

8

Hole-Making Operations

Objective

The learner will be able to:

- Differentiate between hole-making operations and hole modification operations used in machining
- Recognize tools used in hole-making operations
- Explain thread making processes
- Describe various hole-making operations

Orienting Questions

- ✓ What are hole-making tools?
- ✓ What is the difference between hole-making and hole-modification?
- ✓ Which tools are used in hole-modification?

Helpful Tips

- ✓ You can select the **HIGHLIGHTED TERMS** to read more about it.
- ✓ If needed, there are **CLOSED CAPTION** buttons  on the YouTube videos that will enable you to read along while you watch. The Closed Caption buttons are located bottom right of the video screen.
- ✓ Take time and explore about the subject by selecting the **EXPLORE** links in each section of this module.



DRILLING

DRILLING is a process where a cylindrical cutting tool, which is sharpened on its end, is rotated and forced into material to produce a hole. It is the most efficient way of producing holes in work pieces (see **Figure 1**). They can be mounted in hand held drill motors or milling machines and drill presses.

Explore: Video Below demonstrates Basic Drilling Technique



Figure 1: Basic Drilling Technique (Video by Eddie Humphries, 2013)

CENTER DRILLS

Before a hole can be drilled in a precise location, it is highly recommended that an indentation be made marking that location. This can be done with a center punch however the center punch makes crude indentations. The more common method in machining is to use a **CENTER DRILL** or spot drill. The center drill is a hole-starting tool that has both a pilot drill and countersink (See **Figure 2**). This will help the drill to center itself into the hole and prevent “walking”. When walking occurs, the drill will drift off location producing an inaccurate hole.



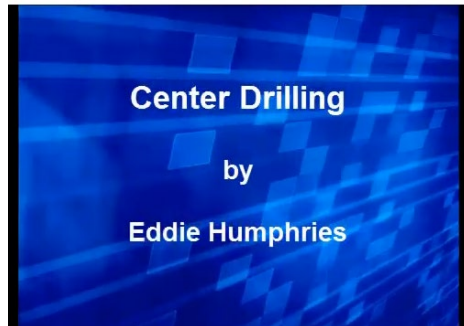
Explore: Video Below demonstrates Center Drilling Technique

Figure 2: Center Drilling (video by Eddie Humphries, 2013)

TWIST DRILLS

Any hand drilling can be done with a general purpose **TWIST DRILL**. Twist drills get their name from the cutting flutes which spiral up the body of the drill. While making the drill appear twisted, these flutes enable the drill to evacuate chips from the hole that is being produced.

Twist drills are sized by diameter. There are four classes of designation:

- Letter sizes – drill sizes are arranged by letter from “A” (0.234”) to “Z” (0.413).
- Number sizes – drill sizes range from #1 (0.2280”) to #80 (0.0130”).
- Fractional sizes – drill sizes range from 1/64” to 2.5” in 1/64” increments.
- Metric sizes – drill sizes range from 0.050 mm to 32 mm.

It is important to know that, although diameter sizes are stamped on drills, they always cut a slightly larger hole than designated. There are other variables to consider when an over-sized hole results. Drill bit condition, material, and the cutting method are the basic variables that can be considered.

Drill bits will have the tendency to “grab”, when passing through the material completely. Holes that pass through the material completely are called “through holes”. It is very important to be aware of this when drilling through thinner and softer materials. As a rule, pressure should be reduced when the drill is about to break through the material.



HOLE MODIFICATION

HOLE modification could be described as the process by which the properties of a plain hole are changed. The basic operations that are utilized to modify holes are counter-boring, counter-sinking, and spot-facing. These are not tools that can be used to create new holes but rather enhance pre-existing holes.

COUNTER-BORING

COUNTER-BORING is a process that increases the diameter of a hole to a certain depth. This is sometimes necessary when the head of a bolt needs to be recessed for clearance. The depth that is counter-bored should be flush or slightly below the work piece surface (see **Figure 3**). Counter-bore tools have a pilot that aligns and guides the tool into the hole that was previously drilled. The pilot diameter is typically .003” to .005” smaller than the existing hole diameter. It is important to have the right amount of clearance for the pilot. Too much clearance will cause the tool to walk off the location and too little clearance will cause binding and prevent the tool from performing the operation.

Counter-bore tools have sizes that are stamped on their shank that identify the size bolt that can be used after the hole has been recessed.

Explore: Video below demonstrates basic drilling and Counter-Boring Technique



Figure 3: Basic Drilling and Counter-Boring Technique (Video by Eddie Humphries, 2013)



SPOT-FACING

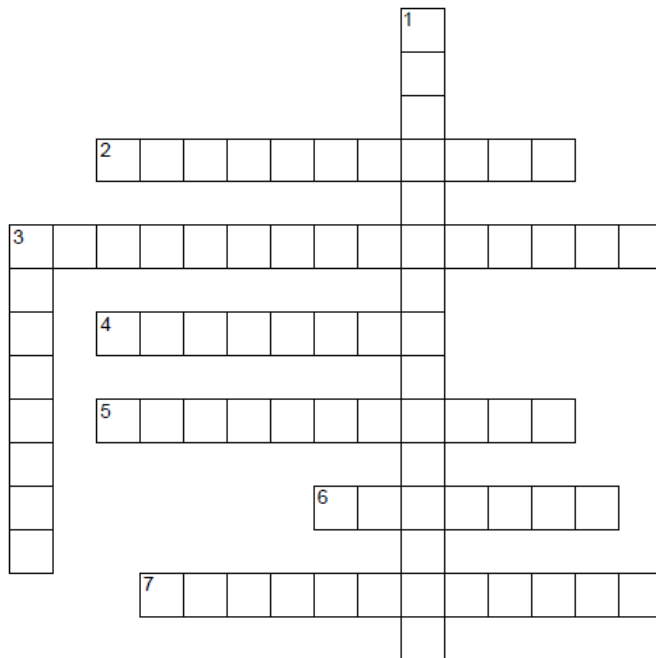
SPOT-FACING is a process that is used to create a flat bearing surface. This is most commonly found on rough castings where a flat surface is needed for a bolt head or washer to sit flat. This enables bolts, washers, and nuts to be seated so that they can be tightened properly. Counter-bore tools can be used to perform this operation.

COUNTER-SINKING

COUNTER-SINKING is a process by which an angled opening is cut into the work piece surface so that a flathead bolt can sit flat or slightly below that surface. This process is also used to create a chamfer at the opening of a hole to allow easier entry of pins, taps, and to deburr the opening for aesthetic reasons. Countersinks have included angles ground to a sharp edge. The angles that are commonly found on counter-sinks are 60°, 82°, 90°, and 100°. The 82° is the most commonly used counter-sink used in machining because most flathead screws and bolts have it. The 100° countersink is used in the aerospace industry.

ACTIVITY 1

Solve the following crossword puzzle using the topics covered in Module 1. For answers that consist of more than one word, a space is needed between each word. Also, forward slashes (/) must be included if necessary.



Across

- 2.) An operation used to recess a hole to create a flat bearing surface.
- 3.) A process that cuts a tapered opening in a hole.
- 4.) A process that uses a cutting tool with two flutes to create a hole.
- 5.) A tool with spiral grooves in its sides with an appearance of being twisted.
- 6.) Increases the size of a hole to a precise size but cannot create a hole.
- 7.) Can be distinguished by the square on the end of its shank.



Down

- 1.) Tools that have a straight shank, driven by a power tool, and create a precisely sized hole.
- 3.) Angles that are cut on the ends of parts that help with assembly.

REAMING

REAMING is a process that is used to finish a hole to precise diameters and produce a smooth finish. This process is performed with tools called reamers. To perform this process a hole must be created with a drill bit first. The drilled hole must be slightly smaller than the size that is to be achieved with a reamer. When determining the need for reaming, it is important to look closely at the tolerances required for that hole size. When tolerances are tight it is most likely a hole that requires a reaming operation.

Remembering that drills tend to produce holes that are slightly larger than their size designation, it is necessary to drill the hole 1/64” or .015” under the hole size. Reamers are designed to cut small amounts of material and will only follow the hole that is produced. Make sure that the previously drilled hole is straight and perpendicular to the top surface.

HAND REAMING

HAND REAMERS are designed to be driven by hand of course. Their square shank is much the same as a square shank on a tap. These are driven into the material with a special tool designed to also drive taps. It is called a tap handle. The **TAP HANDLE** has two handles that provide balance and allow an equal amount of pressure to be applied to the reamer. Hand reamers come with straight or reverse spiral flutes. The cutting action is performed by the flutes that are located on the periphery rather than the tip. The reverse spiral flutes allow for reaming in conditions where the material is interrupted with gaps.

The three basic types of hand reamers commonly found are:

- General-purpose – tapered at the end for easier starting and produce finished holes.
- Expansion – have the ability to be adjusted to remove more material from a hole and are also capable of cutting holes to an exact size.
- Tapered pin – are used to create tapered holes that allow for the insertion of tapered pins.



CHUCKING REAMERS

CHUCKING REAMERS have straight or spiral flutes and are used in most machining situations. They have longer flutes that enable them to ream deep holes. It is important to note that the actual cutting action is performed on the tip of the reamer. If the reamer has a worn tip the resulting hole may be larger than intended.

THREADING

THREADS are what allows parts to be screwed together and are sometimes used to transmit power and motion. Most items that are used everyday have a type of thread on them. It is important to know the difference between external threads and internal threads. External threads are found on the outside diameter of a work piece and internal threads are found on the inside of parts. A bolt would be an example of outside threads and a nut has internal threads. When mated they can tighten to hold parts together. Threads are produced by a few different methods. One way that external threads can be cut is with a threading die. A tap is the choice for producing internal threads.

TAPPING

Tapping is a process performed with a tool called a tap. **TAPS** are the cutting tools that produce the threads with teeth that are ground to a sharp edge on its periphery. Taps are selected based on their size and pitch designation. This can be measured by threads per inch. Drilled holes are first created before a tap can be used (see Figure 4). The size drill that is necessary is based on a tap drill chart. This chart contains information about standard as well as metric taps. If a tap happens to break during the tapping process, the broken portion of the tap should be removed. In some cases the broken tap can cause a scrapped part.

Explore: Video below demonstrates Tap Drilling Technique



Figure 4: Tap Drilling Technique (Video by Eddie Humphries, 2013)

TYPES OF TAPS

There are three main types of taps that are commonly found in industry:

- **HAND TAP** – have straight flutes that curl chips tightly and break them into smaller pieces. The chips are kept in the flutes while the process of tapping occurs. Hand taps come in 2, 3, and 4 flute designations.
- **SPIRAL-POINT TAPS** – have straight flutes but with a specially ground angle on the end. These taps are most commonly used on through holes because they produce stringy chips that are pushed ahead of the tap. If used on holes that have bottoms, breakage can occur.
- **SPIRAL-FLUTE TAPS** – have spiral flutes that resemble the flutes of a drill bit. These flutes are designed to pull the stringy chips out of the hole. These are best used when tapping holes that are blind.

TAP CHAMFER STYLES

Taps are made with **CHAMFERS** ground on their ends. These chamfers enable the tap to easily start into the pre-drilled holes (See figure 5). There are three types of chamfer styles available that come as sets. They are as follows:

- Taper – They are most commonly called starter taps. The beginning threads on these taps are chamfered about 7 to 10 threads from the tip of the tap. This enables the tap to start in a hole that is perpendicular to the surface the pre-drilled hole originated. Although these taps are tapered on their ends, they do not produce tapered threads. This style of chamfer is typically used only for through holes.
- Plug – This style is the most commonly used of the three and is usually used after a taper style tap. A plug tap can be used in through holes or in a blind hole (a hole that has a bottom). This style typically has 3 to 5 threads that are chamfered from the tip of the tap.
- Bottoming – This is used when tapping a blind hole. Only 1 to 2 threads are chamfered and produce full threads almost all the way to the bottom.





Taper Tap

Plug Tap

Bottoming Tap

Figure 5: Types of Chamfers found on common taps (Image by Eddie Humphries, 2013)

ACTIVITY 2

1. A hole has to be tapped $\frac{1}{4}$ -20 UNC 2B, with a thread depth of $\frac{1}{2}$ ". List in order, the tools that will be used to create the hole and the operations that will be performed with each tool. Be sure and include the specific sizes of tools and not just the tools generic name.
2. There are three holes that need to be reamed on a block of steel. The sizes of the reamed holes are .251, .376, and .4375. List in order, the tools that will be used to create the reamed holes and the operations that will be performed with each tool. Be sure and include the specific sizes of tools and not just the tools generic name.



KEY CONCEPTS

- Before a hole can be drilled with a twist drill, the location should be marked using a center drill. The center drill provides a starting hole to ensure that the location of the hole is not compromised. Drills are provided in standard sizes and metric sizes. The standard sizes are broken down into fractional, letter, and number sizes.
- Hole modification is used to modify and enhance pre-drilled holes. The tools that are used are named for the action that is performed such as: counter-boring, counter-sinking, spotfacing, reaming, and tapping. It is important to remember that reaming is a process of enlarging the pre-drilled hole to a precise size. This process does not straighten the hole if drilled improperly. The rule for choosing a drill to produce a hole prior to reaming is to use a drill that is sized 1/64" smaller than the reamer size.
- Tapping is a process that creates threads inside of a hole. Tapping is performed with tools that are called taps. They come in three main styles: hand taps, spiral-pointed, and spiral-fluted. These taps also come in different chamfer styles: taper, plug, and bottoming. It is very important to choose the drill specifically designated for a tap in reference to a tap/drill chart. These tap and drill combinations are based on a 75% thread depth which will produce the right amount of threads in a hole so that screws and bolts used stay engaged in the threads.

KEY TERMS

DRILLING

CENTER DRILL

TWIST DRILL

COUNTER-BORING

SPOT-FACING

COUNTER-SINKING

REAMING

HAND REAMERS

CHUCKING REAMERS

THREADS

TAPS

HAND TAP

SPIRAL-POINT TAPS

SPIRAL-FLUTE TAPS

CHAMFERS



ACTIVITY # 3 COMPARE AND CONTRAST KEY TERMS

	Counter-boring	Spot-facing
Describe each process		
Explain the similarities of each process		
Distinguish how each process is different		

	Spiral-fluted Tap	Spiral- pointed Tap
Describe the purpose of each tap		
Explain how the taps are similar		
Explain each taps differences		

	Drilling	Reaming
Define the purpose of each process		
Explain how the processes are similar		
Explain how each process is different and why they are different		

DISCUSSION PROMPTS

DISCUSSION #1

Why is it important for taps to have different chamfer styles? What is each chamfer style used for? Explain your answer in 3 to 5 complete sentences.



DISCUSSION #2

What is the difference between counter-boring and spotfacing? Can these operations be performed with the same tools? Explain your answer in 3 to 5 complete sentences.

ASSESSMENT

MODULE REINFORCEMENT

True or False: Read the following questions and determine whether the statement is true or false.

1. Reaming is the process of drilling a hole to a precise size.
2. Bottoming taps have 7-10 threads chamfered at the start of the tap.
3. Spiral-fluted taps are designed to pull chips out of the hole being tapped.
4. Hand reamers are distinguished from chucking reamers by their square shank.
5. Expansion reamers have the ability to taper holes that are drilled.
6. A nut is an example of an external thread.
7. Plug taps are used to tap holes to maximum thread depth.
8. Tap sizes are designated on their shanks.
9. External threads can be produced by a thread cutting die.
10. Counter-sinking that produces 100° angles are used in the aerospace industry.

Multiple Choice: Read the following questions or statements and select the best answer.

1. Which is not a tap chamfer style?
 - a. Taper
 - b. Bottom
 - c. Plug
 - d. Spiral
2. _____ is a process that will create a recess in the material for the head of a bolt.
 - a. Counter-boring
 - b. Counter-reaming
 - c. Tapping
 - d. Center-drilling
3. Counter-sink tools are most commonly found with an angle of _____.
 - a. 90 degrees
 - b. 82 degrees



- c. 100 degrees
 - d. 60 degrees
4. Center-drills have both a pilot drill and _____ found on them.
- a. Counter-bore
 - b. Counter-drill
 - c. Counter-sink
 - d. Counter-ream
5. Chucking reamers have spiral flutes and _____.
- a. Curved flutes
 - b. Straight flutes
 - c. Twisted flutes
 - d. Threaded flutes
6. Drill bits are not sized by which of the following.
- a. Fractions
 - b. Letters
 - c. Numbers
 - d. Decimal inches
7. _____ is used to create a flat surface for flat seating of fasteners.
- a. Spotfacing
 - b. Counter-boring
 - c. Back-boring
 - d. Reaming
8. The properties of a plain hole can be altered through a _____ process.
- a. Hole enlargement
 - b. Hole modification
 - c. Hole creation
 - d. Tapping
9. The type of tap that is used to start a tapping process is a _____.
- a. Taper
 - b. Bottoming
 - c. Plug
 - d. Series
10. Counter-bore tools have a pilot that _____ the tool into the pre-drilled hole.
- a. Starts
 - b. Guides
 - c. Follows
 - d. Precedes



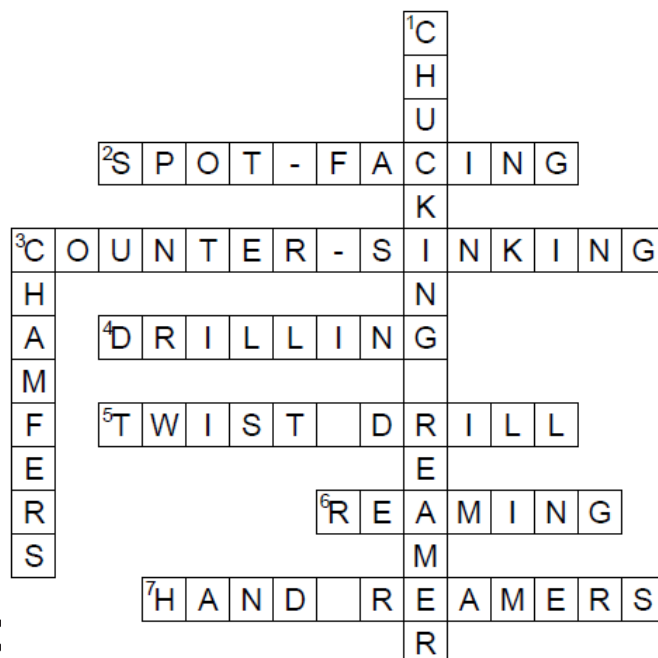
ANSWERS TO ACTIVITIES

Answer Key

True or False	Multiple Choice
1. F	1. D
2. F	2. A
3. T	3. B
4. T	4. C
5. F	5. B
6. F	6. D
7. F	7. A
8. T	8. B
9. T	9. A
10. T	10. B

ACTIVITY # 1

Solve the following crossword puzzle using the topics covered in Module 1. For answers that consist of more than one word, a space is needed between each word. Also, forward slashes (/) must be included if necessary.



Across

2.) An operation used to recess a hole to create a flat bearing surface.

3.) A process that cuts a tapered opening in a hole.

4.) A process that uses a cutting tool with two flutes to create a hole.



- 5.) A tool with spiral grooves in its sides with an appearance of being twisted.
- 6.) Increases the size of a hole to a precise size but cannot create a hole.
- 7.) Can be distinguished by the square on the end of its shank.

Down

- 1.) Tools that have a straight shank, driven by a power tool, and create a precisely sized hole.
- 3.) Angles that are cut on the ends of parts that help with assembly.

ACTIVITY # 2

1. Tools will include: #4 Center Drill, #7 Drill Bit, ¼ - 20 UNC 2B starter tap, and ¼ - 20 UNC 2B bottoming tap. First, center drill the location of the hole to provide a pilot hole for the drill. Second, drill the hole with a #7 drill bit. Third, using the ¼ - 20 UNC 2B starter tap, tap the hole to produce 1 to 3 full threads. Last, using the ¼ - 20 UNC 2B bottoming tap, finish tapping the hole to the desired ½” depth.
2. Tools will include: #4 center drill, 15/64” drill bit, 23/64” drill bit, 27/64” drill bit, .251Ø reamer, .376Ø reamer, and .4375Ø reamer. Center drill all three locations of the holes. Use the 15/64” drill and .251Ø reamer on hole 1. Use the 23/64” drill bit with the .376Ø reamer for hole 2. Use the 27/64” drill bit with the .4375Ø reamer for hole 3. Using those combinations of tools, drill and ream all three holes.

ACTIVITY # 3 RUBRIC SCALE

Compare and Contrast: Counter-boring and Spot-facing	Excellent (50 points max)	Good (45 points max)	Fair (40 points max)	Additional Practice Needed (35 points)
---	--------------------------------------	---------------------------------	---------------------------------	---



1. Described each process correctly.				
	Excellent (45 points max)	Good (40 points max)	Fair (36 points max)	Additional Practice Needed (32 point max)
2. Explained the similarities of each process				
3. Distinguish how each process is different				
	Turned In On Time (5 points)	Turned in 1 day late (4 points)	Turned in 2 days late and gave reason to instructor on tardiness (3 points)	Turned in 3 days late or Did Not Turn In after day 2 tardiness (2 point)
4. Completed the task correctly.				
Total				



Compare and Contrast: Spiral-flute and Spiral-pointed taps	Excellent (50 points max)	Good (45 points max)	Fair (40 points max)	Additional Practice Needed (35 points)
1. Described the purpose of each tap				
	Excellent (45 points max)	Good (40 points max)	Fair (36 points max)	Additional Practice Needed (32 point max)
2. Explained how the taps are similar				
3. Explained each of the taps differences				
	Turned In On Time (5 points)	Turned in 1 day late (4 points)	Turned in 2 days late and gave reason to instructor on tardiness (3 points)	Turned in 3 days late or Did Not Turn In after day 2 tardiness (2 points)
4. Completed the task correctly.				
Total				



Compare and Contrast: Drilling and Reaming	Excellent (50 points max)	Good (45 points max)	Fair (40 points max)	Additional Practice Needed (35 points)
1. Defined the purpose of each process				
	Excellent (45 points max)	Good (40 points max)	Fair (36 points max)	Additional Practice Needed (32 point max)
2. Explained how the processes are similar				
3. Explained how each process is different and why they are different				
	Turned In On Time (5 points)	Turned in 1 day late (4 points)	Turned in 2 days late and gave reason to instructor on tardiness (3 points)	Turned in 3 days late or Did Not Turn In after day 2 tardiness (2 point)
4. Completed the task correctly.				



Total				
--------------	--	--	--	--

ATTRIBUTION TABLE

Author/s	Title	Source	License
Ladell Humphries	Figure 1: Basic Drilling Technique	Author	Creative Commons Attribution 3.0 Unported License
Ladell Humphries	Figure 2: Center Drilling	Author	Creative Commons Attribution 3.0 Unported License
Ladell Humphries	Figure 3: Basic Drilling and Counter-Boring Technique	Author	Creative Commons Attribution 3.0 Unported License
Ladell Humphries	Figure 4: Tap Drilling Technique	Author	Creative Commons Attribution 3.0 Unported License
Ladell Humphries	Figure 5: Types of Chamfers found on common taps	Author	Creative Commons Attribution 3.0 Unported License



This workforce solution was funded by a grant awarded by the U.S. Department of Labor's Employment and Training Administration. The solution was created by the grantee and does not necessarily reflect the official position of the U.S. Department of Labor. The Department of Labor makes no guarantees, warranties, or assurances of any kind, express or implied, with respect to such information, including any information on linked sites, and including, but not limited to accuracy of the information or its completeness, timeliness, usefulness, adequacy, continued availability or ownership.





ASSIST Consortium Curriculum is licensed under a Creative Commons Attribution 3.0 Unported License.

Unless otherwise noted, this work was created by Ladell Humphries, Machine Tool Instructor, humphriesle@cctech.edu. The design layout was created by Lynda Marshall, marshallc@cctech.edu and the template design was created by Clemson University Center for Workforce Development, frady@clemson.edu. ASSIST grant is licensed under the Creative Commons Attribution 3.0 Unported License. To view a copy of this license, visit <http://creativecommons.org/licenses/by/3.0/> or send a letter to Creative Commons, 444 Castro Street, Suite 900, Mountain View, California, 94041, USA.

