Introduction to Selecting Turning Tools

IImportant decisions for the selection of cutting tools for standard turning operations

The variety of shapes and materials machined on modern turning centers makes it imperative for machine operators to understand the decision-making process for selecting suitable cutting tools for each job.

This course curriculum contains 16 hours of material for instructors to get their students ready to make basic decisions about which tools are suitable for standard turning operations.

MachiningCloud
Smart Manufacturing

Table of Contents

Introduction

Turning generates axially symmetric shapes by driving a tool along a profile as the workpiece spins. Factors that must be considered when selecting turning tools include the type of material being machined, the shapes of internal and external profiles, the amount of stock to be removed along internal and external profiles, the desired finish of the part surfaces, and the capabilities of the machine.

Audience

This class is intended for lathe operators and students in a turning operator training program who understand how to set up and run a turning machine. Students should be able to identify common types of turning tools, tool holders, and tool materials. Students should also be able to read and understand blueprints and perform basic shop math.

Purpose

This class teaches the decision-making process of selecting cutting tools for basic turning operations. The student will start with a blueprint and learn the thought process for selecting a tool material based on the part material and selecting tool type, tool shape, and tool holder based on the shape of external and internal profiles. Students finish with a machining process plan for the operations, tools, and cutting data to machine a simple turned part.

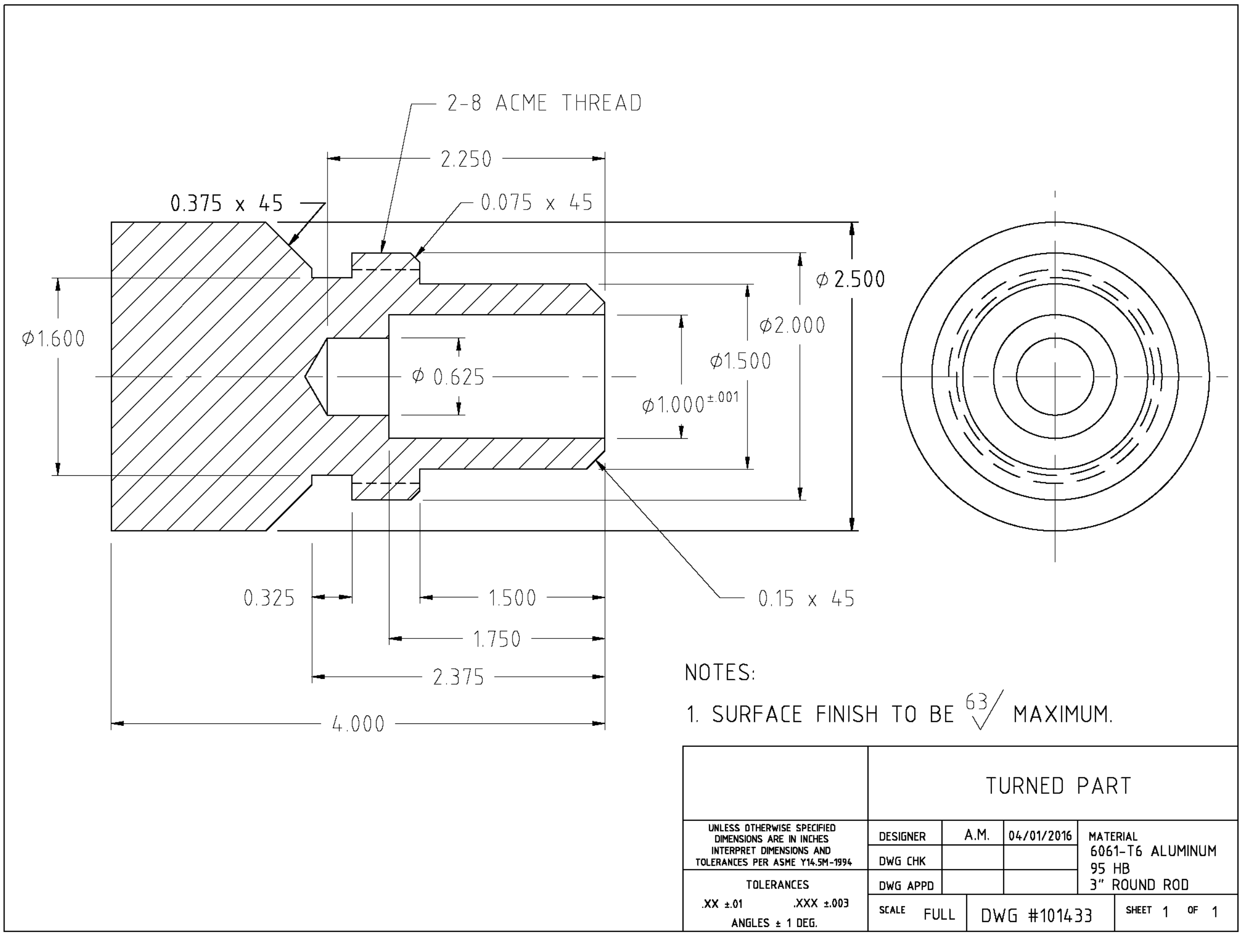
Lesson Objectives

At the end of this class, you will know how to:

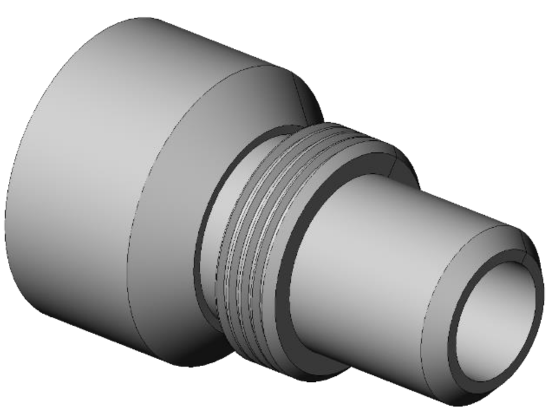
- Use a blueprint to identify areas for material removal on the outer diameter (O.D.), face, and inner diameter (I.D.) of the stock
- Analyze a blueprint to identify the standard types of turning operations that are required to machine the part: facing, turning, boring
- Choose the order in which turning operations are to be applied
- Analyze the geometry of the identified machining profiles to decide which tools and tool holders are suitable to machine the geometry
- Select a tool material based on the part material
- Compare the types of cutting tools to use for turning and profiling on the O.D. and I.D.
- Download and install the MachiningCloud App, an online database of cutting tool data
- Use the MachiningCloud App to search for a holder and its related inserts in an electronic catalog
- Calculate spindle speeds and feed rates

Where to Start: A Blueprint and a Plan

All machining jobs have to start somewhere, and that's usually with a drawing of the finished part. The drawing tells you important information about the size and shape of the part and the stock material.



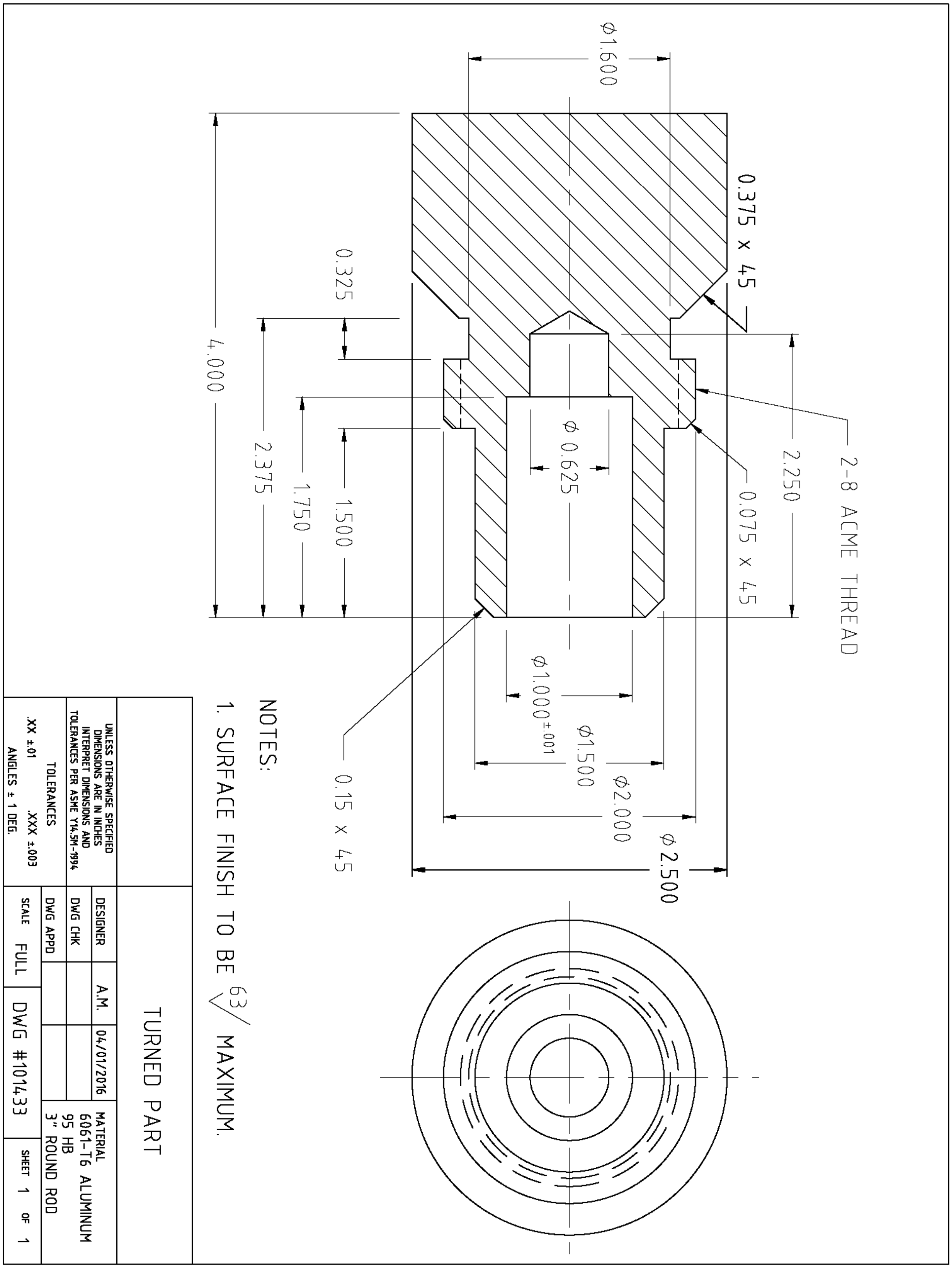
In this class, you will learn how to use a blueprint to make decisions about how to machine the basic turned part shown below.



Most machining jobs are accompanied by a process plan that lists important information for the machine operator such as the type and order of operations, the cutting tool used for each operation, and the cutting specifications for each operation.

PART NO.: 101433		**DATE:**	**MACHINING PROCESS PLAN**			
PART NAME: TURNED PART		**PROGRAMMER:**				
MACHINE: CNC LATHE		**MATERIAL:** 6061-T6 ALUMINUM 95 HB				
SEQ	**OPERATION**	**TOOL**	**STATION**	**SPEED**	**FEED**	**NOTES**
1	FACING (ROUGH)	CNMG-432 INSERT, 30 GRADE DCLN HOLDER, -5°, RH, 1" X 1" X 6" LONG	1	733 rpm	0.012 in/rev	G72
2	O.D. TURNING (ROUGH)	CNMG-432 INSERT, 30 GRADE DCLN HOLDER, -5°, RH, 1" X 1" X 6" LONG	1	733 rpm	0.012 in/rev	G71
3	O.D. TURNING (FINISH)	VNMG-432 INSERT DVJN HOLDER, -3°, RH, 1" X 1" X 6" LONG	2	2100 rpm	0.008 in/rev	G70
4	O.D. GROOVING	GROOVING INSERT, SEAT SIZE 3, CUTTING WIDTH 0.125" MIN TO 0.187" MAX CUTOFF/GROOVING HOLDER, SEAT SIZE 3, .75" X .75" X 5" LONG	3	800 rpm	0.010 in/rev	
5	O.D. THREADING	TRIANGLE LAY DOWN INSERT, 60°, 8 TPI EXTERNAL THREAD HOLDER, RH, .75" X .75" X 4" L	4	250 rpm	0.125 in/rev	
6	DRILLING	5/8" DIAMETER HSS TWIST DRILL NUMBER 35 DOVETAIL CHUCK HOLDER	5	1280 rpm	0.010 in/rev	
7	BORING (ROUGH)	DPGT-211 55° INSERT SDUP HOLDER, STEEL, RH, ∅ 3/8" SHANK X 6" LONG, 0.6" MIN BORE DIAMETER	6	800 rpm	0.012 in/rev	G71
8	BORING (FINISH)	DPGT-211 55° INSERT SDUP HOLDER, CARBIDE, RH, ∅ 3/8" SHANK X 6" LONG, 0.6" MIN BORE DIAMETER	7	1500 rpm	0.008 in/rev	G70

You will use the drawing and the blank machining process plan on the next pages to begin the decision-making process of choosing machining operations and the cutting tools to make the part.


Ø1.600
0.375 x 45
0.325
4.000
2.375
1.750
1.500
Ø 0.625
2.250
2-8 ACME THREAD
0.075 x 45
Ø1.000±.001
Ø1.500
Ø2.000
Ø 2.500
0.15 x 45
NOTES:
1. SURFACE FINISH TO BE 63 MAXIMUM.
TURNED PART
UNLESS OTHERWISE SPECIFIED
DIMENSIONS ARE IN INCHES
INTERPRET DIMENSIONS AND
TOLERANCES PER ASME Y14.5M-1994
TOLERANCES
.XX ±.01 .XXX ±.003
ANGLES ± 1 DEG.
DESIGNER A.M. 04/01/2016
DWG CHK
DWG APPD
SCALE FULL
DWG #101433
MATERIAL
6061-T6 ALUMINUM
95 HB
3" ROUND ROD
SHEET 1 OF 1

MACHINING PROCESS PLAN

PART NO.:	DATE:	
PART NAME:	PROGRAMMER:	
MACHINE:	MATERIAL:	

SEQ	OPERATION	TOOL	STATION	SPEED	FEED	NOTES

Decision 1: What type of machining is needed?

As you look at a blueprint, the first question you need to answer is what type of machining is needed. Machining operations can be generally classified as:

- Turning – Turning is used to produce rotational, typically axially symmetric, parts that have features such as holes, grooves, tapers, and various diameter steps.
- Milling – Milling is typically used to produce parts that are not axially symmetric and have features such as holes, slots, pockets, and contours.
- Hole Making – Hole making is a class of machining operations that are specifically used to cut a cylindrical feature in a workpiece. Hole making can be performed on a variety of machines. Hole making operations typically include drilling, reaming, tapping, and boring.

This blueprint shows a part with cylindrical faces. The blueprint also shows a hole aligned along the center line.

So, to start, you know you need:

1. Turning operations
2. Hole making operations

Decision 2: What is the workpiece material?

The workpiece material and the quantity of material that needs to be removed helps you determine suitable cutting tools.

Is the part made of metal, is it soft or hard, and what kind of metal? Or is it wood? Plastic? Materials are generally selected based on strength, weight and price considerations.

Parameters to be considered:

- Material type
- Chip forming
- Material hardness
- Alloy elements

ISO Material Classifications

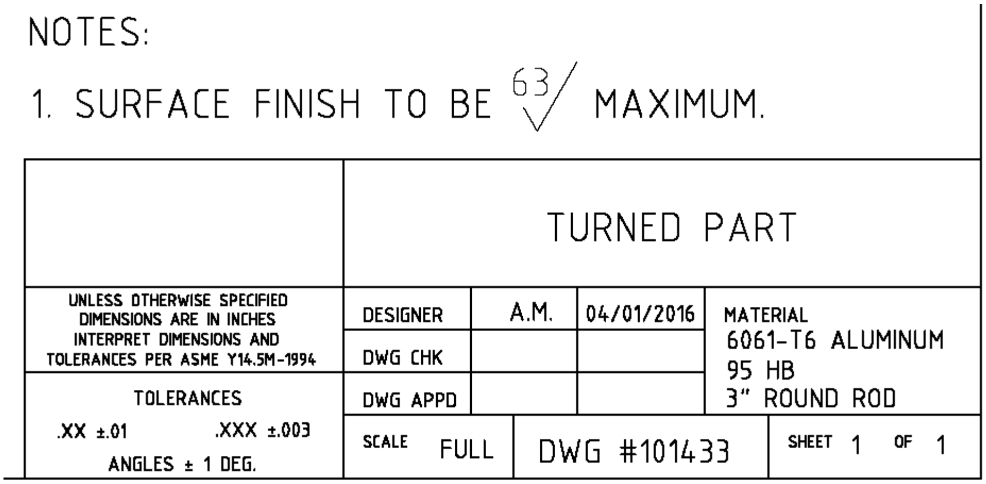
There is such a wide variety of materials used in the metal cutting industry that standards have been established to identify them through a code and color.

ISO Class	Material	Notes
P	Steel	Steel is the most common material group, ranging from unalloyed to high-alloyed material including steel castings. The machinability is normally good, but differs depending on material hardness and content.
M	Stainless Steel	Stainless steels are materials alloyed with a minimum of 12% chromium, other alloys can be nickel and molybdenum. Different conditions make this a large family. They all expose cutting edges to a great deal of heat, notch wear and built-up-edge.
K	Cast Iron	Cast iron is a short-chipping type of material. Grey cast iron (GCI) and malleable cast irons (MCI) are quite easy to machine, while others are more difficult. All cast irons contain silicon carbide (SiC), which is very abrasive to the cutting edge.
N	Non Ferrous	Non-ferrous metals are softer types of metals such as aluminum, copper, brass, etc. Aluminum with a silicon content (Si) of 13% is very abrasive. Generally, high cutting speeds and long tool life can be expected.
S	HRSA and Titanium	Heat-resistant super alloys include a great number of high-alloyed iron, nickel, cobalt and titanium-based materials. They are sticky, create built-up-edge, work harden and generate heat. They are difficult to cut and have a short tool life.
H	Hardened Steel	This group covers steels with a hardness between 45-65 HRc and also chilled cast iron around 400-600 HB. The hardness makes them difficult to machine. The materials generate heat during cutting and are abrasive to cutting edges.

The material is significant because:

- Part material has an influence on the material you choose for your cutting tools
- Part material and tool material together have an influence on the spindle speeds and feed rates you choose for your machining operations

The blueprint in this lesson lists the material as 6061-T6 Aluminum 95 HB.



- 6061: This number designates the alloy. This aluminum alloy contains primarily magnesium and silicon. It is one of the most common aluminum alloys for general-purpose work.
- T6: This number designates the tempering. This metal has an ultimate tensile strength of at least 42,000 psi.
- 95 HB: This number designates the hardness of the metal on the Brinell scale.

The ISO material classification for this part material is:

- N - Non Ferrous

Decision 3: What are the capabilities of my machine?

Most machine shops have a variety of machines for a variety of tasks. Before deciding which machine to use for a job, you must consider the type of work being done and whether the machine you need is available.

Parameters to be considered when choosing a machine:

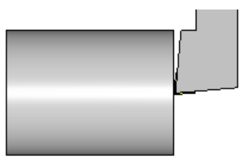
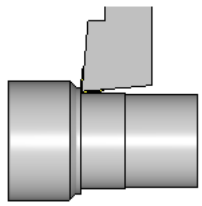
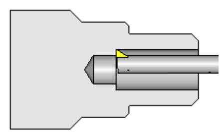
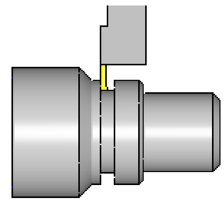
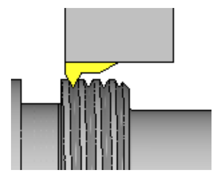
- Condition of the machine
  - Available power
  - Stability
  - Horizontal/vertical machine
  - Spindle type and size
  - Number of axes/configuration
  - Workpiece clamping
- Tool holding
  - Holding strength/rigidity
  - Axial/radial runout
  - Tool overhang

All features of this part can be machined on a single spindle and reached by a tool mounted on a single turret. This part can be machined using a standard turning center.

Decision 4: What machining operations are needed?

A machining process defines a process in which a piece of raw material is cut into a desired final shape and size by a series of machining operations.

Common types of turning operations include the following:

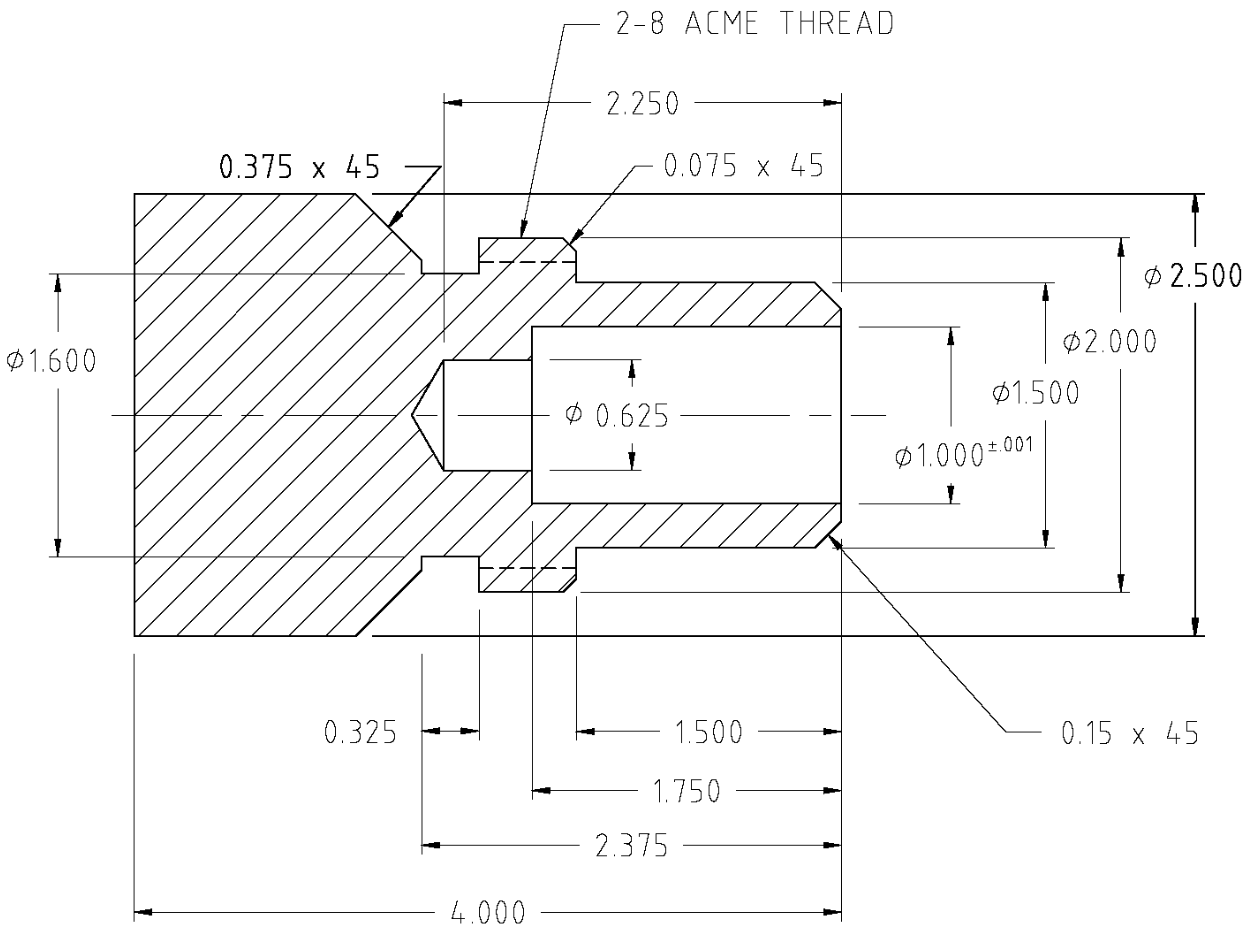
Face Turning	**O.D. Turning**	**I.D. Turning (Boring)**	**Grooving and Profiling**	**Threading**
				
Machining of a face on the part.	Machining of a profile on the outside diameter the part.	Machining of a profile on the inside diameter the part.	Machining between two edges on the part.	Machining of threads on the part.

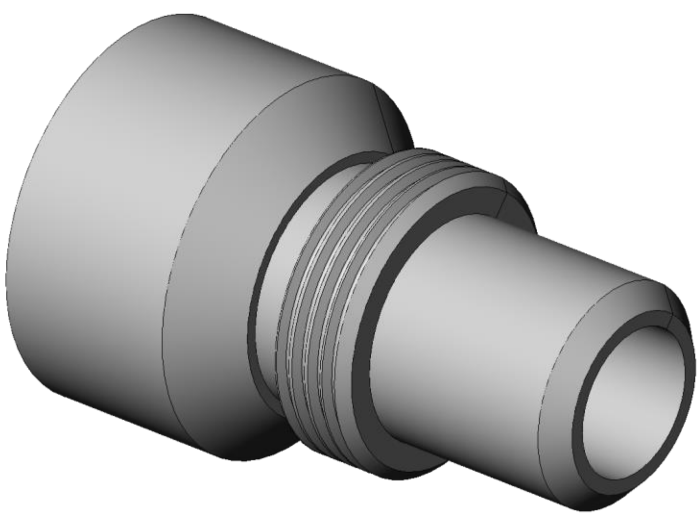
Customer needs must also be considered in the process. What cutting quality to they want? Do they want the finish to be rough or smooth? Are there edge life requirements?

Other parameters to be considered for a turned part:

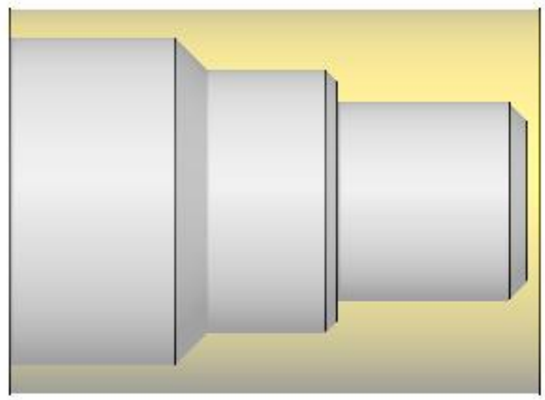
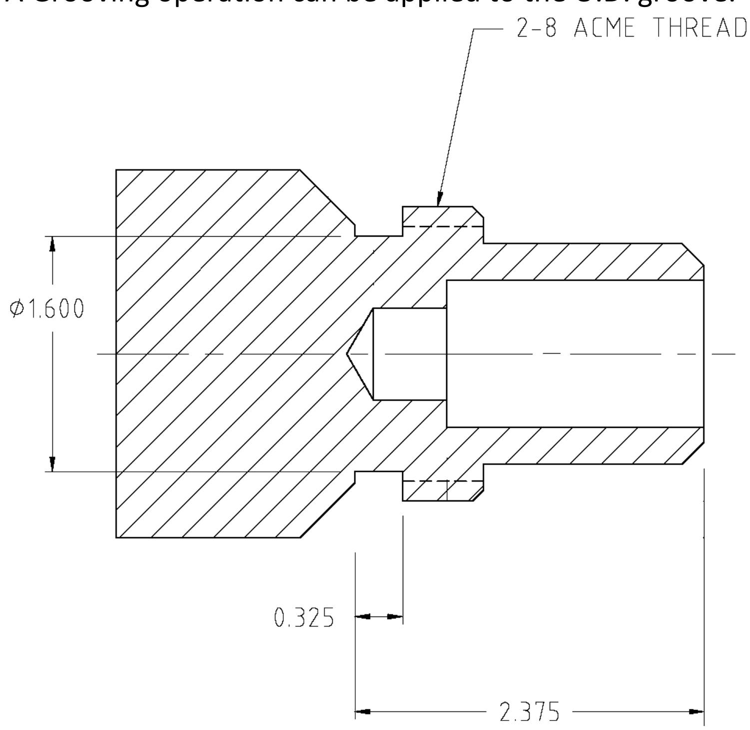
- Geometric shape
  - Length of cylindrical surfaces
  - Increasing/decreasing angles on the profile
  - Deep grooves
  - Deep holes
- Tolerances
  - Dimensional accuracy
  - Surface finish
  - Part distortion
  - Surface integrity

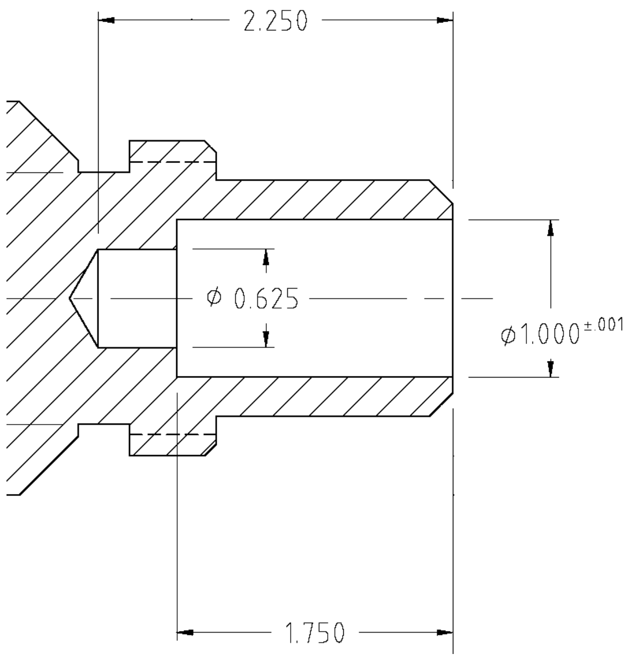
Let's take a closer look at the geometry of this part.





This part has many common turning features: cylindrical faces of varying diameters and lengths, chamfered edges, threads, a groove, and a hole on the center line.

Observations	Machining Requirements
First, take a look at the overall size and shape of the stock and the part. The stock is a 3" round bar. The largest diameter of the part is 2.50". The smallest outer diameter is 1.50".	What do you need to do to make the outside dimensions of the part the proper size? Turning operations can be applied to the face and the O.D. of the stock to produce a rough outline of the final shape.  A Finishing operation along the O.D. profile will produce the desired surface finish.
Next, take a look at the details on the O.D. There is a flange with external threads. The flange is 2" in diameter x .55" long. The flange has external threads with a pitch of 8 threads per inch (TPI). Behind the flange is a groove that is 0.325" wide X 0.200" deep.	A Threading operation can be applied to the 2" diameter. A Grooving operation can be applied to the O.D. groove. 

Now take a look at any features inside the part. There is a hole with two different diameters and depths. The largest hole has a diameter of 1” with a tolerance of plus or minus .001”. The smallest hole has a diameter of 0.625” and has a standard tolerance.	The 1” hole has a precise tolerance and a square shoulder at the bottom. For a hole this size and configuration, it is better to use a Boring operation to machine the diameter to the precise size. A Drilling operation can be applied to the 5/8” hole with a standard twist drill. 

Decision 5: In what order should operations be performed?

There are many ways to program the same workpiece – and all may produce the same finished part. Your goal is to seek the most efficient, timely and accurate way to produce each part.

Operations are typically organized by which ones remove the most material to the least. For example, if the face of a part has a groove, the stock material to the top of the groove is removed first, and then the groove is cut. Otherwise, the operator would waste time grooving out material that will be removed anyway.

Other points to consider when deciding the order of operations:

- Minimize tool changes
- Minimize the travel distance between operations
- Achieve the shortest cycle time possible
- Maintain consistency so machine operators know what to expect

The best way to start is by machining the raw stock to the outer boundaries of the part. In other words, you need to “size” the part.

In this case, the 3” bar stock needs to be machined to a diameter of 2.5” and a length of 4”, then to a diameter of 2” for a length of 2.375” and then chamfered up at a 45° angle, then down to 1.5” for a length of 1.5”.

1. A facing operation is typically performed first. Facing produces a smooth flat surface at the front of the part. You can use a G72 Rough Facing Cycle to generate the operation with only one line (or possibly two lines) of NC code.
2. Next you can rough the profile on the O.D. This operation will remove the largest amount of material in the least amount of time. You can use a G71 Rough Turning Cycle to generate the operation with only one line of NC code.
3. Now you should finish the same O.D. profile with a G70 Finish Turning Cycle to produce a smooth surface finish.

You can start filling out your machining process plan using the information from the blueprint.

PART NO.: 101433		**DATE:**	**MACHINING PROCESS PLAN**			
PART NAME: TURNED PART		**PROGRAMMER:**				
MACHINE: CNC LATHE		**MATERIAL:** 6061-T6 ALUMINUM 95 HB				
SEQ	**OPERATION**	**TOOL**	**STATION**	**SPEED**	**FEED**	**NOTES**
1	FACING (ROUGH)					G72
2	O.D. TURNING (ROUGH)					G71
3	O.D. TURNING (FINISH)					G70

The only other features on the O.D. are the groove and the thread. The roughing and finishing operations removed the material down to the 2” diameter, so now the groove is easier to cut. Cutting the groove first will make it faster and easier to cut the threads.

4. Apply a Grooving operation to remove all the material in back of the threaded area.
5. Apply a Threading operation to cut the external thread.

PART NO.: 101433		DATE:	MACHINING PROCESS PLAN			
PART NAME: TURNED PART		PROGRAMMER:				
MACHINE: CNC LATHE		MATERIAL: 6061-T6 ALUMINUM 95 HB				
SEQ	**OPERATION**	**TOOL**	**STATION**	**SPEED**	**FEED**	**NOTES**
1	FACING (ROUGH)					G72
2	O.D. TURNING (ROUGH)					G71
3	O.D. TURNING (FINISH)					G70
4	O.D. GROOVING					
5	O.D. THREADING					

The last two features to consider are the holes on the inside of the part.

6. Drill the 0.625” hole to a depth of 2.25”. Drilling the hole first provides space for an I.D. turning tool.
7. Rough the profile on the I.D. Because you are removing material on the I.D., this will be a Boring operation. You can again use a G71 Rough Turning Cycle to simplify programming the NC code.
8. Finish bore the 1” diameter to the precise size and surface finish that is specified on the blueprint. Again, use a G70 Finish Turning Cycle.

PART NO.: 101433		DATE:	MACHINING PROCESS PLAN			
PART NAME: TURNED PART		PROGRAMMER:				
MACHINE: CNC LATHE		MATERIAL: 6061-T6 ALUMINUM 95 HB				
SEQ	OPERATION	TOOL	STATION	SPEED	FEED	NOTES
1	FACING (ROUGH)					G72
2	O.D. TURNING (ROUGH)					G71
3	O.D. TURNING (FINISH)					G70
4	O.D. GROOVING					
5	O.D. THREADING					
6	DRILLING					
7	BORING (ROUGH)					G71
8	BORING (FINISH)					G70

Decision 6: What types of tools are needed?

You now know the operations you need to machine this part and the order of those operations. The next step is to decide the type and size of tool to use for each operation.

Parameters to be considered:

- Type of operation
- Part material/tool material
- Depth of cut
- Smallest concave radius

Tool Material

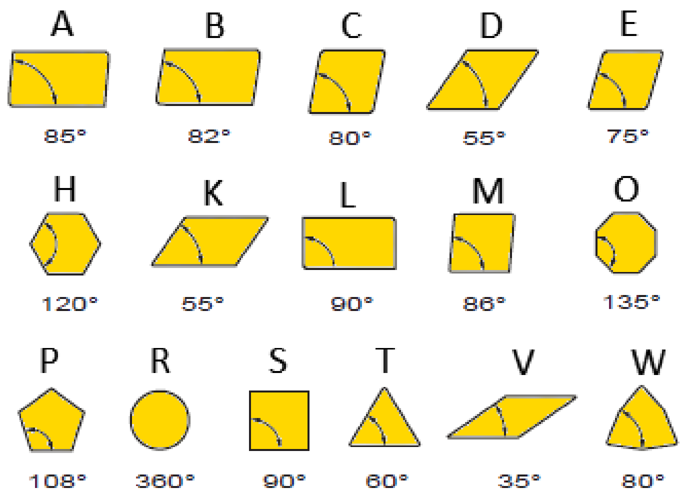
Turning inserts are typically made of carbide, though ceramic, cermet or diamond inserts can be applied to more demanding applications.

This part is made of aluminum, a relatively soft metal. In this case, general performance carbide inserts are suitable.

Turning Inserts

Turning inserts are specified by their ANSI insert designation that provides a shorthand method for identifying the shape and size of an insert.

C	N	M	G	-	4	3	2
Shape	Clearance	Tolerance	Type		I.C. Size	Thickness	Nose Radius



C and W type turning inserts are often used for rough machining due to their larger point angle, which makes them more rigid.

Large point angle:

- Stronger cutting edge
- Higher feed rates
- Increased cutting forces
- Increased vibration

Inserts with a smaller point angle, such as D and V, are often used for finish machining. Although they have less strength, the smaller angle can reach more part details.

Small point angle:

- Weaker cutting edge
- Increased access to part details
- Decreased cutting forces
- Decreased vibration

Tool holders also have ANSI designations.

D	C	L	N	R	16	4	D
Insert Holding	Insert Shape	Style	Insert Relief Angle	Hand	Shank Size	Insert I.C.	Length

The most commonly used insert/holder combination for O.D. rough turning and facing is the C type 80° diamond insert with a 3-5° negative lead angle tool holder. It is often selected because it is the best compromise between strength of insert and end-angle clearance.

The face and the O.D. of this part can be roughed with the same tool. A commonly used and commonly available roughing insert is a CNMG insert (80° Diamond Shape, 0° clearance, ±.002 tolerance, cylindrical opening with double-sided chip breaker). A common size is 1/2" inscribed circle, 3/16” thick, with a nose radius of 1/32. To hold this insert, you can use a common DCLN holder (clamp-style, C insert, 5° side and end cut, 0° clearance angle).

PART NO.: 101433		**DATE:**	**MACHINING PROCESS PLAN**			
PART NAME: TURNED PART		**PROGRAMMER:**				
MACHINE: CNC LATHE		**MATERIAL:** 6061-T6 ALUMINUM 95 HB				
SEQ	**OPERATION**	**TOOL**	**STATION**	**SPEED**	**FEED**	**NOTES**
1	FACING (ROUGH)	CNMG-432 INSERT, 30 GRADE DCLN HOLDER, -5°, RH, 1” X 1” X 6” LONG	1			G72
2	O.D. TURNING (ROUGH)	CNMG-432 INSERT, 30 GRADE DCLN HOLDER, -5°, RH, 1” X 1” X 6” LONG	1			G71
3	O.D. TURNING (FINISH)					G70

To finish the O.D. profile, you should use an insert with a smaller point angle, such as the V shape with a 55° point angle. Be sure to use a holder for a V shaped insert.

PART NO.: 101433		**DATE:**	**MACHINING PROCESS PLAN**			
PART NAME: TURNED PART		**PROGRAMMER:**				
MACHINE: CNC LATHE		**MATERIAL:** 6061-T6 ALUMINUM 95 HB				
SEQ	**OPERATION**	**TOOL**	**STATION**	**SPEED**	**FEED**	**NOTES**
1	FACING (ROUGH)	CNMG-432 INSERT, 30 GRADE DCLN HOLDER, -5°, RH, 1” X 1” X 6” LONG	1			G72
2	O.D. TURNING (ROUGH)	CNMG-432 INSERT, 30 GRADE DCLN HOLDER, -5°, RH, 1” X 1” X 6” LONG	1			G71
3	O.D. TURNING (FINISH)	VNMG-432 INSERT DVJN HOLDER, -3°, RH, 1” X 1” X 6” LONG	2			G70
4	O.D. GROOVING					

Grooving Inserts

Grooving inserts have either a square or round shape on the end. The shape typically corresponds to the shape of the groove being cut, although round inserts are the strongest inserts available and are preferred for roughing in demanding situations.

When the depth of the groove is greater than the width, a multiple grooving operation is preferred. Multiple grooving consists of multiple plunge moves into the groove. The width of the tool is smaller than the width of the groove to allow for multiple side-by-side cutting passes.

When the width of the groove is greater than the depth, a plunge turning operation is a better choice. Plunge turning combines plunging and turning motion, first plunging the tool to a shallow depth of cut, then moving laterally to the opposite end of the groove before plunging again to the next depth.

For the tool holder, the tool overhang should be minimized to improve the stability of the tool during plunge moves.

The width and depth of the groove on this part are about the same size, so the grooving operation can consist of multiple grooving. The width of the insert should be between 1/8" and 3/16" to allow for multiple plunges across the .325" wide groove.

PART NO.: 101433		**DATE:**	**MACHINING PROCESS PLAN**			
PART NAME: TURNED PART		**PROGRAMMER:**				
MACHINE: CNC LATHE		**MATERIAL:** 6061-T6 ALUMINUM 95 HB				
SEQ	**OPERATION**	**TOOL**	**STATION**	**SPEED**	**FEED**	**NOTES**
1	FACING (ROUGH)	CNMG-432 INSERT, 30 GRADE DCLN HOLDER, -5°, RH, 1" X 1" X 6" LONG	1			G72
2	O.D. TURNING (ROUGH)	CNMG-432 INSERT, 30 GRADE DCLN HOLDER, -5°, RH, 1" X 1" X 6" LONG	1			G71
3	O.D. TURNING (FINISH)	VNMG-432 INSERT DVJN HOLDER, -3°, RH, 1" X 1" X 6" LONG	2			G70
4	O.D. GROOVING	GROOVING INSERT, SEAT SIZE 3, CUTTING WIDTH 0.125" MIN TO 0.187" MAX CUTOFF/GROOVING HOLDER, SEAT SIZE 3, .75" X .75" X 5" LONG	3			

Threading Inserts

Threads are cut using a 3-point insert called a laydown triangle threading insert.

There are three types of threading inserts: Full Profile, Partial Profile, and Multi-Tooth.

- Full profile inserts cut the full shape of the thread groove, from bottom to top. These are useful for high productivity in threading.
- Partial profile inserts cut the bottom of the thread groove but leave clearance at the top. These are useful for cutting a range of thread sizes with a small inventory of inserts.
- Multi-Tooth inserts feature multiple teeth in a series to reduce the number of passes required to cut a thread. These are useful for mass production in threading.

The major diameter of the thread has already been cut on the O.D., so you do not need to use a full profile threading insert to cut the top of the thread. For versatility, you will use a partial profile insert with a 60° taper angle that is designed to cut an ACME thread with 8 threads per inch.

PART NO.: 101433		**DATE:**	**MACHINING PROCESS PLAN**			
PART NAME: TURNED PART		**PROGRAMMER:**				
MACHINE: CNC LATHE		**MATERIAL:** 6061-T6 ALUMINUM 95 HB				
SEQ	**OPERATION**	**TOOL**	**STATION**	**SPEED**	**FEED**	**NOTES**
1	FACING (ROUGH)	CNMG-432 INSERT, 30 GRADE DCLN HOLDER, -5°, RH, 1" X 1" X 6" LONG	1			G72
2	O.D. TURNING (ROUGH)	CNMG-432 INSERT, 30 GRADE DCLN HOLDER, -5°, RH, 1" X 1" X 6" LONG	1			G71
3	O.D. TURNING (FINISH)	VNMG-432 INSERT DVJN HOLDER, -3°, RH, 1" X 1" X 6" LONG	2			G70
4	O.D. GROOVING	GROOVING INSERT, SEAT SIZE 3, CUTTING WIDTH 0.125" MIN TO 0.187" MAX CUTOFF/GROOVING HOLDER, SEAT SIZE 3, .75" X .75" X 5" LONG	3			
5	O.D. THREADING	TRIANGLE LAY DOWN INSERT, 60°, 8 TPI EXTERNAL THREAD HOLDER, RH, .75" X .75" X 4" L	4			

Drilling Tools

HSS twist drills are by far the most common tool used for drilling operations. Twist drills are available in many sizes and diameters.

Choose a twist drill the same size as the 5/8" hole. You will also need a tool holder that is designed to hold a drill on a lathe turret.

PART NO.: 101433		**DATE:**	**MACHINING PROCESS PLAN**			
PART NAME: TURNED PART		**PROGRAMMER:**				
MACHINE: CNC LATHE		**MATERIAL:** 6061-T6 ALUMINUM 95 HB				
SEQ	**OPERATION**	**TOOL**	**STATION**	**SPEED**	**FEED**	**NOTES**
1	FACING (ROUGH)	CNMG-432 INSERT, 30 GRADE DCLN HOLDER, -5°, RH, 1" X 1" X 6" LONG	1			G72
2	O.D. TURNING (ROUGH)	CNMG-432 INSERT, 30 GRADE DCLN HOLDER, -5°, RH, 1" X 1" X 6" LONG	1			G71
3	O.D. TURNING (FINISH)	VNMG-432 INSERT DVJN HOLDER, -3°, RH, 1" X 1" X 6" LONG	2			G70
4	O.D. GROOVING	GROOVING INSERT, SEAT SIZE 3, CUTTING WIDTH 0.125" MIN TO 0.187" MAX CUTOFF/GROOVING HOLDER, SEAT SIZE 3, .75" X .75" X 5" LONG	3			
5	O.D. THREADING	TRIANGLE LAY DOWN INSERT, 60°, 8 TPI EXTERNAL THREAD HOLDER, RH, .75" X .75" X 4" L	4			
6	DRILLING	5/8" DIAMETER HSS TWIST DRILL NUMBER 35 DOVETAIL CHUCK HOLDER	5			

Boring Tools

Cylindrical boring bars are used for internal turning. Boring bars consist of a round shaft with one insert pocket. Boring bars are available in steel, solid carbide, and carbide-reinforced steel.

You will need a boring bar that will fit into the 5/8" hole that has already been drilled. When the insert is mounted on the boring bar, it also needs to fit inside the 5/8" hole.

Choose a screw-down 3/8" boring bar with a negative 3° lead angle. For both roughing and finishing, choose a D shape 55° insert. For the roughing operation, a boring bar made of HSS is adequate. For the finishing operation, carbide is more suitable for stability and accuracy.

PART NO.: 101433		**DATE:**	**MACHINING PROCESS PLAN**			
PART NAME: TURNED PART		**PROGRAMMER:**				
MACHINE: CNC LATHE		**MATERIAL:** 6061-T6 ALUMINUM 95 HB				
SEQ	**OPERATION**	**TOOL**	**STATION**	**SPEED**	**FEED**	**NOTES**
1	FACING (ROUGH)	CNMG-432 INSERT, 30 GRADE DCLN HOLDER, -5°, RH, 1" X 1" X 6" LONG	1			G72
2	O.D. TURNING (ROUGH)	CNMG-432 INSERT, 30 GRADE DCLN HOLDER, -5°, RH, 1" X 1" X 6" LONG	1			G71
3	O.D. TURNING (FINISH)	VNMG-432 INSERT DVJN HOLDER, -3°, RH, 1" X 1" X 6" LONG	2			G70
4	O.D. GROOVING	GROOVING INSERT, SEAT SIZE 3, CUTTING WIDTH 0.125" MIN TO 0.187" MAX CUTOFF/GROOVING HOLDER, SEAT SIZE 3, .75" X .75" X 5" LONG	3			
5	O.D. THREADING	TRIANGLE LAY DOWN INSERT, 60°, 8 TPI EXTERNAL THREAD HOLDER, RH, .75" X .75" X 4" L	4			
6	DRILLING	5/8" DIAMETER HSS TWIST DRILL NUMBER 35 DOVETAIL CHUCK HOLDER	5			
7	BORING (ROUGH)	DPGT-211 55° INSERT SDUP HOLDER, STEEL, RH, ∅ 3/8" SHANK X 6" LONG, 0.6" MIN BORE DIAMETER	6			G71
8	BORING (FINISH)	DPGT-211 55° INSERT SDUP HOLDER, CARBIDE, RH, ∅ 3/8" SHANK X 6" LONG, 0.6" MIN BORE DIAMETER	7			G70

Decision 7: Are the required tools available?

Machine shops usually have a range of cutting tools in their tool crib to choose from. When you create a process plan, you must ensure that the tools you specified are actually available either in the tool crib or from your cutting tool vendor.

Cutting tool manufacturers provide catalogs of their cutting tools – both in print and on their website. Searching for a specific tool in a catalog can be time-consuming and confusing.

However, an online application makes it simpler and easier to search for the tools you need from different cutting tool manufacturers.

Searching for tools in an online database

MachiningCloud is an independent provider of CNC cutting tool product data. A single source of access to the most current product data from a variety of suppliers, in digital format, available from your desktop. MachiningCloud provides the most up-to-date information, directly obtained from the manufacturers. The data is formatted very closely to each manufacturer's catalog system so it is familiar to users.

Download the MachiningCloud App

MachiningCloud is a cloud-based software, so it can be downloaded and instantly installed on your desktop computer or tablet. All your information is always available on any device upon logging into your account.

1. From your computer or mobile device, open your Internet browser.
2. Enter the following URL address: https://www.machiningcloud.com
3. On the home page, click **DOWNLOAD APP**.
4. Choose your desired platform from the dropdown list. Choices include Android, iPad, Windows Desktop.
5. Then click **Download MachiningCloud**.
6. On your desktop, look for your software to be downloading. If you are prompted to Open/Run or Save the file, click the Save button.
7. On your mobile device, tap **Install**.

Install the MachiningCloud App

1. On your desktop, click the downloaded program to start the installation.
2. Choose your preferred language and click **OK**.
3. Scroll through and read the End User License Agreement. Accept the terms of the license agreement and click **Next**.
4. Choose the **Standard** installation and click **Next**.
5. Click **Install**. Installation of the app only takes a few seconds.
6. Click **Run App**.
7. You will need to sign up for an account so you can store your tooling data in the cloud. Click **Sign up for an account**.
8. Type your information in the mandatory fields. You must enter your:
    - First Name
    - Last Name
    - Your Email
    - Confirm Email
9. When you are finished entering information, click **Register**. You will be prompted for a password.
10. Enter a unique **Password** and then enter the same password to **Confirm Password**.

Search for cutting tools in the MachiningCloud online databases

The MachiningCloud has partnered with several cutting tool manufacturers. This lesson will show you how to select a tool manufacturer and search for tools in their online database.

You are not restricted to one manufacturer in the MachiningCloud. You will see how to select cutting tools from different tool manufacturers and add them to your personal tool list.

1. Tap or double-click the MachiningCloud icon on your desktop or device to launch the App.
2. On the Home page, tap or click **Select Tool Manufacturer**.

   Several tool manufacturers are available. You will first take a look at the Kennametal database of tools.

3. Choose **Kennametal**.

   You are returned to the Home page automatically and the logo for Kennametal is displayed at the top of the screen.

Search for an O.D. roughing tool

1. On the Home page, click **Search for a tool in the electronic catalog**.
2. Tap or click **O.D. and I.D. Turning**.

3. Tap or click **General Performance Inserts**.
4. Tap or click **Negative Style Inserts**.
5. Tap or click **CNMG-F**.
6. In the list of inserts, select **CNMG432F**.
7. Click **Add To** and then choose **My Tools**.

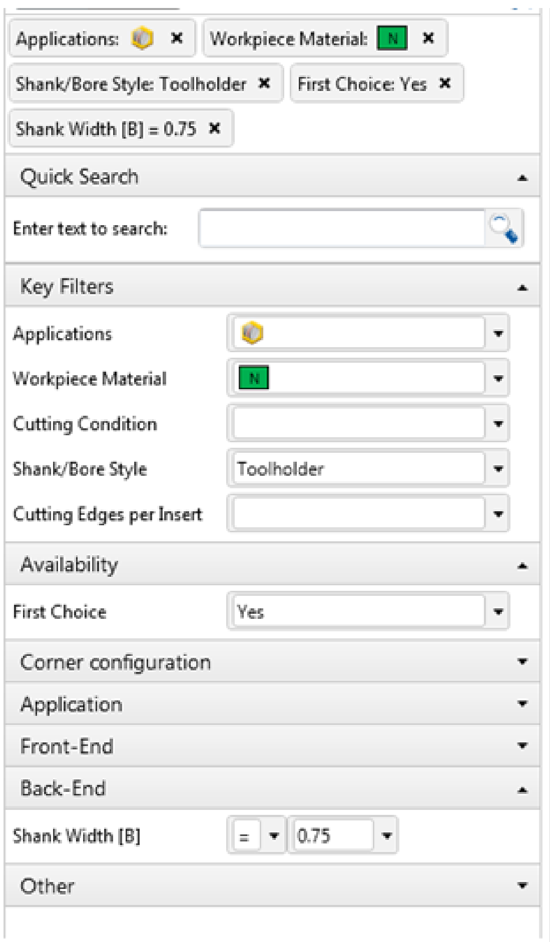
   The insert is added to your personal tool list under My Workshop. Now you can search for a tool holder.

8. With the tool selected, click **Open**.
9. Click **Add Tool Item**.
10. Notice the Filters window at the right side of the screen. You can use filters to narrow your search. You are looking for a DCLN holder, so type **DCLN** in the search box and then click the Search icon.
11. Click **DCLN-KC -5°**.
12. Select the holder with the 1” x 1” x 6” long shank and click **Add to Tool Assembly**.

Search for an O.D. grooving tool

1. Click the **Home** button at the left top corner of the screen.

2. Click **Search for a tool in the electronic catalog**.
3. Click **Turning**.
4. Click **Grooving and Cut-Off**.
5. Click **Grooving**.
6. In the Filters screen, set **Workpiece Material** to **N Non-Ferrous Materials**.
7. Open the **Insert Geometry** filter and set **Insert Seat Size** to **3** and click **Apply Filter**.
8. Open the **Front-End** filter and choose a **Cutting Width** of **0.125**.
9. Set **Insert Shape** to **Groove/Cutoff** and click **Apply Filter**.
10. Open the **Corner Configuration** filter and set **Corner Radius** to **.016**.
11. In the main screen, click the **GMN** insert type.
12. Select an insert with a grade suitable for aluminum.
13. At the bottom of the screen, click **Add To** and add the insert to **My Tools**.
14. In the tools screen, click **Open** and then click **Add Tool Item**.
15. Use the filters to search for a tool holder for grooving applications with a 0.75" shank.



16. Choose a holder and add it to your tool assembly.

Your list of tools is saved in the cloud and you can return to it at any time.

Decision 8: What feeds and speeds should I use?

Surface Feet Per Minute (SFM) is a combination of the cut diameter and RPM. SFM is a constant, with RPM as a variable based upon cut diameter.

Materials will run better at specific SFMs. The recommended SFM for cutting aluminum on a lathe falls in a range from 150 to 400.

When the SFM constant is known for a specific material, the formula below can be used to determine spindle speed for turning various materials.

RPM = SFM x (12/Pi) / Cut Diameter

Revolutions Per Minute (RPM) relates directly to the speed, or velocity, of the spindle. It annotates the number of turns completed in one minute around a fixed axis. RPM maintains the same revolutions per minute throughout the entire operation.

RPM mode (G97) is useful for:

- Center cutting operations (drilling)
- When the diameter at the beginning and end of a cut only differs slightly
- During threading to allow the perfect synchronization between spindle revolution and Z-axis motion to allow precise threads

Most CNC lathes have CSS (constant surface speed) to counteract the natural decrease in surface speed, which speeds up the spindle as the tool moves closer to the turning axis. CSS adjusts the revolutions per minute to maintain a constant surface speed at every distance from the center.

CSS mode (G96) is useful for:

- A uniform surface finish
- When the diameter at the beginning a cut will differ significantly from the diameter at the end of the cut
- Better tool life and machining time because tools will always cut at the appropriate speed

Feed rate can be defined as Inches Per Minute (IPM) or Inches Per Revolution (IPR).

The value for IPR is easily calculated with the following formula:

IPR = IPM / RPM

Approximate cutting speeds for aluminum is 400-700 with a high-speed tool and 800-1000 with a carbide tool.

Approximate feed per revolution for aluminum is .003-.030 in/rev with a high-speed tool and .008-.045 in/rev with a carbide tool.

Note: Speeds and feeds can vary greatly depending on the type of tool, cutting technique, capability of the machine, and desired finish. Operations like threading or parting off require much slower speeds, generally 1/3 to 1/4 of the calculated RPM.

1. If we assume an SFM of 275 and the average work diameter of the roughing operations is 1.5" (3" + 0" ÷ 2), then the RPM for the roughing operations is:

   RPM = 275 x 4 / 1.5 = **733 RPM**

2. Use a moderate feed per revolution of 0.012".
3. Enter the Speed and Feed values for the two O.D. roughing operations in your machining process plan.

PART NO.: 101433		DATE:	MACHINING PROCESS PLAN			
PART NAME: TURNED PART		PROGRAMMER:				
MACHINE: CNC LATHE		MATERIAL: 6061-T6 ALUMINUM 95 HB				
SEQ	**OPERATION**	**TOOL**	**STATION**	**SPEED**	**FEED**	**NOTES**
1	FACING (ROUGH)	CNMG-432 INSERT, 30 GRADE DCLN HOLDER, -5°, RH, 1" X 1" X 6" LONG	1	733 rpm	0.012 in/rev	G72
2	O.D. TURNING (ROUGH)	CNMG-432 INSERT, 30 GRADE DCLN HOLDER, -5°, RH, 1" X 1" X 6" LONG	1	733 rpm	0.012 in/rev	G71

4. For the O.D. finishing operation, you should increase the spindle speed and reduce the feed.

PART NO.: 101433		DATE:	MACHINING PROCESS PLAN			
PART NAME: TURNED PART		PROGRAMMER:				
MACHINE: CNC LATHE		MATERIAL: 6061-T6 ALUMINUM 95 HB				
SEQ	**OPERATION**	**TOOL**	**STATION**	**SPEED**	**FEED**	**NOTES**
1	FACING (ROUGH)	CNMG-432 INSERT, 30 GRADE DCLN HOLDER, -5°, RH, 1" X 1" X 6" LONG	1	733 rpm	0.012 in/rev	G72
2	O.D. TURNING (ROUGH)	CNMG-432 INSERT, 30 GRADE DCLN HOLDER, -5°, RH, 1" X 1" X 6" LONG	1	733 rpm	0.012 in/rev	G71
3	O.D. TURNING (FINISH)	VNMG-432 INSERT DVJN HOLDER, -3°, RH, 1" X 1" X 6" LONG	2	2100 rpm	0.008 in/rev	G70
4	O.D. GROOVING	GROOVING INSERT, SEAT SIZE 3, CUTTING WIDTH 0.125" MIN TO 0.187" MAX CUTOFF/GROOVING HOLDER, SEAT SIZE 3, .75" X .75" X 5" LONG	3			

5. For the O.D. grooving operation, use an SFM of 400. The work diameter is 2”.

PART NO.: 101433		**DATE:**	**MACHINING PROCESS PLAN**			
PART NAME: TURNED PART		**PROGRAMMER:**				
MACHINE: CNC LATHE		**MATERIAL:** 6061-T6 ALUMINUM 95 HB				
SEQ	**OPERATION**	**TOOL**	**STATION**	**SPEED**	**FEED**	**NOTES**
1	FACING (ROUGH)	CNMG-432 INSERT, 30 GRADE DCLN HOLDER, -5°, RH, 1” X 1” X 6” LONG	1	733 rpm	0.012 in/rev	G72
2	O.D. TURNING (ROUGH)	CNMG-432 INSERT, 30 GRADE DCLN HOLDER, -5°, RH, 1” X 1” X 6” LONG	1	733 rpm	0.012 in/rev	G71
3	O.D. TURNING (FINISH)	VNMG-432 INSERT DVJN HOLDER, -3°, RH, 1” X 1” X 6” LONG	2	2100 rpm	0.008 in/rev	G70
4	O.D. GROOVING	GROOVING INSERT, SEAT SIZE 3, CUTTING WIDTH 0.125” MIN TO 0.187” MAX CUTOFF/GROOVING HOLDER, SEAT SIZE 3, .75” X .75” X 5” LONG	3	800 rpm	0.010 in/rev	
5	O.D. THREADING	TRIANGLE LAY DOWN INSERT, 60°, 8 TPI EXTERNAL THREAD HOLDER, RH, .75” X .75” X 4” L	4			

6. Reduce the spindle speed by about 30% for the threading operation. For 8 threads per inch, use a feed rate 1/8” per revolution.

PART NO.: 101433		**DATE:**	**MACHINING PROCESS PLAN**			
PART NAME: TURNED PART		**PROGRAMMER:**				
MACHINE: CNC LATHE		**MATERIAL:** 6061-T6 ALUMINUM 95 HB				
SEQ	**OPERATION**	**TOOL**	**STATION**	**SPEED**	**FEED**	**NOTES**
1	FACING (ROUGH)	CNMG-432 INSERT, 30 GRADE DCLN HOLDER, -5°, RH, 1” X 1” X 6” LONG	1	733 rpm	0.012 in/rev	G72
2	O.D. TURNING (ROUGH)	CNMG-432 INSERT, 30 GRADE DCLN HOLDER, -5°, RH, 1” X 1” X 6” LONG	1	733 rpm	0.012 in/rev	G71
3	O.D. TURNING (FINISH)	VNMG-432 INSERT DVJN HOLDER, -3°, RH, 1” X 1” X 6” LONG	2	2100 rpm	0.008 in/rev	G70
4	O.D. GROOVING	GROOVING INSERT, SEAT SIZE 3, CUTTING WIDTH 0.125” MIN TO 0.187” MAX CUTOFF/GROOVING HOLDER, SEAT SIZE 3, .75” X .75” X 5” LONG	3	800 rpm	0.010 in/rev	
5	O.D. THREADING	TRIANGLE LAY DOWN INSERT, 60°, 8 TPI EXTERNAL THREAD HOLDER, RH, .75” X .75” X 4” L	4	250 rpm	0.125 in/rev	
6	DRILLING	5/8” DIAMETER HSS TWIST DRILL NUMBER 35 DOVETAIL CHUCK HOLDER	5			

7. Assume an SFM of 200 for the drilling operation. 200 x 4 / .625 = 1280 RPM.

PART NO.: 101433		DATE:	MACHINING PROCESS PLAN			
PART NAME: TURNED PART		PROGRAMMER:				
MACHINE: CNC LATHE		MATERIAL: 6061-T6 ALUMINUM 95 HB				
SEQ	OPERATION	TOOL	STATION	SPEED	FEED	NOTES
1	FACING (ROUGH)	CNMG-432 INSERT, 30 GRADE DCLN HOLDER, -5°, RH, 1" X 1" X 6" LONG	1	733 rpm	0.012 in/rev	G72
2	O.D. TURNING (ROUGH)	CNMG-432 INSERT, 30 GRADE DCLN HOLDER, -5°, RH, 1" X 1" X 6" LONG	1	733 rpm	0.012 in/rev	G71
3	O.D. TURNING (FINISH)	VNMG-432 INSERT DVJN HOLDER, -3°, RH, 1" X 1" X 6" LONG	2	2100 rpm	0.008 in/rev	G70
4	O.D. GROOVING	GROOVING INSERT, SEAT SIZE 3, CUTTING WIDTH 0.125" MIN TO 0.187" MAX CUTOFF/GROOVING HOLDER, SEAT SIZE 3, .75" X .75" X 5" LONG	3	800 rpm	0.010 in/rev	
5	O.D. THREADING	TRIANGLE LAY DOWN INSERT, 60°, 8 TPI EXTERNAL THREAD HOLDER, RH, .75" X .75" X 4" L	4	250 rpm	0.125 in/rev	
6	DRILLING	5/8" DIAMETER HSS TWIST DRILL NUMBER 35 DOVETAIL CHUCK HOLDER	5	1280 rpm	0.010 in/rev	
7	BORING (ROUGH)	DPGT-211 55° INSERT SDUP HOLDER, STEEL, RH, ∅ 3/8" SHANK X 6" LONG, 0.6" MIN BORE DIAMETER	6			G71
8	BORING (FINISH)	DPGT-211 55° INSERT SDUP HOLDER, CARBIDE, RH, ∅ 3/8" SHANK X 6" LONG, 0.6" MIN BORE DIAMETER	7			G70

8. Assume an SFM of 160 for the rough boring operation and an average work diameter of .8".

PART NO.: 101433		DATE:	MACHINING PROCESS PLAN			
PART NAME: TURNED PART		PROGRAMMER:				
MACHINE: CNC LATHE		MATERIAL: 6061-T6 ALUMINUM 95 HB				
SEQ	OPERATION	TOOL	STATION	SPEED	FEED	NOTES
1	FACING (ROUGH)	CNMG-432 INSERT, 30 GRADE DCLN HOLDER, -5°, RH, 1" X 1" X 6" LONG	1	733 rpm	0.012 in/rev	G72
2	O.D. TURNING (ROUGH)	CNMG-432 INSERT, 30 GRADE DCLN HOLDER, -5°, RH, 1" X 1" X 6" LONG	1	733 rpm	0.012 in/rev	G71
3	O.D. TURNING (FINISH)	VNMG-432 INSERT DVJN HOLDER, -3°, RH, 1" X 1" X 6" LONG	2	2100 rpm	0.008 in/rev	G70
4	O.D. GROOVING	GROOVING INSERT, SEAT SIZE 3, CUTTING WIDTH 0.125" MIN TO 0.187" MAX CUTOFF/GROOVING HOLDER, SEAT SIZE 3, .75" X .75" X 5" LONG	3	800 rpm	0.010 in/rev	
5	O.D. THREADING	TRIANGLE LAY DOWN INSERT, 60°, 8 TPI EXTERNAL THREAD HOLDER, RH, .75" X .75" X 4" L	4	250 rpm	0.125 in/rev	
6	DRILLING	5/8" DIAMETER HSS TWIST DRILL NUMBER 35 DOVETAIL CHUCK HOLDER	5	1280 rpm	0.010 in/rev	
7	BORING (ROUGH)	DPGT-211 55° INSERT SDUP HOLDER, STEEL, RH, ∅ 3/8" SHANK X 6" LONG, 0.6" MIN BORE DIAMETER	6	800 rpm	0.012 in/rev	G71
8	BORING (FINISH)	DPGT-211 55° INSERT SDUP HOLDER, CARBIDE, RH, ∅ 3/8" SHANK X 6" LONG, 0.6" MIN BORE DIAMETER	7			G70

9. For the finish boring operation, you should increase the spindle speed and decrease the feed rate. Assume an SFM of 375 for the work diameter of 1”.

PART NO.: 101433		**DATE:**	**MACHINING PROCESS PLAN**			
PART NAME: TURNED PART		**PROGRAMMER:**				
MACHINE: CNC LATHE		**MATERIAL:** 6061-T6 ALUMINUM 95 HB				
SEQ	**OPERATION**	**TOOL**	**STATION**	**SPEED**	**FEED**	**NOTES**
1	FACING (ROUGH)	CNMG-432 INSERT, 30 GRADE DCLN HOLDER, -5°, RH, 1” X 1” X 6” LONG	1	733 rpm	0.012 in/rev	G72
2	O.D. TURNING (ROUGH)	CNMG-432 INSERT, 30 GRADE DCLN HOLDER, -5°, RH, 1” X 1” X 6” LONG	1	733 rpm	0.012 in/rev	G71
3	O.D. TURNING (FINISH)	VNMG-432 INSERT DVJN HOLDER, -3°, RH, 1” X 1” X 6” LONG	2	2100 rpm	0.008 in/rev	G70
4	O.D. GROOVING	GROOVING INSERT, SEAT SIZE 3, CUTTING WIDTH 0.125” MIN TO 0.187” MAX CUTOFF/GROOVING HOLDER, SEAT SIZE 3, .75” X .75” X 5” LONG	3	800 rpm	0.010 in/rev	
5	O.D. THREADING	TRIANGLE LAY DOWN INSERT, 60°, 8 TPI EXTERNAL THREAD HOLDER, RH, .75” X .75” X 4” L	4	250 rpm	0.125 in/rev	
6	DRILLING	5/8” DIAMETER HSS TWIST DRILL NUMBER 35 DOVETAIL CHUCK HOLDER	5	1280 rpm	0.010 in/rev	
7	BORING (ROUGH)	DPGT-211 55° INSERT SDUP HOLDER, STEEL, RH, ∅ 3/8” SHANK X 6” LONG, 0.6” MIN BORE DIAMETER	6	800 rpm	0.012 in/rev	G71
8	BORING (FINISH)	DPGT-211 55° INSERT SDUP HOLDER, CARBIDE, RH, ∅ 3/8” SHANK X 6” LONG, 0.6” MIN BORE DIAMETER	7	1500 rpm	0.008 in/rev	G70

You now have a complete process plan with the operations, tools and machining data to cut a turned part.

Important: Make sure that the maximum RPM recommended for the cutter is not exceeded.