Introduction to Selecting Milling Tools

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

The variety of shapes and materials machined on modern milling machines 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 milling operations.

MachiningCloud
Smart Manufacturing

Table of Contents

Introduction

Milling tools remove material by their movement in the machine and from their shape. Factors that must be considered when selecting milling 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 milling operators and students in a milling operator training program who understand how to set up and run a milling machine. Students should be able to identify common types of milling 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 milling 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 milled part.

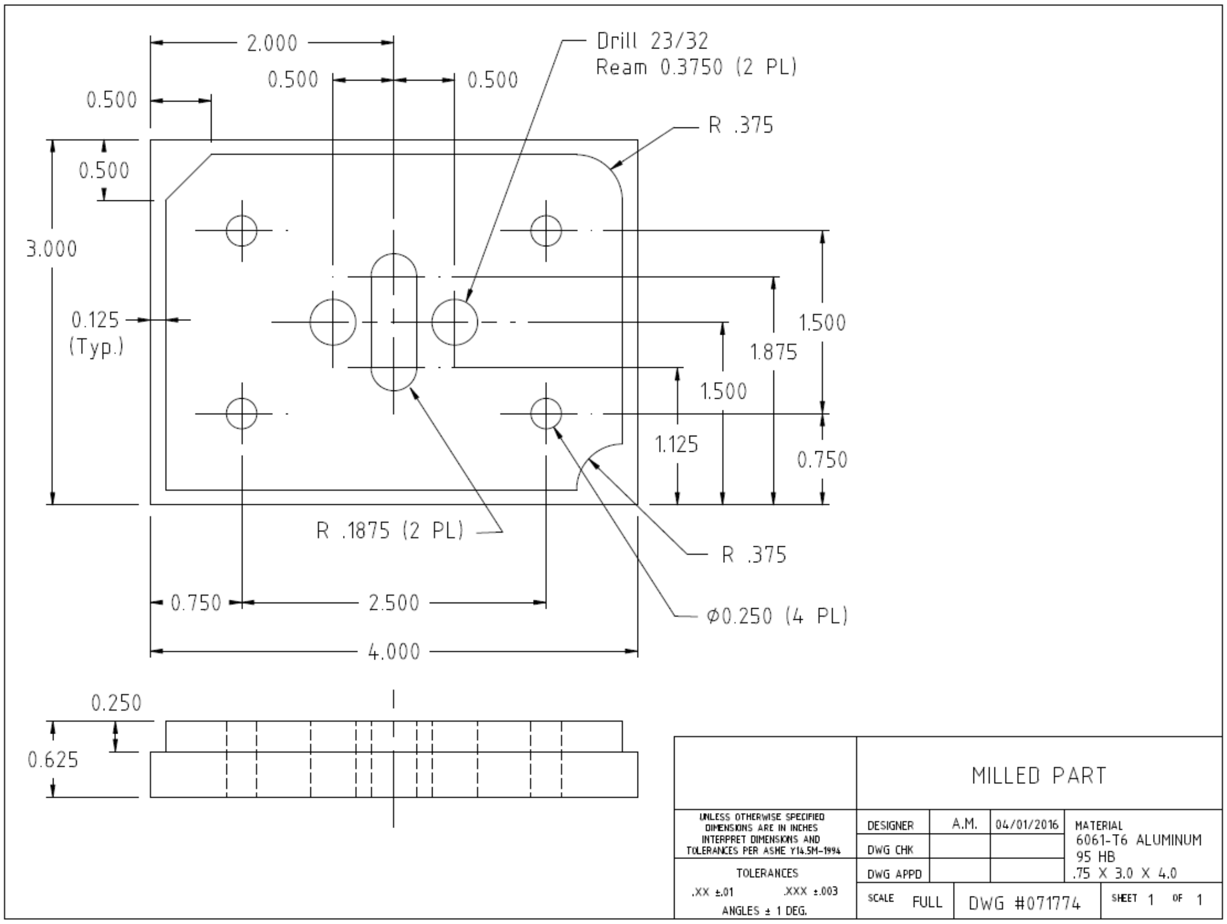
Lesson Objectives

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

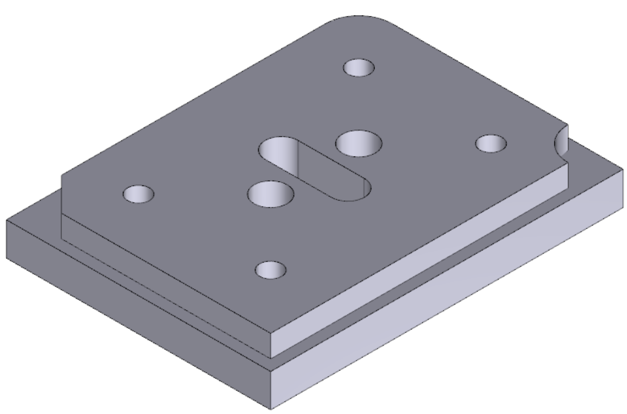
- Use a blueprint to identify areas for material removal on the outside, face, and inside of the stock such as faces, contours, pockets and holes
- Analyze a blueprint to identify the standard types of milling operations that are required to machine the part: facing, contouring, pocketing, drilling
- Choose the order in which milling operations are to be applied
- Analyze the geometry of the identified machining features 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 facing, contouring, pocketing, and drilling
- Download and install the MachiningCloud App, an online database of cutting tool data
- Use the MachiningCloud App to search for a tool and its related items 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.



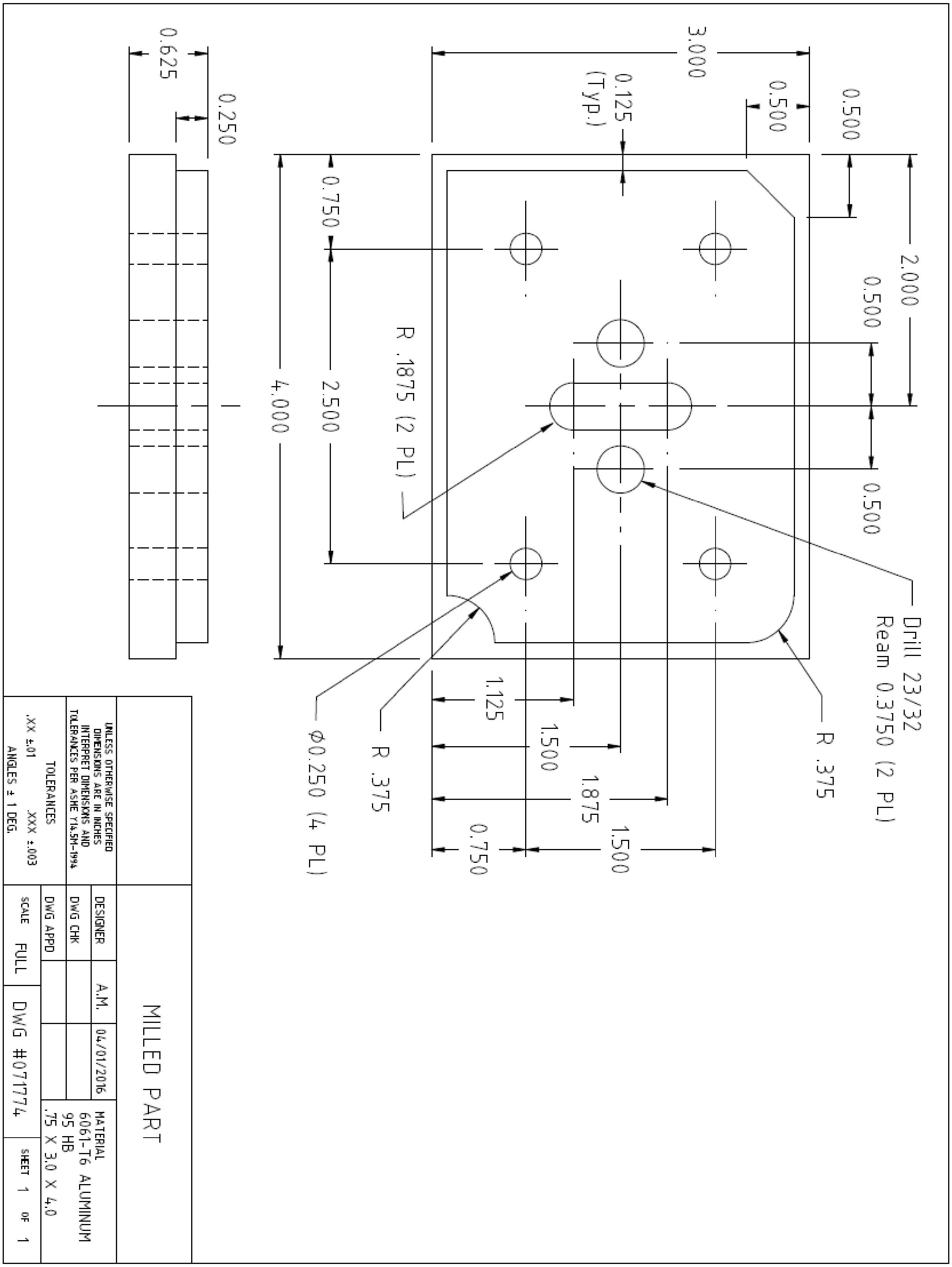
In this class, you will learn how to use a blueprint to make decisions about how to machine the basic milled part shown below.



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.: 071774		DATE: 01/01/2016	MACHINING PROCESS PLAN			
PART NAME: MILLED PART		PROGRAMMER: AJ				
MACHINE: VERTICAL MILL		MATERIAL: 6061-T6 ALUMINUM 95 HB				
SEQ	**OPERATION DESCRIPTION**	**TOOL DESCRIPTION**	STATION	**SPEED**	**FEED**	**NOTES**
1	Facing	Face mill or shell mill, ∅2", 4 inserts	1	894 rpm	21.2 ipm	
2	Mill Outer Contour	∅0.750" HSS Solid End Mill, 4 flutes	2	3312 rpm	53.0 ipm	
3	Center drill all holes	#3 HSS Center Drill	3	4471 rpm	39.3 ipm	
4	Drill (2) 23/64 holes	23/64" HSS Drill, 2 flutes	4	3460 rpm	35.4 ipm	
5	Drill (4) 0.25 holes	∅0.25" HSS Drill, 2 flute	5	4968 rpm	40.3 ipm	
6	Ream (2) 0.375 holes	∅0.375" Reamer, Straight Flute, 6	6	828 rpm	6.5 ipm	
7	Mill Inner Contour	∅0.375" HSS Solid End Mill, 4 flutes	7	5299 rpm	59.3 ipm	
8	Mill Slot	∅0.375" HSS Solid End Mill, 4 flutes	7	5299 rpm	59.3 ipm	

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.


3.000
0.500
0.500
0.125 (Typ.)
0.625
0.250
0.750
2.000
0.500
0.500
R .1875 (2 PL)
4.000
2.500
Drill 23/32
Ream 0.3750 (2 PL)
R .375
R .375
Ø0.250 (4 PL)
1.125
1.500
1.875
0.750
1.500
MILLED 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
MATERIAL
6061-T6 ALUMINUM
95 HB
.75 X 3.0 X 4.0
SCALE FULL
DWG #071774
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 length, width, and depth. It also has flat horizontal faces and vertical walls. The blueprint also shows several holes of different sizes.

So, to start, you know you need:

1. Milling 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

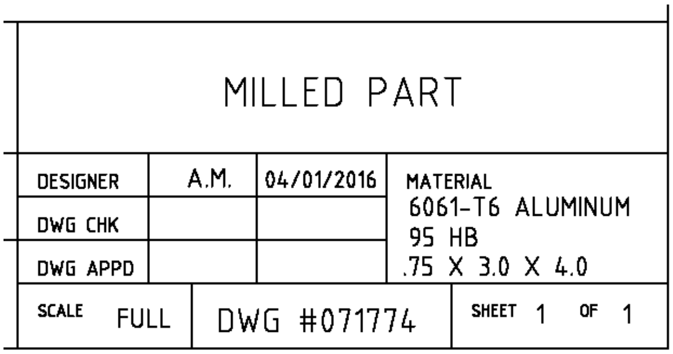
There is such a wide variety of materials used in the metal cutting industry that standards have been established to identify them through a code and color.

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:

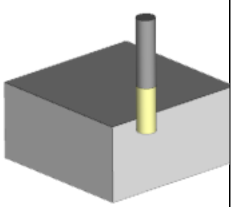
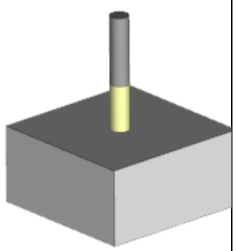
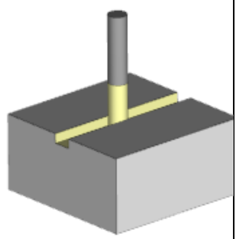
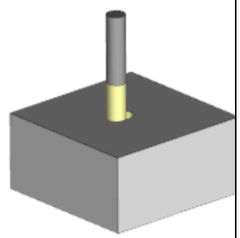
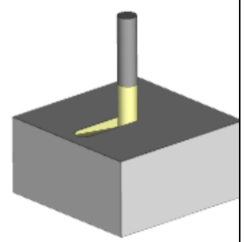
- Condition of the machine
    - Available power
    - Stability
    - Horizontal/vertical machine
    - Spindle type and size
    - Number of axes/configuration
    - Workpiece clamping
- Tool holding
    - Holding strength/rigidity
    - Axial/radial runout
    - Tool overhang

All features of this part can be reached by a tool mounted on a vertical tool spindle. This part can be machined using a standard mill.

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 milling operations include the following:

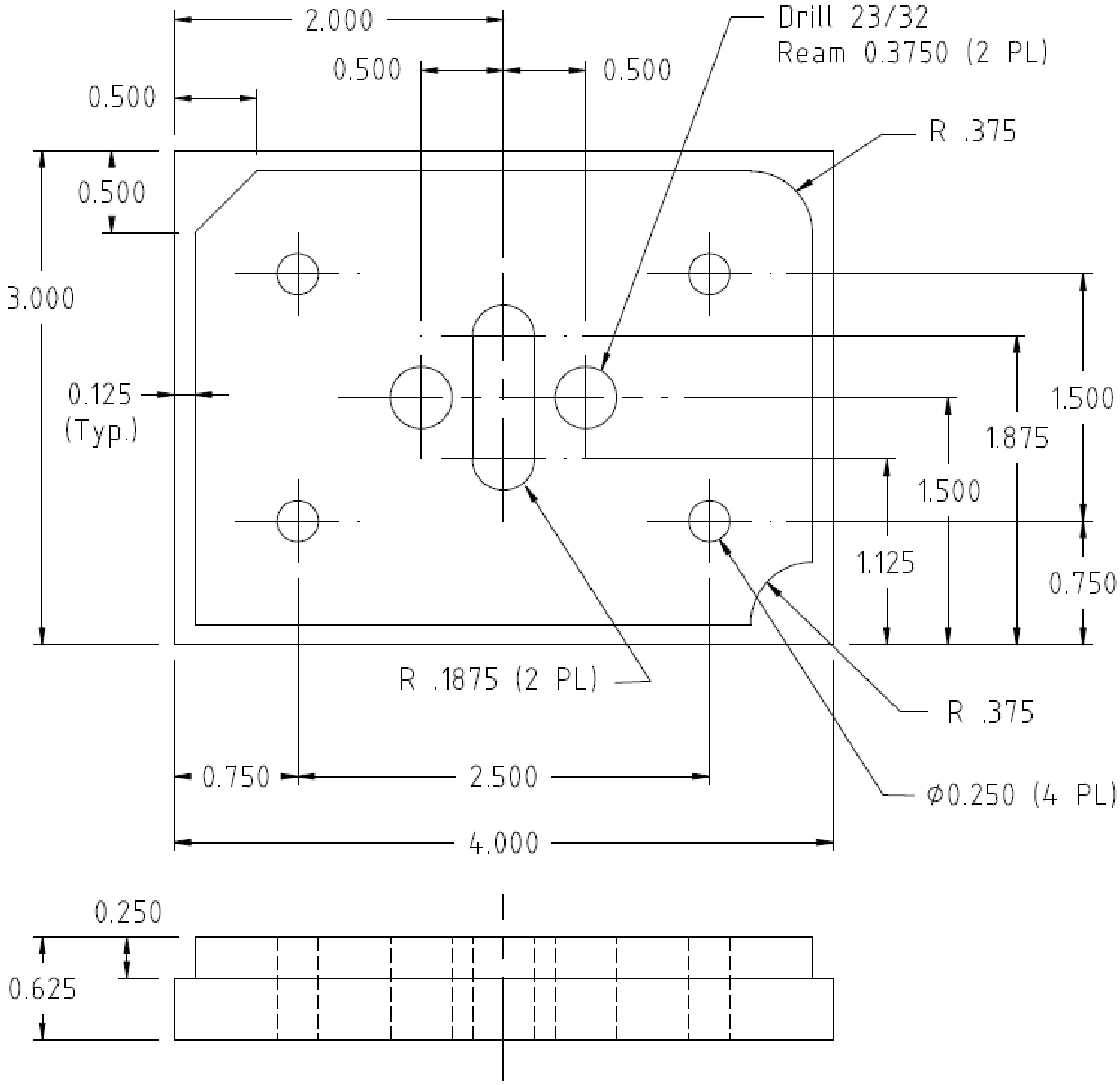
Side Milling	**Facing**	**Slotting**	**Plunging & Pocketing**	**Ramping**
				
Machining of an edge surface on the part.	Machining of a top face on the part.	Machining between two edge surfaces.	Axial feeding into a part along the Z axis. Requires a center cutting end mill.	Axial feeding into a part along the Z axis as well as X or Y. Requires a center cutting end mill.

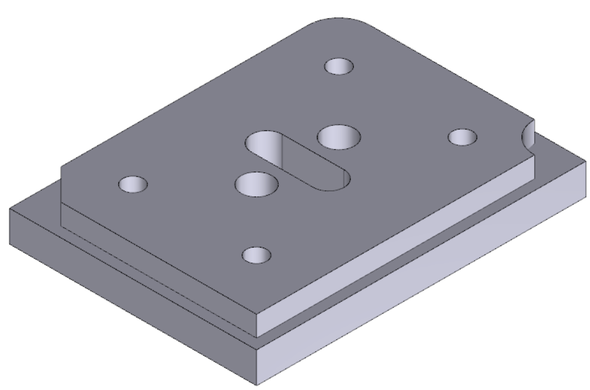
Customer needs must also be considered in the process. What cutting quality to they want? Do they want the finish to be rough or smooth? Are there edge life requirements?

Other parameters to be considered for a milled part:

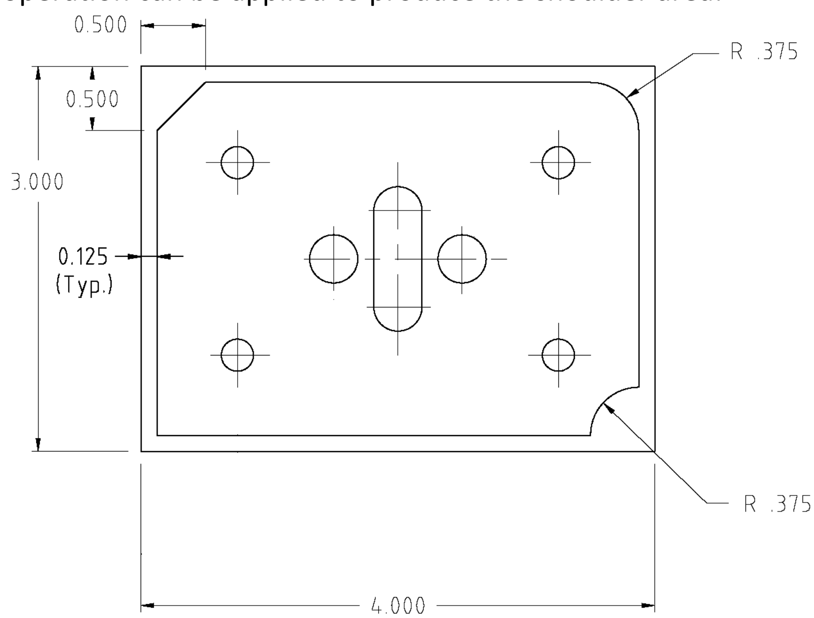
- Geometric shape
    - Planar surfaces
    - Deep cavities
    - Thin walls
    - Slots
- Tolerances
    - Dimensional accuracy
    - Surface finish
    - Part distortion
    - Surface integrity

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

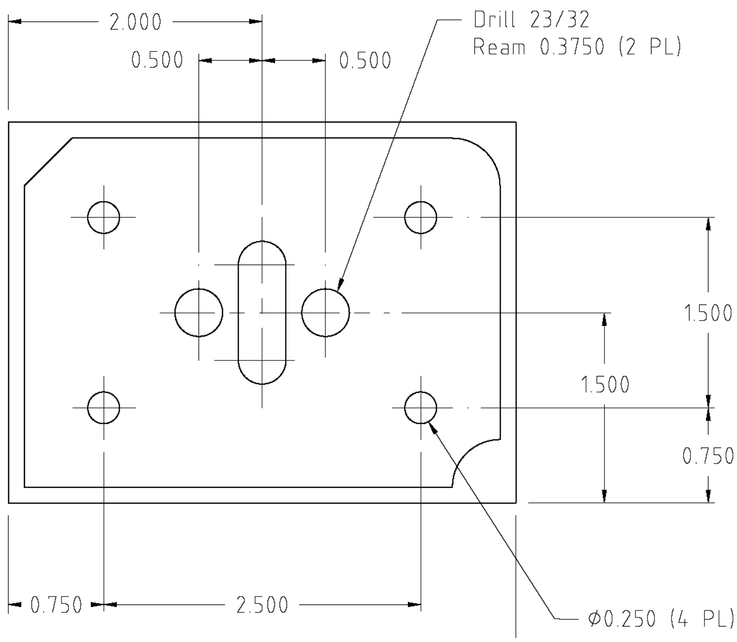




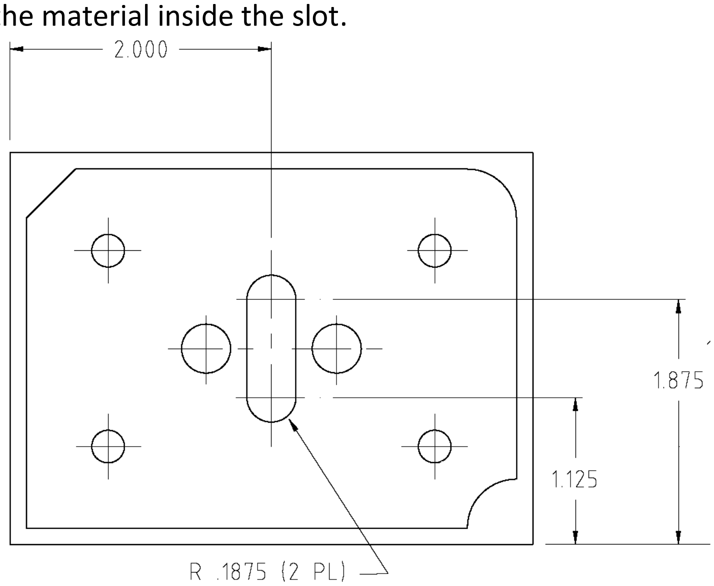
This part has many common milling features: planar faces, sharp and rounded corners, holes, and a rounded slot.

Observations	**Machining Requirements**
First, take a look at the overall size and shape of the stock and the part. The stock is a 0.75” thick plate that is 4.0” long x 3.0” wide. The part is also 4.0” long x 3.0” wide, but the thickness is only 0.625”.	What do you need to do to make the outside dimensions of the part the proper size? A Facing operation can be applied to the top of the stock to reduce the height of the stock by 0.125”. A Side Milling (contouring) operation can be applied to the outside edges of the stock to make sure the part is the proper size.
Next, take a look around the edges of the part. There is a shoulder area inside the part boundary that is a minimum of 0.125” wide. It is 0.250” deep. The walls are vertical. The profile has a sharp corner, a chamfer corner, a rounded corner, and an inverted radius corner.	Since this contour has a single edge, a Side Milling (contouring) operation can be applied to produce the shoulder area. 

<table>
<tr>
<td>Most milled parts have holes that allow the part to be attached to other components.

This part has a total of six holes.
• There are four 1/4" holes that go completely through the part
• There are two precise 3/8” holes that go completely through the part</td>
<td>A single Spot Drilling operation can be applied to all the holes to create a starting point for each hole.

A Drilling operation can be applied to the 1/4" holes with a 0.25” twist drill.

The precise 3/8” holes should be drilled first with a twist drill that is slightly smaller than the finished hole. Then, a Drilling operation can be applied with a reamer that is the exact diameter required.</td>
</tr>
<tr>
<td>Now take a look at any features inside the boundary of the part.

There is a slot that is 3/8” wide with a total length of 1.125” that goes completely through the part.</td>
<td>This slot has no open edges, so a Pocketing operation must be applied that will plunge the tool into the material to remove all the material inside the slot.
</td>
</tr>
</table>

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 a part had a pocket feature with holes at the bottom of the pocket, the material inside the pocket would be milled out before the holes are drilled. Otherwise, the operator would waste time drilling holes in 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
- 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 raw stock needs to be machined to 4.0" long X 3.0" wide X 0.625" thick.

1. A facing operation is typically performed first. Facing produces a smooth flat surface at the top of the part. All other operations can then be performed from a uniformly flat surface.
2. Follow the facing operation with an operation to contour around the outside edges of the part. Now the top of the part is flat and the outside walls are perpendicular.

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

PART NO.: 071774		**DATE:**	**MACHINING PROCESS PLAN**			
PART NAME: MILLED PART		**PROGRAMMER:**				
MACHINE: VERTICAL MILL		**MATERIAL:** 6061-T6 ALUMINUM 95 HB				
SEQ	**OPERATION DESCRIPTION**	**TOOL DESCRIPTION**	**STATION**	**SPEED**	**FEED**	**NOTES**
1	Facing					
2	Mill Outer Contour					

Your next choice is whether to drill the holes or mill the profiles. Either choice will have the same result because the milling and drilling operations do not depend on each other.

For this lesson, you will drill the holes next.

3. When a part has several holes, the location of each hole is usually pre-drilled to a very shallow depth to create a good starting point for the later drilling operations. This is

called Center Drilling because the center of each hole is the only concern for this operation. One tool with a small diameter is used to pre-drill all the holes.

4. The next step is to drill the holes with the largest diameter. In this case, those are the 2 holes with the 3/8” diameter. However, since they require reaming, you will use a drill that is slightly smaller than 3/8”.
5. Drill holes with the next smaller diameter. In this case, the 1/4" holes.
6. Ream holes that require a precise diameter. In this case, those are the holes that were drilled in step 4.

PART NO.: 071774		**DATE:**	**MACHINING PROCESS PLAN**			
PART NAME: MILLED PART		**PROGRAMMER:**				
MACHINE: VERTICAL MILL		**MATERIAL:** 6061-T6 ALUMINUM 95 HB				
SEQ	**OPERATION DESCRIPTION**	**TOOL DESCRIPTION**	**STATION**	**SPEED**	**FEED**	**NOTES**
1	Facing					
2	Mill Outer Contour					
3	Center drill all holes					
4	Drill (2) 23/64 holes					
5	Drill (4) 0.25 holes					
6	Ream (2) 0.375 holes					

The last two features to consider are the contour for the shoulder around the outside of the part and the slot in the middle of the part.

Points to be considered for pockets and slots

The slot could be milled a few different ways. When you are milling a pocket or slot, you must consider how chips will be evacuated out of the pocket because there is no other direction to go but up. In a large pocket or hardened metal, this causes a problem with the cutter getting clogged or re-cutting the same chips.

One strategy for this type of slot is to drill each end with a 3/8” twist drill and then mill only the web of material between the two holes. The pre-drilled holes provide an easy exit for the chips.

Another strategy for pocket milling is to use two cutters and two operations, one to rough the pocket and the second to finish the pocket. The roughing operation would remove most of the material except for a small amount of stock on the walls and/or floors. The finishing operation mills the pocket to the final size.

Finally, if a pocket has an open edge along one of the sides, the tool should always approach the part from that edge instead of plunging into the pocket. When milling starts at an open edge, the chips do not get trapped inside the pocket.

To keep things simple for this lesson, you will use a single milling operation for the slot because the depth is only 0.625” and the material is relatively soft.

7. Create a milling operation to contour around the inner profile to a depth of 0.250”.
8. Create a milling operation to pocket all the material inside the slot profile.

PART NO.: 071774		DATE:	MACHINING PROCESS PLAN			
PART NAME: MILLED PART		PROGRAMMER:				
MACHINE: VERTICAL MILL		MATERIAL: 6061-T6 ALUMINUM 95 HB				
SEQ	OPERATION DESCRIPTION	TOOL DESCRIPTION	STATION	SPEED	FEED	NOTES
1	Facing					
2	Mill Outer Contour					
3	Center drill all holes					
4	Drill (2) 23/64 holes					
5	Drill (4) 0.25 holes					
6	Ream (2) 0.375 holes					
7	Mill Inner Contour					
8	Mill Slot					

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
- Solid cutter/indexable
- Number of flutes/teeth
- Depth of cut
- Maximum diameter
- 45 degree face mill or 90 degree
- Center cutting/non-center cutting

Tool Material

Milling tools are available as solid High Speed Steel (HSS), solid carbide, and solid bodies with indexable inserts. HSS provides good wear resistance and costs less than carbide. Carbide is considerably harder, more rigid, and more wear resistant than HSS.

This part is made of aluminum, a relatively soft metal. In this case, High Speed Steel (HSS) will work well and cost less than carbide tools.

Facing Tool

Several types of tools could be used for a facing operation: a solid HSS end mill, a solid carbide end mill, an indexable end mill, or a face mill.

Face mills are designed specifically to create a flat surface. They remove material with the bottom of the cutter much faster than an end mill. Face mills are large-diameter tools, usually starting at a diameter of 1.5 or 2”. Face mills also feature an array of inserts that can be easily replaced as the edges get worn. Face mills with a 45° lead angle are used most frequently, although 90° shoulder mills are also available for compact spaces.

It is neither economical nor practical to use solid end mills for facing. First, using a solid end mill to cut primarily with the bottom of the tool and not the edges not only underutilizes the cutting edges, but when the cutting edges get worn the entire tool must be replaced. Second, a solid end mill cuts faces slower than a face mill. In a production machining environment, time is money.

An indexable end mill features replaceable inserts like a face mill, but again it is not designed specifically for facing. When considering tooling, you must take into account the intended use of the cutter and the efficiency of cutting time.

The face of this part is only 4” x 3”, so a face mill with a 2” diameter is more than adequate. Face mills of this size typically have 4 inserts.

PART NO.: 071774		**DATE:**	**MACHINING PROCESS PLAN**			
PART NAME: MILLED PART		**PROGRAMMER:**				
MACHINE: VERTICAL MILL		**MATERIAL:** 6061-T6 ALUMINUM 95 HB				
SEQ	**OPERATION DESCRIPTION**	**TOOL DESCRIPTION**	**STATION**	**SPEED**	**FEED**	**NOTES**
1	Facing	Face mill or shell mill, ∅2”, 4 inserts	1			
2	Mill Outer Contour					
3	Center drill all holes					
4	Drill (2) 23/64 holes					
5	Drill (4) 0.25 holes					
6	Ream (2) 0.375 holes					
7	Mill Inner Contour					
8	Mill Slot					

Contouring Tool

End mills are used for contouring along walls. For aluminum, a 2-flute or 4-flute end mill could be used.

For simplicity in this lesson, the outer contour of the part is roughed and finished with one operation and one tool. For versatility, a 3/4" 4-flute solid end mill is a good choice.

PART NO.: 071774		**DATE:**	**MACHINING PROCESS PLAN**			
PART NAME: MILLED PART		**PROGRAMMER:**				
MACHINE: VERTICAL MILL		**MATERIAL:** 6061-T6 ALUMINUM 95 HB				
SEQ	**OPERATION DESCRIPTION**	**TOOL DESCRIPTION**	**STATION**	**SPEED**	**FEED**	**NOTES**
1	Facing	Face mill or shell mill, ∅2”, 4 inserts	1			
2	Mill Outer Contour	∅0.750” HSS Solid End Mill, 4 flutes	2			
3	Center drill all holes					
4	Drill (2) 23/64 holes					
5	Drill (4) 0.25 holes					
6	Ream (2) 0.375 holes					
7	Mill Inner Contour					
8	Mill Slot					

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.

- For the 1/4" holes, choose a twist drill the same size
- For the 3/8” precision hole, choose a drill that is slightly smaller. In this case, 23/64” (3/8 = 24/64).

The size of the center drill must be smaller than the smallest hole. The size of a center drill is designated by a number.

Center Drill Dimensions

Size	Body Dia.	Tip Dia.	Tip Length
00	1/8	.025	.030
0	1/8	1/32	.038
1	1/8	3/64	3/64
2	3/16	5/64	5/64
3	1/4	7/64	7/64
4	5/16	1/8	1/8
5	7/16	3/16	3/16
6	1/2	7/32	7/32
7	5/8	1/4	1/4
8	3/4	5/16	5/16

Reamers are available in precise fractional sizes, with straight flutes or helical flutes. Straight flute reamers are designed for general purpose reaming in most ferrous and non-ferrous materials.

PART NO.: 071774		DATE:	MACHINING PROCESS PLAN			
PART NAME: MILLED PART		PROGRAMMER:				
MACHINE: VERTICAL MILL		MATERIAL: 6061-T6 ALUMINUM 95 HB				
SEQ	OPERATION DESCRIPTION	TOOL DESCRIPTION	STATION	SPEED	FEED	NOTES
1	Facing	Face mill or shell mill, ∅2", 4 inserts	1			
2	Mill Outer Contour	∅0.750" HSS Solid End Mill, 4 flutes	2			
3	Center drill all holes	#3 HSS Center Drill	3			
4	Drill (2) 23/64 holes	23/64" HSS Drill, 2 flutes	4			
5	Drill (4) 0.25 holes	∅0.25" HSS Drill, 2 flute	5			
6	Ream (2) 0.375 holes	∅0.375" Reamer, Straight Flute, 6	6			
7	Mill Inner Contour					
8	Mill Slot					

Pocketing Tool

To cut an enclosed slot or pocket, the tool cannot enter from the side and has to plunge down into the material. For plunge cutting, a center cutting end mill is required.

The radius at both ends of the slot is 0.1875", so you cannot use a tool with a diameter greater than 0.375".

For simplicity and efficiency in this lesson, you will use a 3/8" end mill to cut both the inner profile and the slot.

PART NO.: 071774		DATE:	MACHINING PROCESS PLAN			
PART NAME: MILLED PART		PROGRAMMER:				
MACHINE: VERTICAL MILL		MATERIAL: 6061-T6 ALUMINUM 95 HB				
SEQ	OPERATION DESCRIPTION	TOOL DESCRIPTION	STATION	SPEED	FEED	NOTES
1	Facing	Face mill or shell mill, ∅2", 4 inserts	1			
2	Mill Outer Contour	∅0.750" HSS Solid End Mill, 4 flutes	2			
3	Center drill all holes	#3 HSS Center Drill	3			
4	Drill (2) 23/64 holes	23/64" HSS Drill, 2 flutes	4			
5	Drill (4) 0.25 holes	∅0.25" HSS Drill, 2 flute	5			
6	Ream (2) 0.375 holes	∅0.375" Reamer, Straight Flute, 6	6			
7	Mill Inner Contour	∅0.375" HSS Solid End Mill, 4 flutes	7			
8	Mill Slot	∅0.375" HSS Solid End Mill, 4 flutes	7			

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 on your computer

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 a free 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.

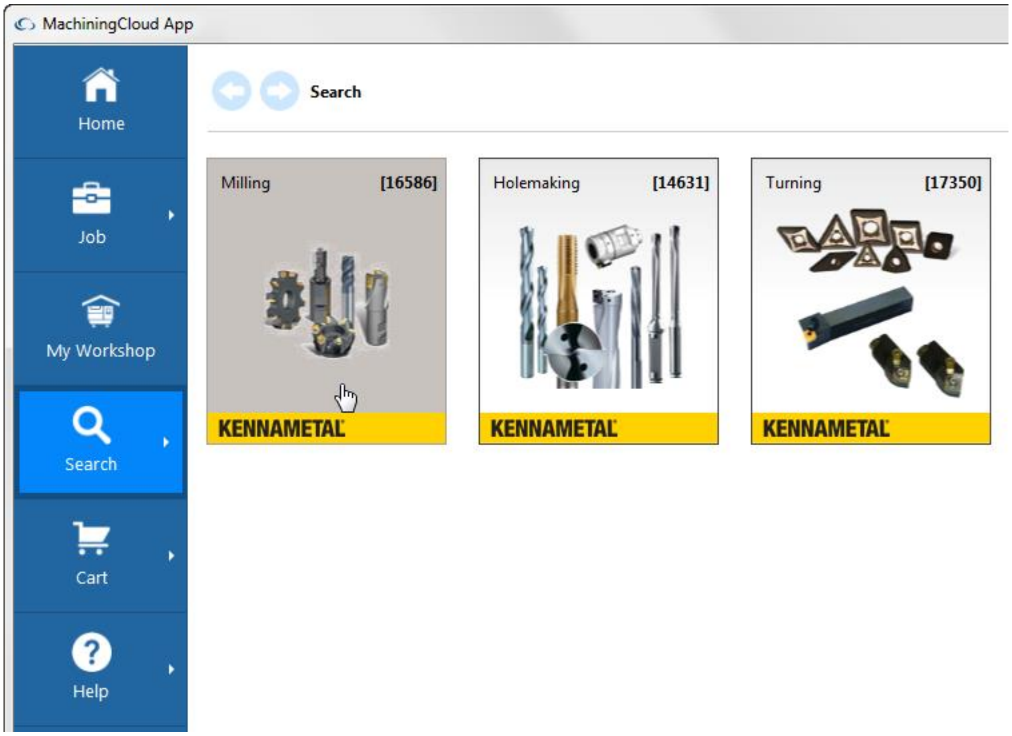
3. Choose **Kennametal**.

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

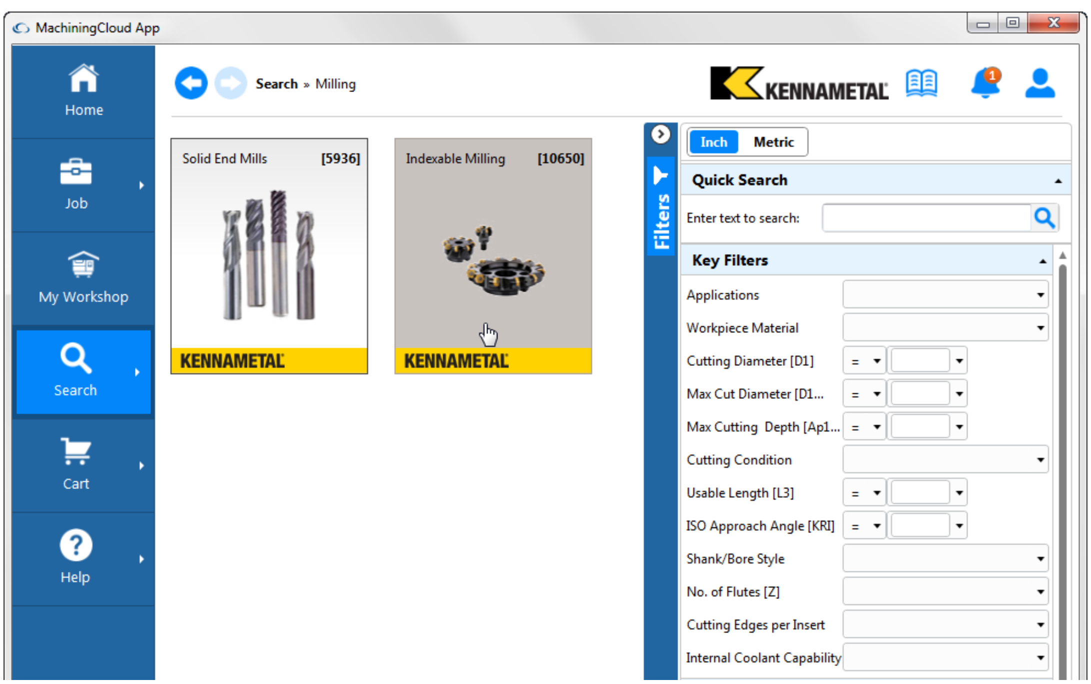
Search for a Face Mill

1. On the **Home** page, click **Search for a tool in the electronic catalog**.

2. You will see choices for different types of machining. Click or tap **Milling**.

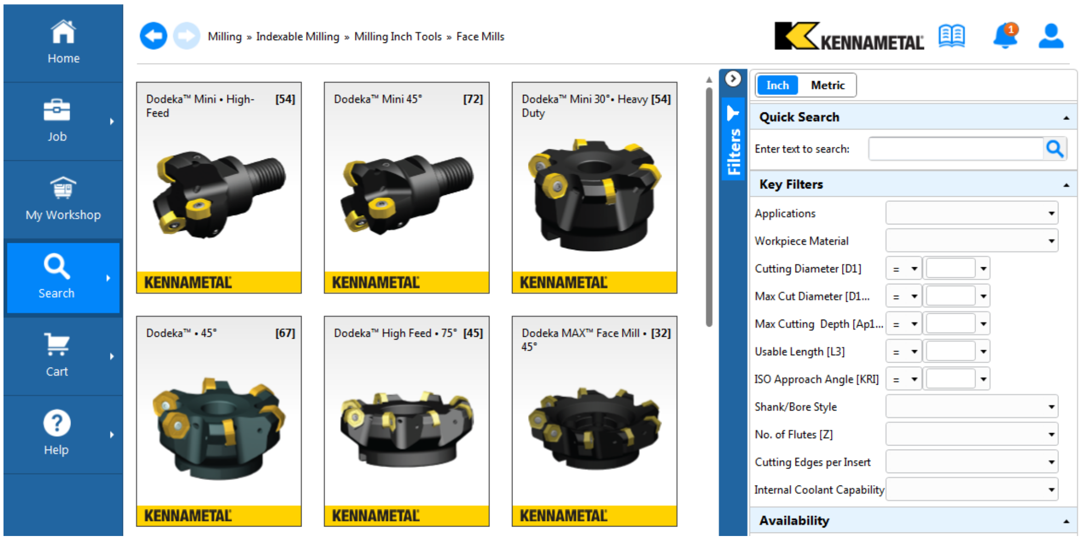


3. Now your choices are narrowed to solid or indexable end mills. Tap or click **Indexable Milling**.



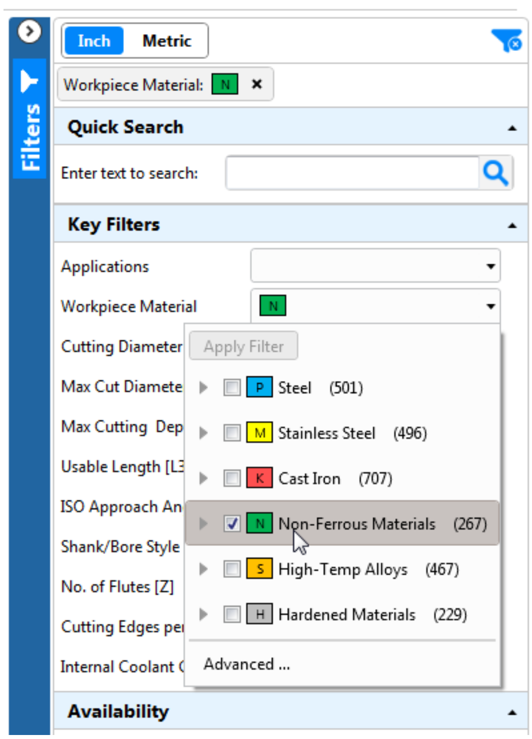
4. Tap or click **Milling Inch Tools**.
5. Tap or click **Face Mills**.

6. Notice the **Filters** window at the right side of the screen.



You can narrow your search by entering information about the face mill you are looking for. Each time you enter new information, tools that do not match your criteria are removed from the list. This helps you narrow your search.

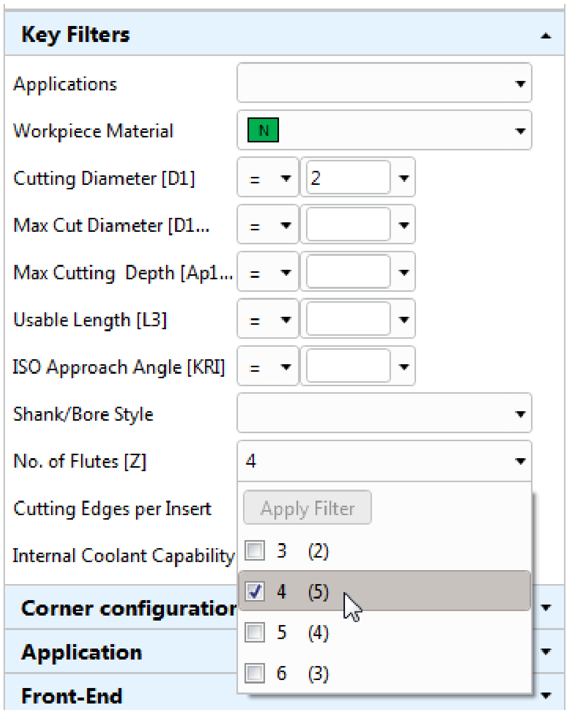
7. Under **Key Filters**, set **Workpiece Material** to **N Non-Ferrous Materials** and click **Apply Filter**.



Workpiece Material is added to the top of the filter list. It can be removed anytime by clicking the small x.

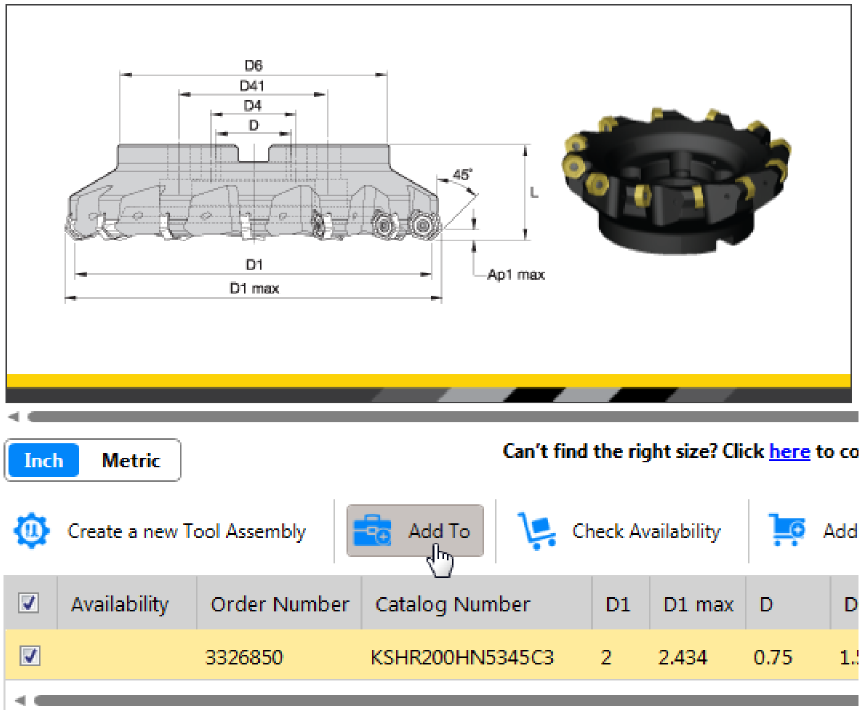
8. Set **Cutting Diameter** to **2** (you will need to scroll down the list of diameter sizes). Now the number of available cutters is reduced.
9. Set **No. of Flutes (Z)** to **4** and click **Apply Filter**.

   Each time you enter new criteria, the number shown in parentheses is updated with the number of tools in the database that match your criteria.

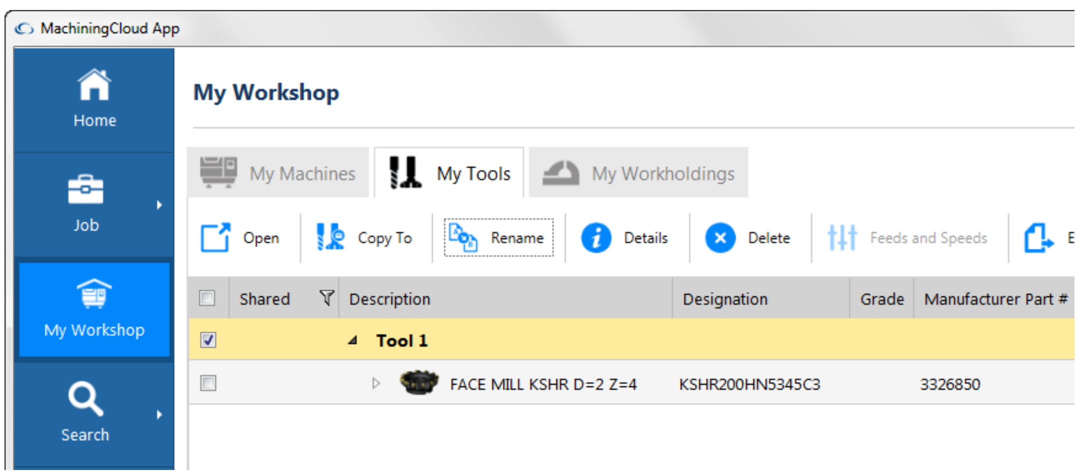


10. You want a face mill with a 45° lead angle, so click or tap **Dodeka™ 45°**.

11. There is one tool that meets your criteria. Select the tool so you can save it to your list of tools.
12. Click **Add To**.



13. Choose **My Tools** to add the tool to your personal list of tools under **My Workshop**.

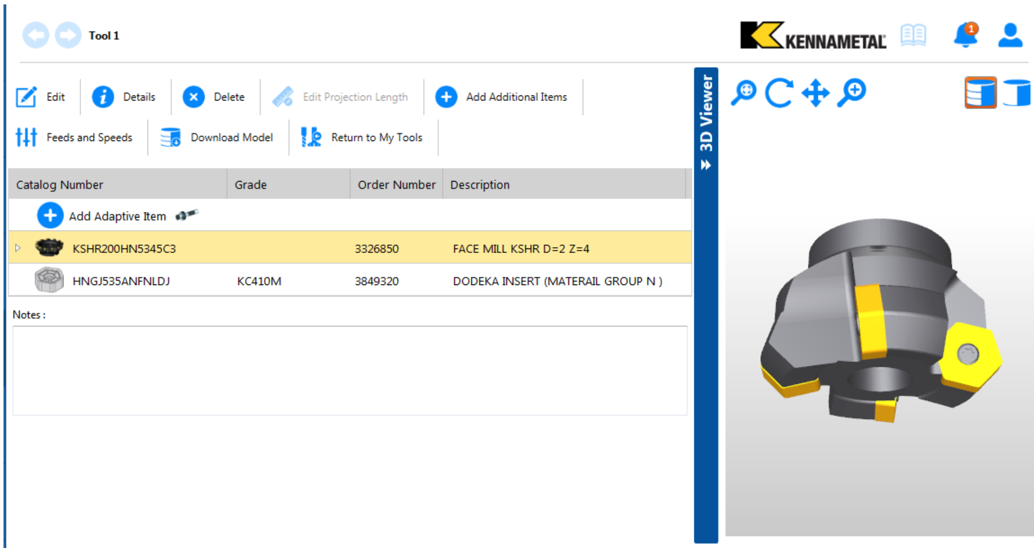


14. Click **Open** to view details about the tool assembly. The shell for the face mill has been added to the assembly. Now you need to add inserts to your face mill.
15. Click **Add Insert**.

    You are taken to a new page where you can search for inserts and add them to your face mill.

16. In the **Filters** list, set **Workpiece Material** to **N Non-Ferrous Material** and click **Apply Filter**.
17. The search is narrowed to an HNGJ (hexagonal) insert. Select the insert in the list.
18. Click **Add to Tool Assembly**.

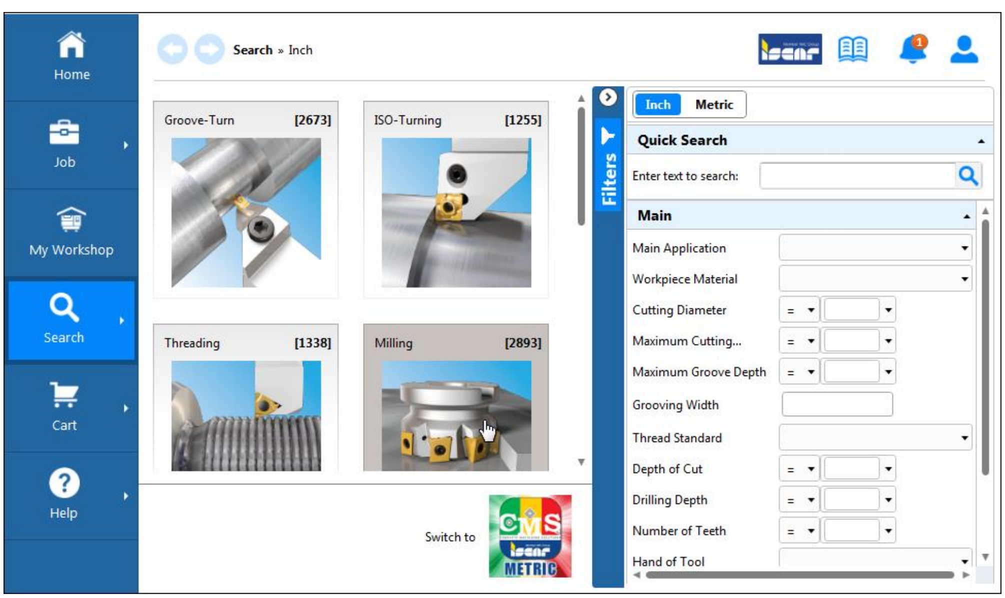
19. You are returned to the Tool Assembly page. At the side of the screen, click **3D Viewer**. Use your mouse or the motion icons on the screen to rotate and zoom the 3-dimensional model of your tool assembly.



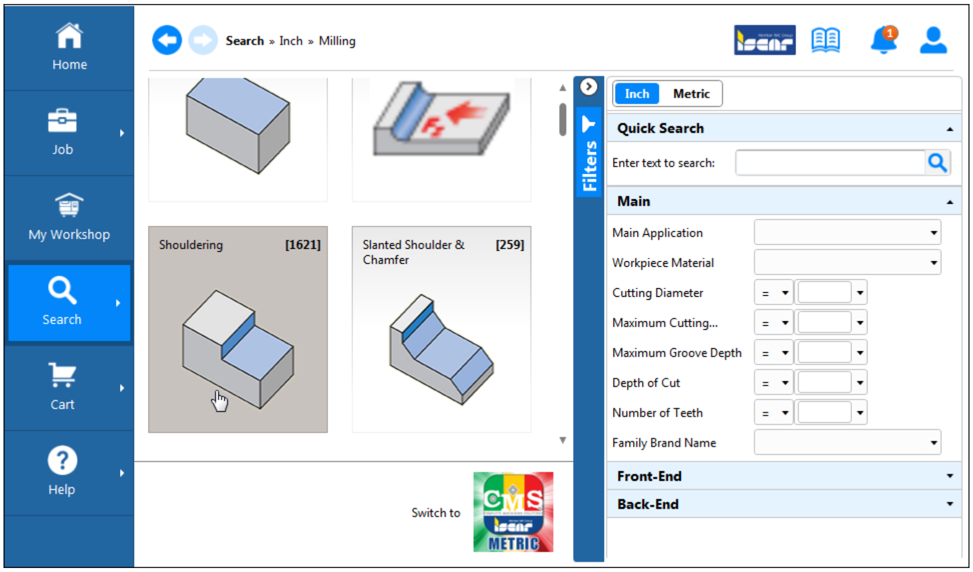
Search for End Mills from a different tool manufacturer

Now you will go back to the Home page so you can select a different cutting tool manufacturer.

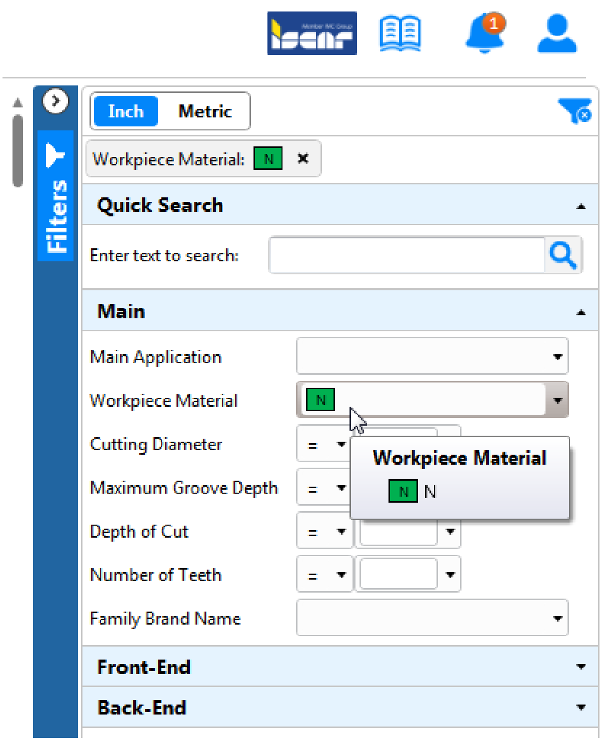
1. At the top left corner of the screen, click **Home**.
2. Click **Select Tool Manufacturer**.
3. Select **Iscar**.
4. When you return to the Home page, click **Search for a tool in the electronic catalog**.
5. Click or tap **Milling**.



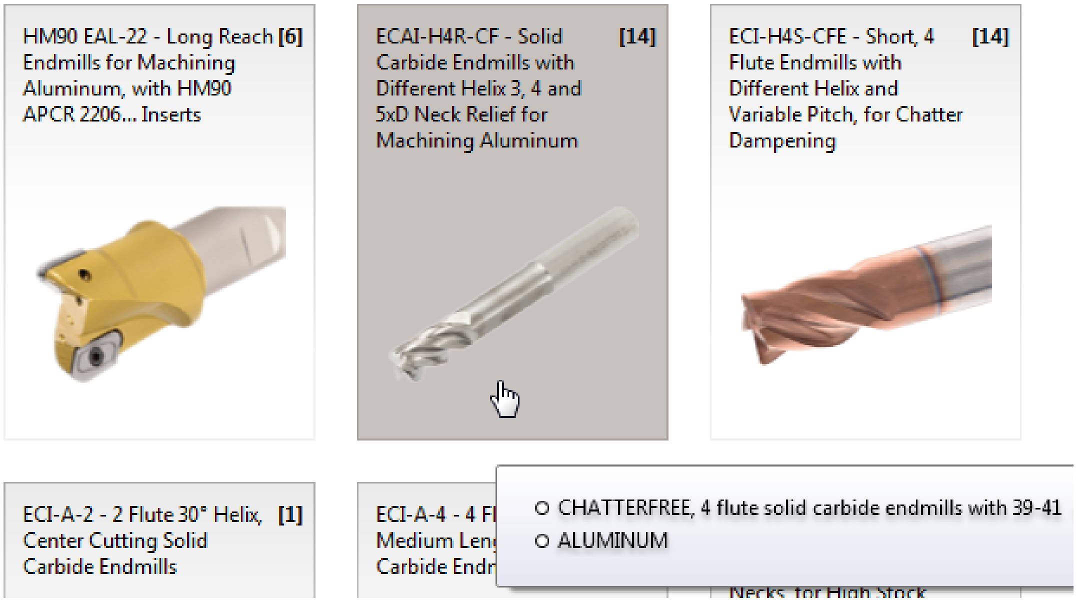
6. You are looking for a contouring tool, so click or tap **Shouldering**.



7. In the Filters list, set **Workpiece Material** to **N** (for Non Ferrous) and click **Apply Filter**.

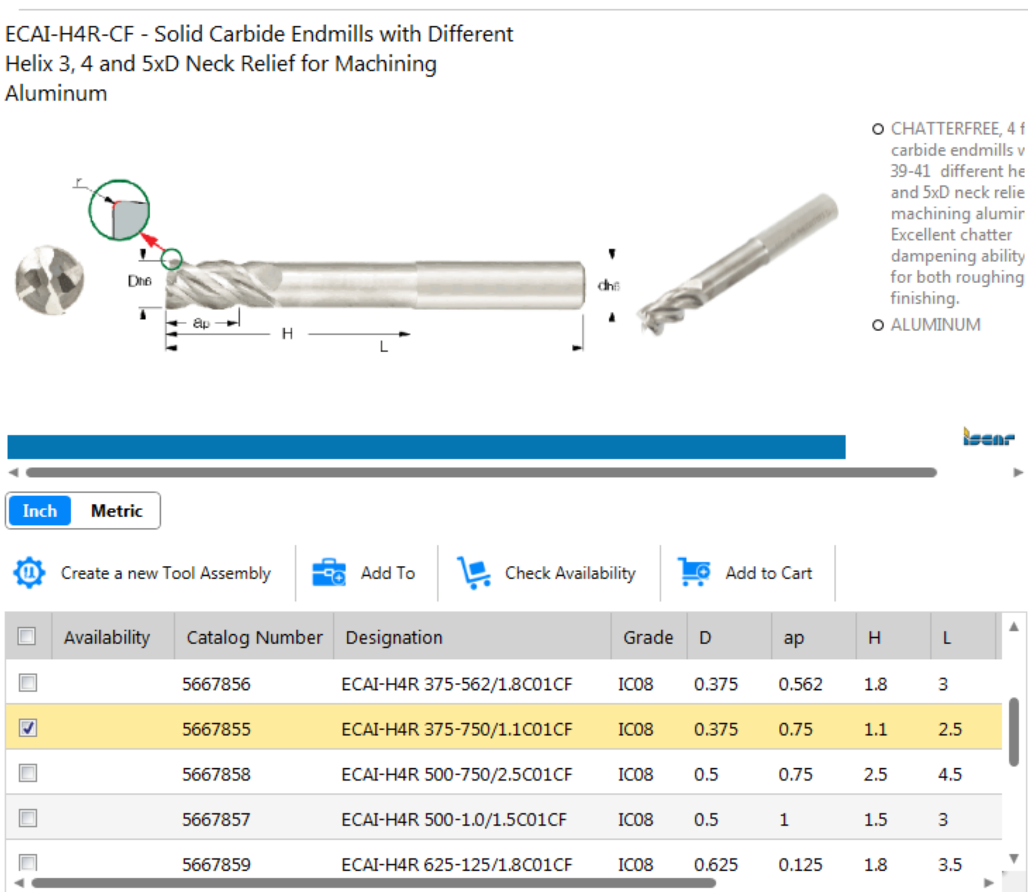


8. Quite a few types of end mills are available. In the main screen, scroll down and select **Solid Carbide Endmills for Machining Aluminum**.



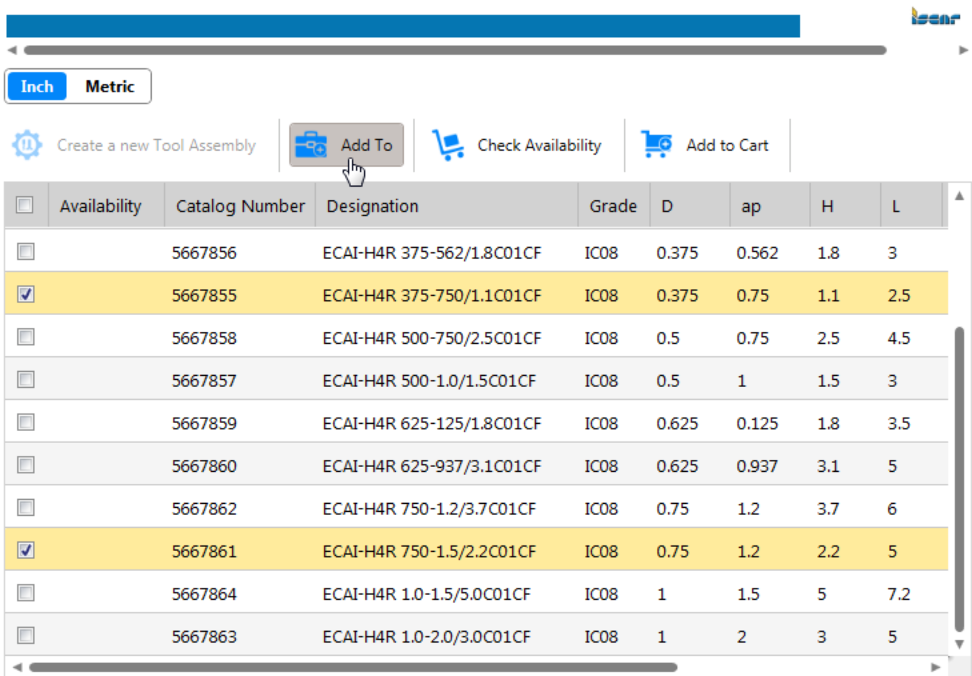
You are looking for two end mills, one with a diameter of .375 and the other with a diameter of .750 inch.

9. Select a tool with a diameter **D** of **0.375** and a cutting length **ap** of **0.75**.



	Availability	Catalog Number	Designation	Grade	D	ap	H	L
☐		5667856	ECAI-H4R 375-562/1.8C01CF	IC08	0.375	0.562	1.8	3
☑		5667855	ECAI-H4R 375-750/1.1C01CF	IC08	0.375	0.75	1.1	2.5
☐		5667858	ECAI-H4R 500-750/2.5C01CF	IC08	0.5	0.75	2.5	4.5
☐		5667857	ECAI-H4R 500-1.0/1.5C01CF	IC08	0.5	1	1.5	3
☐		5667859	ECAI-H4R 625-125/1.8C01CF	IC08	0.625	0.125	1.8	3.5

10. Select a tool with a diameter **D** of **0.75** and a length **L** of **5**.
11. Click **Add To**.



☐	Availability	Catalog Number	Designation	Grade	D	ap	H	L
☐		5667856	ECAI-H4R 375-562/1.8C01CF	IC08	0.375	0.562	1.8	3
☑		5667855	ECAI-H4R 375-750/1.1C01CF	IC08	0.375	0.75	1.1	2.5
☐		5667858	ECAI-H4R 500-750/2.5C01CF	IC08	0.5	0.75	2.5	4.5
☐		5667857	ECAI-H4R 500-1.0/1.5C01CF	IC08	0.5	1	1.5	3
☐		5667859	ECAI-H4R 625-125/1.8C01CF	IC08	0.625	0.125	1.8	3.5
☐		5667860	ECAI-H4R 625-937/3.1C01CF	IC08	0.625	0.937	3.1	5
☐		5667862	ECAI-H4R 750-1.2/3.7C01CF	IC08	0.75	1.2	3.7	6
☑		5667861	ECAI-H4R 750-1.5/2.2C01CF	IC08	0.75	1.2	2.2	5
☐		5667864	ECAI-H4R 1.0-1.5/5.0C01CF	IC08	1	1.5	5	7.2
☐		5667863	ECAI-H4R 1.0-2.0/3.0C01CF	IC08	1	2	3	5

12. Choose **My Tools** to add both tools to your tool list.

Your list of tools is saved in the cloud and you can return to it at any time by clicking My Workshop and then My Tools.

Decision 8: What feeds and speeds should I use?

Specific materials will run better at certain SFMs. Surface Feet Per Minute (SFM) is a combination of the tool diameter and RPM. SFM is a constant, with RPM as a variable based upon tool diameter.

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

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

Feed rate can be defined as Inches Per Minute (IPM) or Inches Per Tooth (IPT).

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

IPM = RPM x Recommended IPT x Number of Flutes

The recommended IPT for aluminum falls in a range from .001 for a 1/8" end mill up to .007 for a 1" cutter.

Speeds and feeds can vary greatly depending on the type to end mill that you use for aluminum.

1. If we assume the SFM for aluminum is 500, then the RPM for the facing operation with a 2" face mill is:

   RPM = 500 x 3.82 / 2.0 = **955 RPM**

2. If we assume the IPT for a 2" cutter is .010", then the IPM for a face mill with 4 inserts is:

   IPM = 955 x .010 x 4 = **38 IPM**

3. Enter the Speed and Feed values for the facing operation in your machining process plan.

PART NO.: 071774		**DATE:**	**MACHINING PROCESS PLAN**			
PART NAME: MILLED PART		**PROGRAMMER:**				
MACHINE: VERTICAL MILL		**MATERIAL:** 6061-T6 ALUMINUM 95 HB				
SEQ	**OPERATION DESCRIPTION**	**TOOL DESCRIPTION**	**STATION**	**SPEED**	**FEED**	**NOTES**
1	Facing	Face mill or shell mill, ∅2", 4 inserts	1	955 rpm	38 ipm	
2	Mill Outer Contour	∅0.750" HSS Solid End Mill, 4 flutes	2			
3	Center drill all holes	#3 HSS Center Drill	3			
4	Drill (2) 23/64 holes	23/64" HSS Drill, 2 flutes	4			
5	Drill (4) 0.25 holes	∅0.25" HSS Drill, 2 flute	5			
6	Ream (2) 0.375 holes	∅0.375" Reamer, Straight Flute, 6	6			
7	Mill Inner Contour	∅0.375" HSS Solid End Mill, 4 flutes	7			
8	Mill Slot	∅0.375" HSS Solid End Mill, 4 flutes	7			

4. Calculate the spindle speed and feed rate for the rest of the tools:
    - 0.750” End Mill, 4 flutes: 500 SFM, .006 IPT
    - #3 Center Drill: 325 SFM, .004 IPT
    - 23/64” Drill, 2 flutes: 325 SFM, .004 IPT
    - 0.375” Reamer, 6 flutes: 100 SFM, .004 IPT
    - 0.250” Drill, 2 flutes: 325 SFM, .004 IPT
    - 0.375” End Mill, 4 flutes: 500 SFM, .003 IPT

PART NO.: 071774		DATE:	MACHINING PROCESS PLAN			
PART NAME: MILLED PART		PROGRAMMER:				
MACHINE: VERTICAL MILL		MATERIAL: 6061-T6 ALUMINUM 95 HB				
SEQ	**OPERATION DESCRIPTION**	**TOOL DESCRIPTION**	**STATION**	**SPEED**	**FEED**	**NOTES**
1	Facing	Face mill or shell mill, ∅2”, 4 inserts	1	955 rpm	38 ipm	
2	Mill Outer Contour	∅0.750” HSS Solid End Mill, 4 flutes	2	2550 rpm	60 ipm	
3	Center drill all holes	#3 HSS Center Drill	3	4965 rpm	40 ipm	
4	Drill (2) 23/64 holes	23/64” HSS Drill, 2 flutes	4	3450 rpm	28 ipm	
5	Drill (4) 0.25 holes	∅0.25” HSS Drill, 2 flute	5	3310 rpm	26 ipm	
6	Ream (2) 0.375 holes	∅0.375” Reamer, Straight Flute, 6	6	1020 rpm	25 ipm	
7	Mill Inner Contour	∅0.375” HSS Solid End Mill, 4 flutes	7	5095 rpm	61 ipm	
8	Mill Slot	∅0.375” HSS Solid End Mill, 4 flutes	7	5095 rpm	61 ipm	

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

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