025	0.30	4300–6400	500–800
Stainless steels	10–20	0.025	0.18	2100–4300	250–500
Titanium alloys	6–20	0.010	0.15	1300–4300	150–500
Cast irons	20–60	0.025	0.30	4300–12,000	500–1500
Thermoplastics	30–60	0.025	0.13	6400–12,000	800–1500
Thermosets	20–60	0.025	0.10	4300–12,000	500–1500

Note: As hole depth increases, speeds and feeds should be reduced. The selection of speeds and feeds also depends on the specific surface finish required.

Material-removal rate

- The material-removal rate (MRR) is volume of material removed per unit time.

$$MRR = \left(\frac{\pi D^2}{4}\right) f N$$

- D – diameter of the drill [mm^2]
- f – feed speed (distance the drill penetrates per unit revolution) [mm/rev]
- N – rotational speed in RPM [rev/min]

Drilling Design Considerations

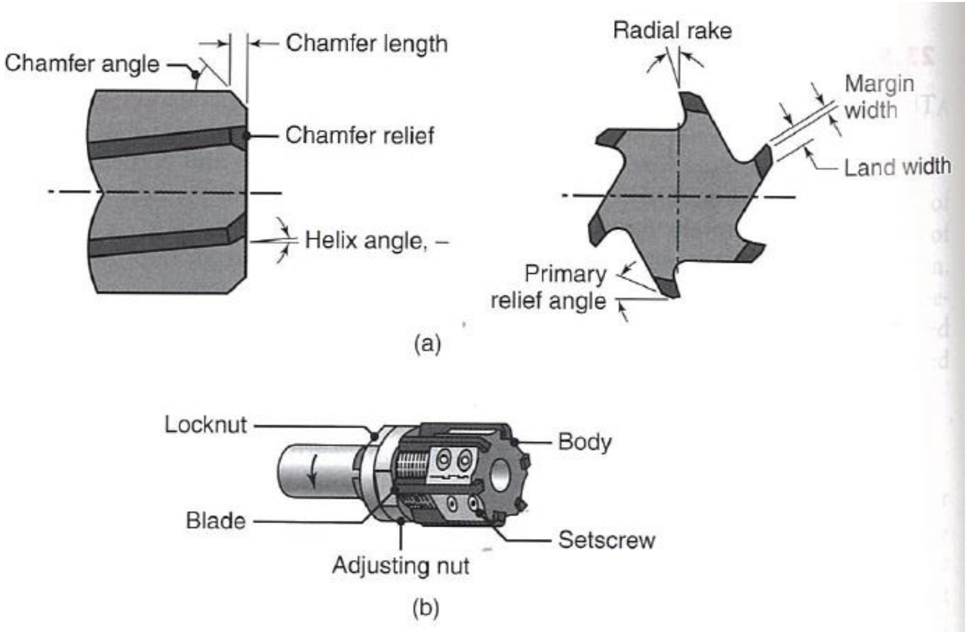
- Working with metal is unforgiving, so we need to think about how best to approach hole making.
- Designs should allow holes to be drilled on flat surfaces and perpendicular to the drill motion.
  - https://www.youtube.com/watch?v=Z7pclwu-7Bk
- Hole bottoms should match the drill point. Flat bottoms should be avoided
- Through holes are preferred to blind holes.
- Large diameter holes should utilise a pre-existing hole made during the fabrication process
- Blind holes must be drilled deeper than subsequent reaming or tapping operations

Reaming and Reamers

- It is often desired to improve the quality of a hole in terms of its dimensional accuracy or surface finish.
  - This is known as reaming.
- Hole accuracy is improved by using the following order of operations
  1. Centering
  2. Drilling
  3. Boring
  4. Reaming
- For even better accuracy and finish, additional operations such as burnishing, internal grounding and honing may be done.

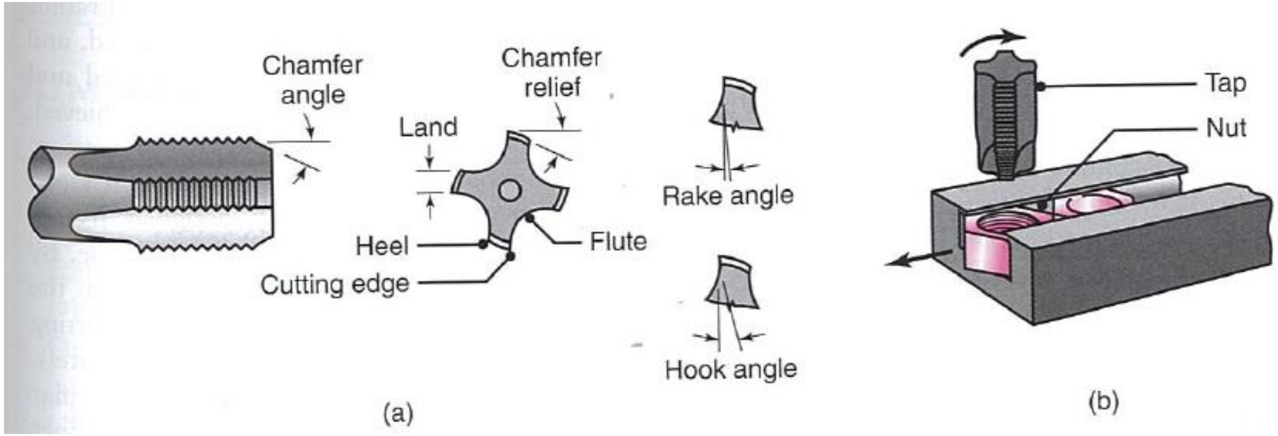
Reaming and Reamers

- Reaming employs a multiple-cutting-edge tool with either straight or helical flutes
- Each cutting faces removes very little material
  - Soft metals: 0.2 mm
  - Hard metals: 0.13 mm
- Removing less material is not advised as risk of burning increases.
- Reaming speed is one-half that of the same-size drill and three times the feed rate.



Tapping

- In order to use screws as a fastening method, we need to create internal threads within the material.
- This can be produced by tapping.
- A tap is a chip-producing threading tool with multiple cutting teeth.



Tapping

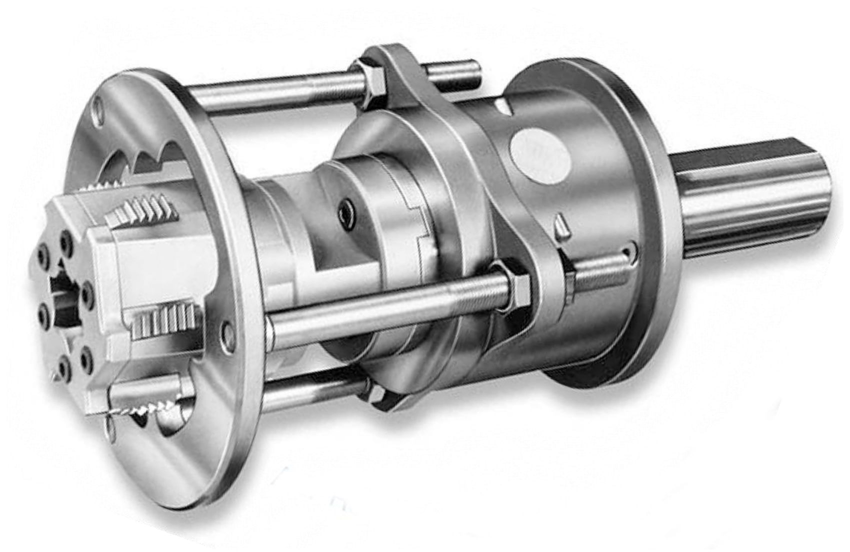
- The most common tap is the two-flute spiral point tap
- Taps are also available in three or four flute variants. The more flutes, the stronger the taps are.
- Tap sizes range up to 100mm diameter
- There are a few types of taps available
  - Tapered taps: designed to reduce torque required for through hole tapping
  - Bottoming taps: designed to tap blind holes to their full depth

Tapping

- Chip removal is a significant issue during tapping.
- Chip build up can result in a torque increase that results in tap breakage.
- This is typically managed by employing a cutting fluid and periodic reversal of the tap to remove the chips.
- For higher tapping productive, we can employ drapping!
- https://www.youtube.com/watch?v=misfp9RHRY0

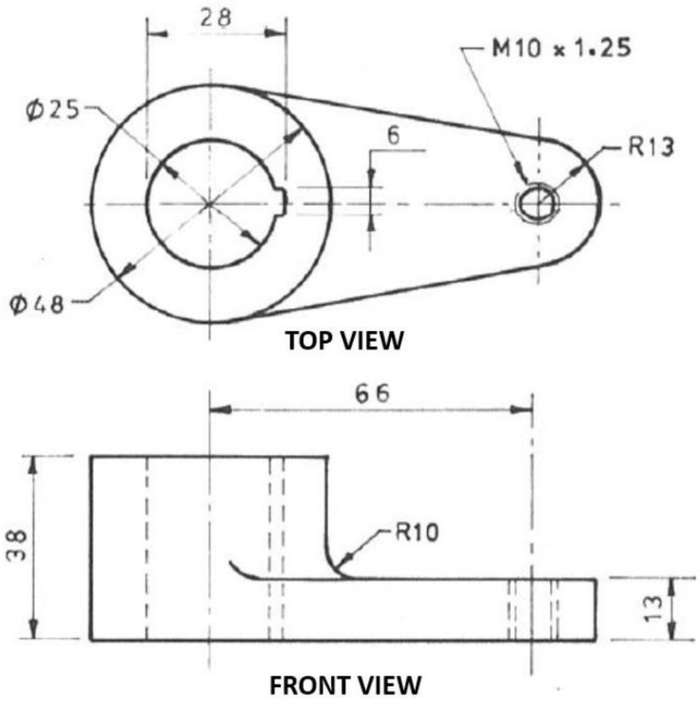
Tapping

- Collapsible taps
  - used for large diameter holes.
  - These collapse down and are removed from the hole without rotation.
- https://www.youtube.com/watch?v=wIEi1ElHa6w
  - 13:21 – 15:01

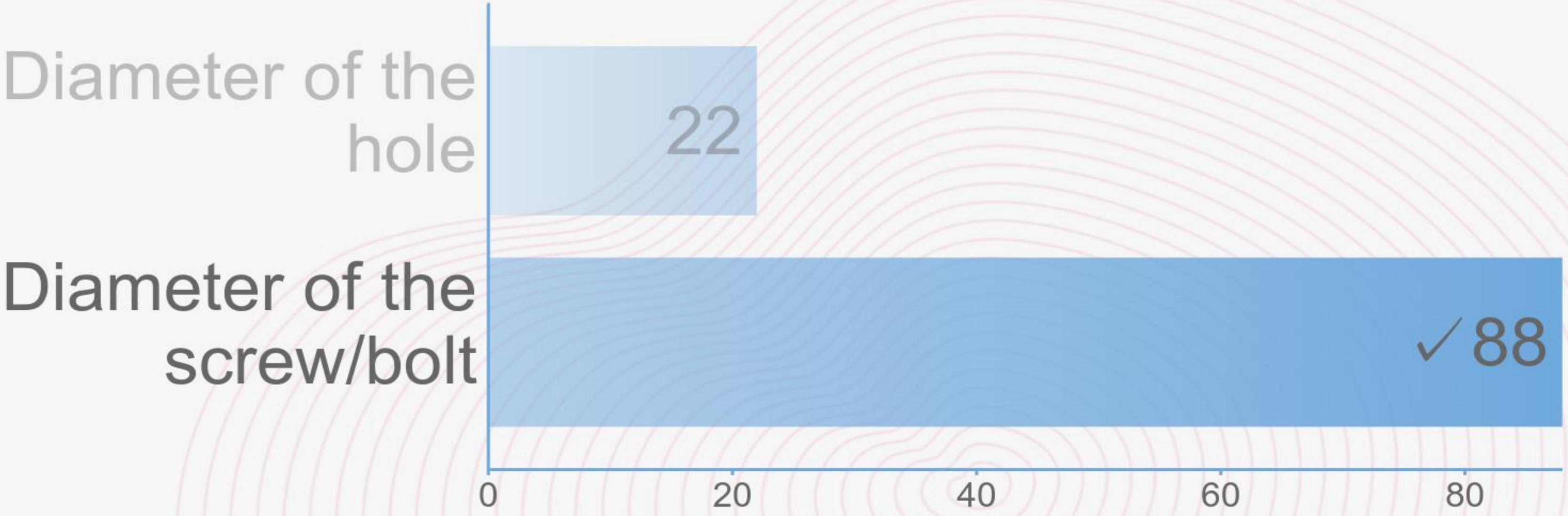


ISO Metric Screw Threads

- The most common screw designation system is known as the ISO Metric Screw Thread.
- You have already come across this in one of your CAD lab exercises.
- Let’s do a quick sanity check to see what we know about the realities of tapping holes…



What does the "10" in "M10" refer to?

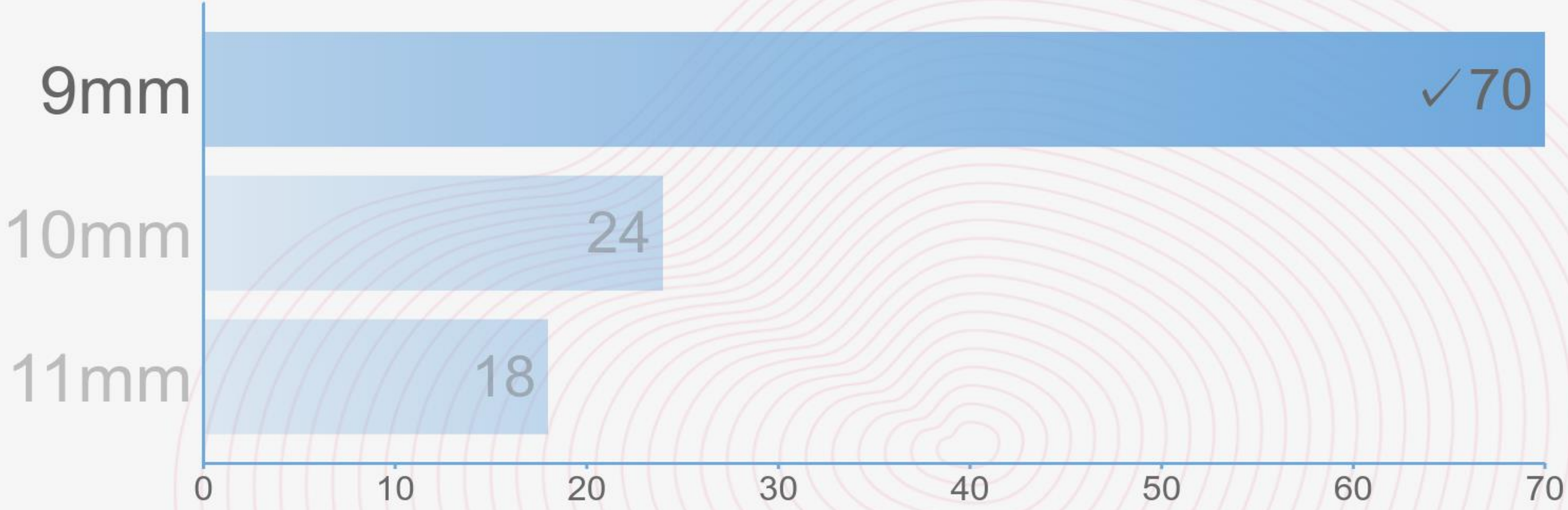


Total Results: 110

Powered by Poll Everywhere

Start the presentation to see live content. For screen share software, share the entire screen. Get help at **pollev.com/app**

What diameter hole must we drill before we can tap an M10 thread?

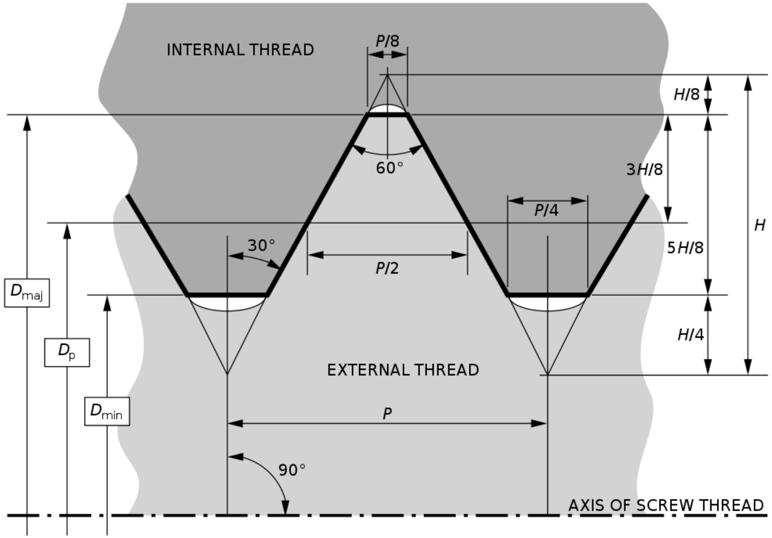


Total Results: 112

Powered by Poll Everywhere
Start the presentation to see live content. For screen share software, share the entire screen. Get help at **pollev.com/app**

ISO Metric Screw Threads

- Let's break down ISO Metric designation
- MD_{maj} x p x l
  - M: Metric
  - D_{maj}: Nominal diameter of the bolt/screw
  - p: the pitch
  - l: length of the thread
- M10 x 1.25 x 20
- Remember, the diameter of the hole must always be smaller than the thread you want to tap



What’s on next week?

- Tutorial
  - Engineers, Technical Operators and the Drawings Bridge That Binds Them

- CAD/CAM Consultation Labs
  - Engineering Drawings

- Lectures
  - Wednesday Lecture: Turning Processes
  - Friday Lecture: Milling Processes